Small Bear Project Shield

A Senior Project

presented to

the Faculty of the Computer Engineering

California Polytechnic State University, San Luis Obispo

In Partial Fulfillment

of the Requirements for the Degree

Bachelor of Science

by

Janice A. Gelacio

June, 2013

TABLE OF CONTENTS

ABSTRACT

The objective of the project is to design an Arduino prototype shield that resembles the Power Control Board used in the CPE 200, Small Bear Class. By designing a printed circuit board (PCB) with the same characteristics, future students will have the capability to work with a smaller, more organized power board.

Even though I'm familiar with PCB design and this is only a 2-layer board, one of the main challenges faced was learning the new software, EAGLE PCB Software. When the parts list was obtained, each component's mechanical drawings were analyzed before drafting the schematic. The schematic design is critical since it defines the connection between the components on the board. Once the schematic was reviewed and approved, it is followed by careful placement of the component within a specified board outline and routing the connection between each node. Fabrication files were then generated. These were used in ordering the prototype boards.

Another challenge faced was time and cost. The longer it took to complete the design, the less time we had in ordering the boards. Negotiating a good price with a quick turn was necessary in order to ensure a timely arrival of the boards prior to the quarter's end, but at the same time avoid spending a ridiculous amount of money. Shipping cost is another matter in itself. In parallel, the components were ordered online and scheduled for approximately the same arrival time as the boards. One prototype was assembled by soldering each component onto the bare PCB. The prototype shield is then mated with the Arduino UNO board, ready to be tested and programmed.

ACKNOWLEDGMENTS

I would like to thank Dr. Hugh Smith for giving me the opportunity to work on this project. Thank you for your time, help, advice, and guidance. I would also like to thank David Burke who has worked with me throughout this project, providing me the necessary input and feedback in order to obtain the requirements for a successful design. Thank you for being patient with me and answering all my questions.

The CPE 200, Small Bear Class is an elective class taught by Dr. Hugh Smith, assisted by student David Burke. Based on Burke and Tang's report, the objective is to design and build an animated stuffed animal. The course is divided into three sections: "power board construction, frame construction, and software development" [1]. The power board is shown in Figure 1. It was built with a protoboard, screw down headers, various through-hole components, and multiple colors of different lengths of wires. As one can imagine, the task of organizing and managing the wires alone can be quite an obstacle.

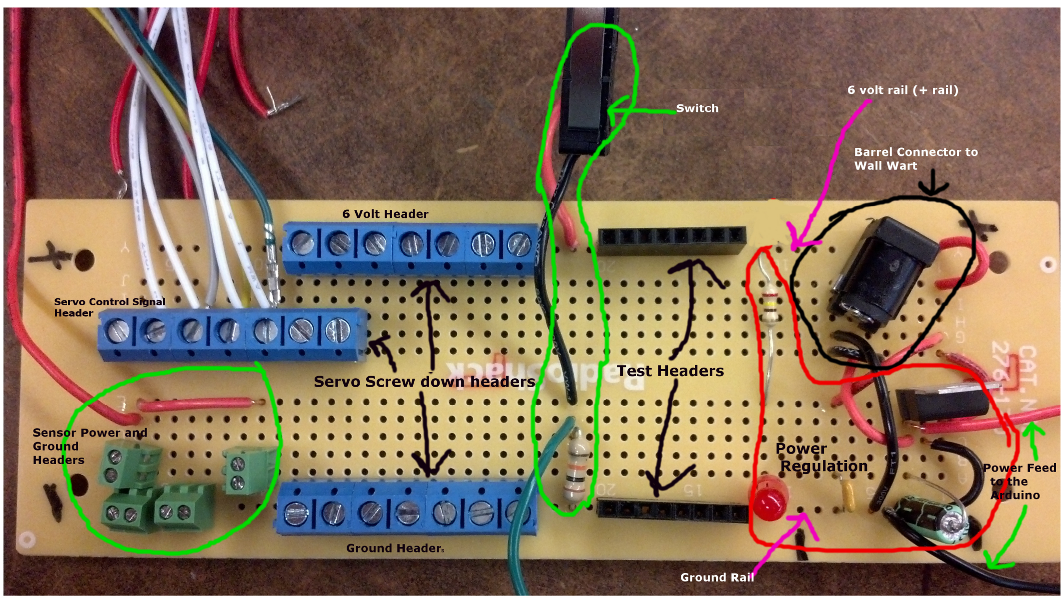


Figure 1. Power Control Board, Version 4 [2]

In the pursuit of improving the content of the class, and with the students in mind, the idea of designing a printed circuit board (PCB) was introduced. The goal is to design a PCB that performs the same function as what is shown in Figure 1. In addition, the new design shall provide an upgraded set-up to avoid unwanted situations, such as the "grounding problem," where the "students [are] either forgetting to synchronize the grounds between the power board

or plugging it in improperly” [1]. Another problem is the “reversal of the power connector…the 9V and 5V lines have been switched, resulting in a fried Arduino board” [1]. Solutions to these problems will eliminate delays to the class and allow the students to focus on the two remaining sections.

2.1 What is a Printed Circuit Board?

According to Wikipedia, “A printed circuit board, or PCB, is used to mechanically support and electrically connect electronic components using conductive pathways, tracks or signal traces etched from copper sheets laminated onto a non-conductive substrate” [3]. In other words, it is a frame that embodies the parts used for a circuit.

For each component, or part, a footprint is laid out with copper pads that allow the parts to be soldered on to, or through, the board. The pad size and shape will vary. The footprint must fit according to the physical form of each part.

There are two types of components, classified as “surface mount” or “through-hole technology” [3]. The main difference is how they are attached to the board. With surface mount technology (SMT), the pads are placed on the top and/or bottom side of the board and components are soldered on the surface. SMT components are smaller. This type allows for a much compact and denser board, but may require automated machines for mounting the components. It’s more ideal for high volume production, but rework for defective assemblies can become a challenge.

With through-hole technology, the pads need a plated through-hole, usually in the center, where the wires or leads are inserted and soldered through the board. Though-hole components tend to be bigger than SMT components. They have “axial leads” that look like arms or “radial leads” that look like legs. This form allows hand-assembly, is likely to be more flexible, and usually easier for rework.

For connections between the components, lines are drawn, or “etched”, and embedded within the board’s body to create a route among each of the pads. This is more practical than

having multiple wires sticking out of the circuit, which can be burdensome, especially if the wires are not in complete contact with the circuit or have the tendency to get disconnected.

Overall, a PCB provides an organized structure for a circuit assembly. While the old fashion protoboards are used for testing and prototyping, PCB prototypes have become an ideal alternative.

2.2 Overview of the EAGLE PCB Design Software

Present time PCB design relies heavily on PCB design software. This is where the bulk of the design process happens. The designer applies the requirements and creates workable files for the manufacture of the board. The EAGLE PCB Design Software is used in this project.

From their website, “the name EAGLE is an acronym, which stands for Easily Applicable Graphical Layout Editor” [4]. It offers schematic layout, board layout, autorouter, and other enhanced features, depending on what version/license is installed on the computer. I’m using the Freeware license. Because it is free software, it contains restrictions [4]:

> The following limitations apply to the EAGLE Light Edition in general:
>
> - The useable board area is limited to 100 x 80 mm (4 x 3.2 inches).
> - Only two signal layers can be used (Top and Bottom).
> - The schematic editor can only create one sheet.
>
> …
>
> - Support is only available via email or through our forum (no fax or phone support).
> - Use is limited to non-profit applications or evaluation purposes.

Nonetheless, the EAGLE Light Edition has capabilities within the scope of this project.

One of the best part of EAGLE Light is the availability of a wide range of library of components. Upon installation, these libraries are readily accessible through the control panel (Figure 2). In other cases, when there is no good match for the component, the library editor can be used to modify existing libraries or even create a new one.

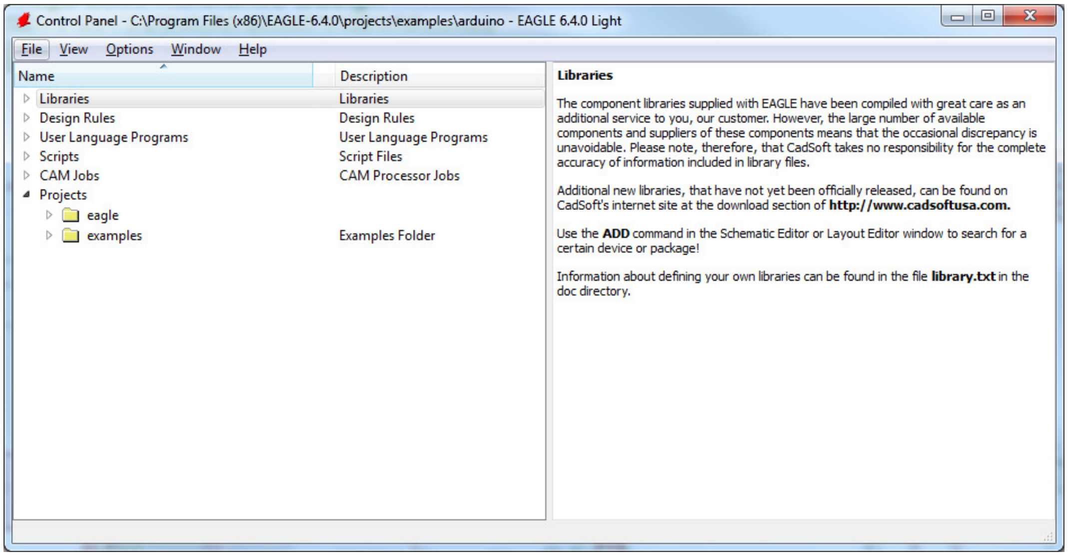


Figure 2. EAGLE Control Panel

Another feature that's worth noting on this software is its ability to link the schematic and board layout. It's almost impossible to work with one without the other. Once a schematic design is created, a board file is also generated with it. Having both of these files open creates consistency between the schematic and PCB.

While a great portion of the time was spent learning how to use EAGLE, I won't go too much into details since there are manuals and tutorials available to help with the software use. EAGLE has a "Training" link on their website where the user can find a good amount of learning guides, webinars, videos, and much more [4].

2.3 What is an Arduino?

Arduino is "an open-source electronics prototyping platform" [5]. It's a "tool" that allows a user, whether it be an "inventor, designer, artist, hobbyist, [student, or] anyone interested" [5], to create projects that can interact with the "physical world" [5]. It is affordable,

can run on various operating systems, simple, and open-source both on software and hardware [5].

There are multiple types of Arduino models available. The Small Bear Project (SBP) uses the Arduino UNO, Rev. 3 (Figure 3), which operates with a “microcontroller board based on the ATmega328.” [6]. The EAGLE schematic (*arduino-uno-Rev3-schematic.pdf*) and reference design files (*arduino-uno-Rev3-reference-design.zip*) are available for download on the website [6]. These files were utilized for reference and for matching its unique board shape and size.

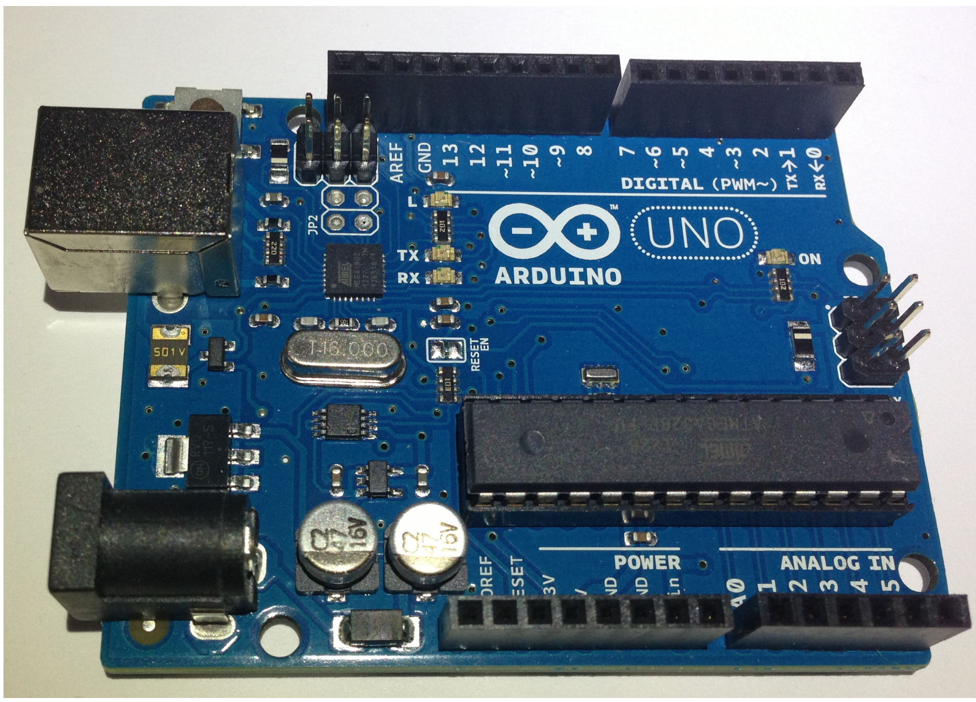


Figure 3. Arduino UNO Board

2.4 What is a Shield?

The purpose of this project is to design a shield for the SBP class. But what exactly is a shield? According to the Arduino website, “shields are boards that can be plugged on top of the

Arduino PCB extending its capabilities" [7]. Basically, it's an "add-on" with a specific function. Figure 4 shows an example, a "prototype shield and a mini breadboard for use with Arduino" [8].

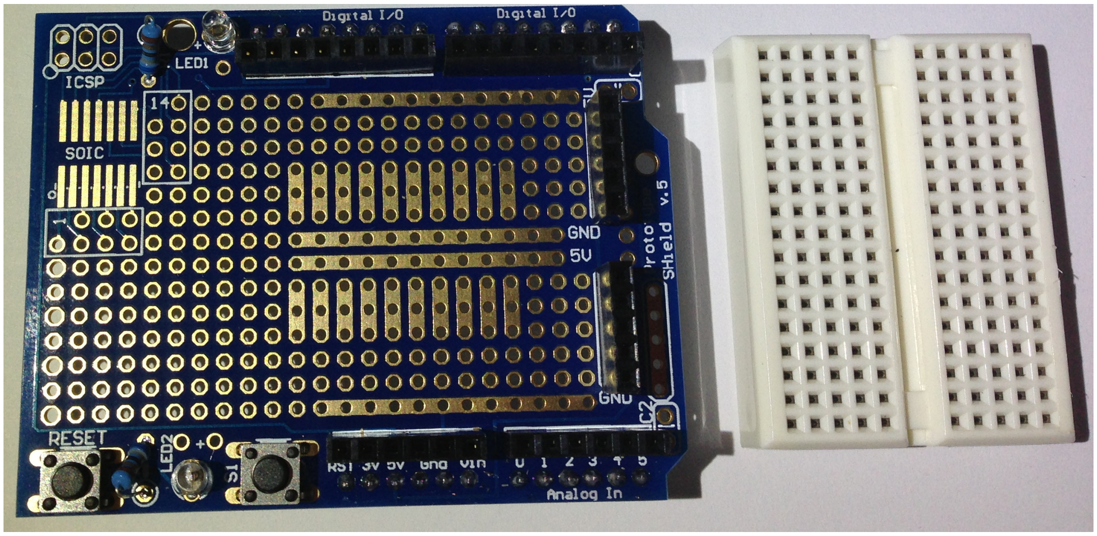


Figure 4. Example: Prototype Shield + Mini Breadboard for Arduino

This shield is generic, where 2 LEDs, 2 resistors, 2 switches, and numerous through-hole pads on a .100" grid are available for soldering components. It also includes a 14-pin SOIC footprint, which connects to two 2x4 header pin pads. Such shields are basic and at $4.50, are quite affordable. There hundreds of shields available at the Arduino Shield's List website [9]. One can browse the database and attempt to find a shield suited for their project. Still, sometimes it's better to just design your own.

The SBP Shield design will be discussed with much detail in the following sections. The project needs this shield in order to provide the current that is not attainable with the Arduino board [1].

3.1 Requirements and Information

In PCB design, before starting, it is very important to first understand what the main requirements are. A good communication between client and designer is always the key. Avoid assumptions. While it can save time, it will lead to nowhere, but delay, faulty design, and/or added cost.

I received preliminary documents from David to help me understand the concept of the Small Bear Class. I purchased an Arduino UNO board from RadioShack to have a sample of the target board, which will also be used in mating with the final assembly.

Once a rough layout of the power board (see Appendix A) was provided, David, Dr. Smith, and I had a conference call to discuss the expectations and restrictions of the component locations and to make sure I understood the information sent.

One restriction was to avoid any components that are direction on top of the Arduino USB connector. Any pins sticking out of the bottom of the SBP shield can potentially short with the USB's metal frame. Also, in order to prevent students from any confusion on the servo lines, S1, S2, and S3 will be distinguished by female headers for S1 & S2, and male headers for S3. This is to help with polarity and avoid plugging in the connectors incorrectly. When the parts list was sent, I was able to move forward with the design.

3.2. Bill of Materials

The parts list I received included the name/description of the part, link to the online resource, and some additional notes from David, which mentions some uncertainties on what component will be used or substituted. Though it was not, yet, complete, it was workable

enough to start gathering datasheets. Throughout the project, David provided updates to finalize the list. When we realized one of the resistors was too big for the board, it was replaced with a similar resistor value, but with a smaller body size.

I used the parts list to create a Bill of Materials (BOM) (Appendix B) in order to keep me on track with the design. It was also a guide to sort out the components.

During the final process, the BOM is used to purchase the components. This document also helps with traceability of assembled components, should there be any reason to troubleshoot the shield. The cost has been added as a reference for future builds.

3.3 Component Analysis

It is important to understand the physical dimensions of each component, especially during the board layout. If the component is too large, it may result in little to no space left for routing. Using the datasheets gathered (Appendix C), the components' mechanical information is analyzed.

For through-hole components, the key items to check is the body size and the hole size. It would be difficult to solder a through-hole component if the leads do not fit in the hole. This could lead to component rework, a search for a substitute, or even a scrapped board.

3.4 Schematic Design

In EAGLE, the designer must first add components in the schematic file by clicking the add component icon (Figure 5). The software does not allow placement on the board layout without the schematic. Everything starts at the schematic level, as it should be. Otherwise, you'd be designing blindly in the board layout.

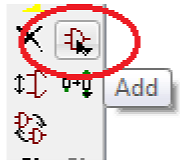


Figure 5. Schematic – Add component

3.4.1 Library Component Selection

A window will appear, listing all the component libraries available for use. Search or browse through to find the component that's best suited for each item listed on the BOM. Be aware that some of these library components may appear similar to the component on the list, but in actuality, they are not. Make sure to check the actual PCB footprint and cross reference it to the datasheet mechanical dimensions to ensure the footprint is correct.

For example, as shown in Figure 6, there is the possibility of utilizing the existing component package "B3F-10XX", found in the Switch-Omron > 10-XX library for the button switches B1 and B2.

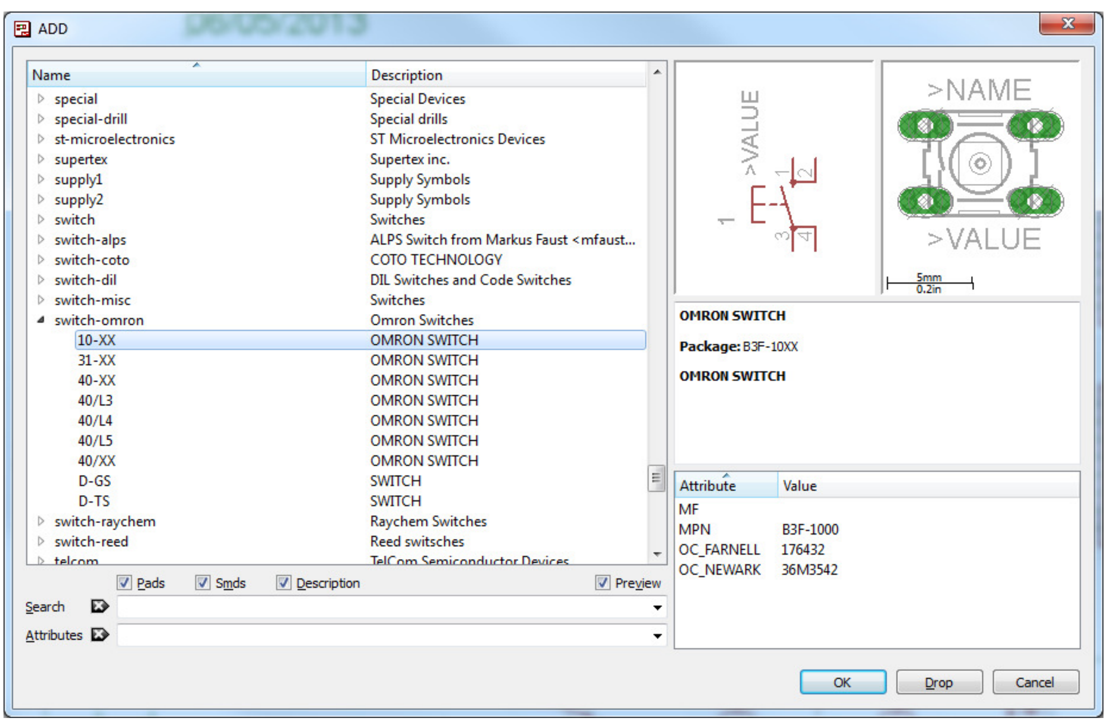


Figure 6. Library selection window – Switch

The top right hand side of the add library window shows both the schematic symbol and the footprint. This gives us a visual of what type of component is listed. When you click ok, the schematic symbol becomes active on the schematic sheet. Left clicking the mouse places the component on the schematic sheet, along with the footprint on the board layout file. Note that with EAGLE, it is not allowed to delete the footprint on the board file. This is the software's way of keeping the component consistent on the schematic and board file.

Switching to the board file, we see the component footprint. Using the dimension tool, we can easily make a quick check whether this is truly the footprint we want to use (Figure 7). Comparing this information to the datasheet for B1 & B2 (Figure 8), we can rely that this is a usable library component.

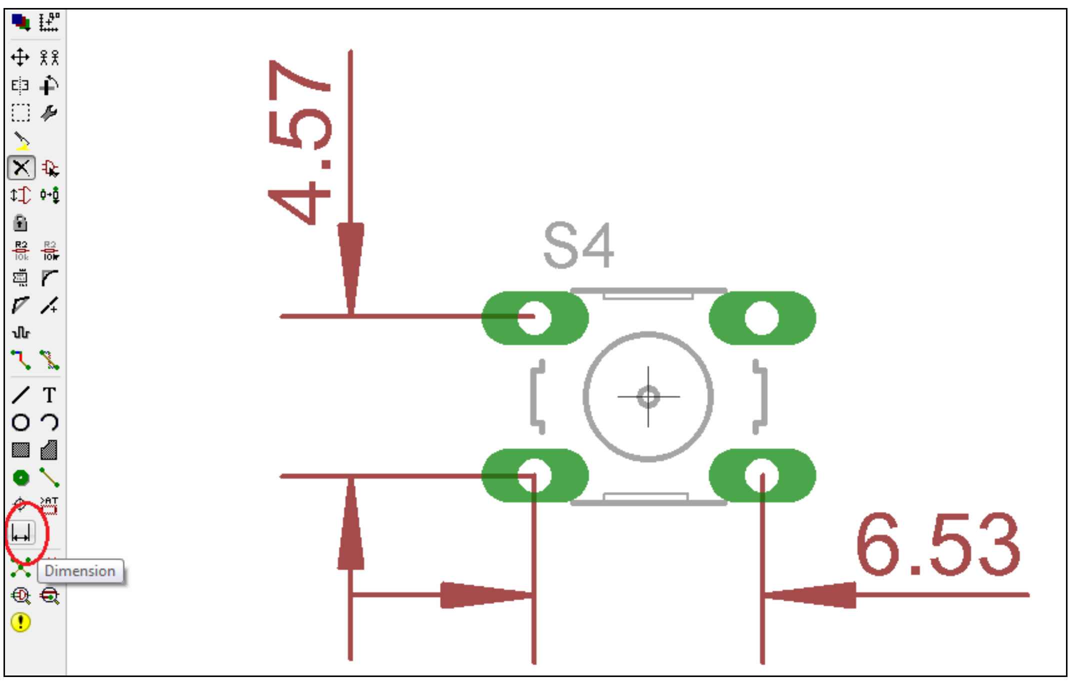


Figure 7. Footprint quick check with dimension tool

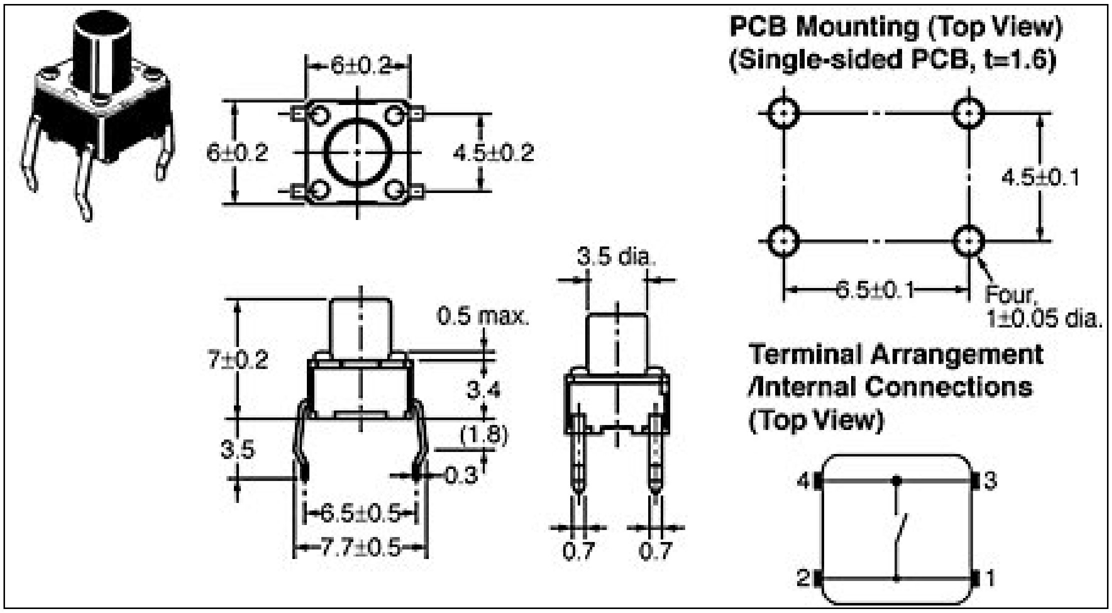


Figure 8. Mechanical Dimensions for B1 & B2

The rest of the components were laid out per the Table 1 below. This summarizes the components use and which library they came from.

Item	Reference Designator	Library	Name	Description	Other info
1	B1,B2	SWITCH-OMRON	10-XX	Package: B3F-10XX Omron Switch	none
2	C1	RCL > CPOL-US	CPOL-USE1.8-4	Package: E1,8-4	grid 1.8 mm, diameter 4 mm
3	C2	RCL > C-US	C-US025-040X050	Package: C025-040X050	grid 2.5 mm, outline 4 x 5 mm
4	J3, JP3 S1, S2	PINHEAD	PINHD-1X6	Package: 1X06, Pin Header	Modified pads: was oblong, is round
5	J1, JP1 J4, JP4	PINHEAD	PINHD-1X8	Package: 1X08, Pin Header	Modified pads: was oblong, is round
6	J2, JP2	PINHEAD	PINHD-1X10	Package: 1X10, Pin Header	Modified pads: was oblong, is round
7	LED1, LED2, LED3	LED > LED	LED5MM	LED5MM	5 mm, round
8	MOLEX	CON-MOLEX	22-23-2041	Package: 22-23-2041	symbol, but modified pcb fooprint according to datasheet
9	R1	RCL > R-US	R-US_0207/10	Package: 0207/10	type 0207, grid 10 mm
10	R2, R3	RCL > R-US	R-US_0204/7	Package: 0204/7	type 0204, grid 7.5 mm
11	RAIL_OFF	SWITCH	M9040P	Package: M9040P	only uses 2 pads, pin 1 no connect
12	S3	PINHEAD	PINHD-1X6	Package: 1X06, Pin Header	Modified pads: was oblong, is round
13	SEL_POW	SWITCH	TOGGLE SWITCH	Package: M9040P	
14	VOLT_REG	V-REG	78MXXL	Package: 78MXXL VOLTAGE REGULATOR	since this is stand up, removed the rest of the silk
15	X1	CON-JACK	DCJ0202	Package: DCJ0202 DC POWER JACK	modified location of pads

Table 1. Library components used and modified

While there was no need to make modifications to any of the schematic symbols, some component footprints required some changes to fit the physical part. An example will be given in Section 3.5 with details.

3.4.2 Schematic Layout

When all the library symbols have been attained, schematic design can proceed. This process went through multiple revisions (rev. P1 through P8). Although that may seem a lot, its key function is to make sure the connections are correct. An incorrect line connection can have a massive effect on the final board design outcome. For example, in the voltage regulator datasheet, it is specified in Figure 9 that the .1uF is connected at Vo.

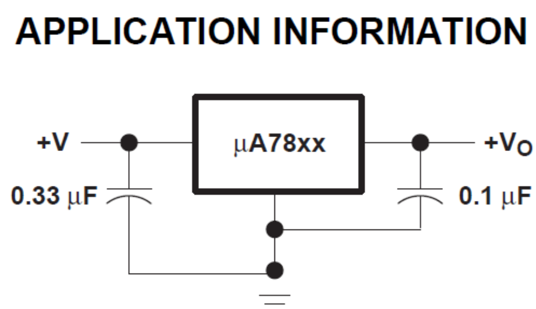


Figure 9. Voltage regulator application information

However, somehow, C1 and C2 were swapped on the schematic, and the capacitors were laid out incorrectly. Fortunately, during a check process, the fault was corrected prior to board manufacture.

The schematic is sent to David and Dr. Smith for final approval. See Appendix D for the final schematic.

3.5 PCB Design

When the schematic layout has been approved, we proceeded on to the PCB layout. Since the PCB footprint was already chosen during the schematic library component selection, we can proceed with placement of the components.

3.5.1 PCB Component Placement

Each footprint is placed according to Appendix A. During this process, however, placement must also be executed with respect to the airwires, or ratsnest (Figure 10). These ratsnests are a guide to show which pads should be connected. By placing the components along or near its connected pad, it makes the designer's routing much easier. Rotating a footprint can mean drawing a simple straight trace, as oppose to drawing around the component.

The less crossing of ratsnests, the better the routing. Regardless, there will be situations when these ratsnest will still cross paths. In these occasions, we have no choice but to route through other layers of the board and maneuver the way.

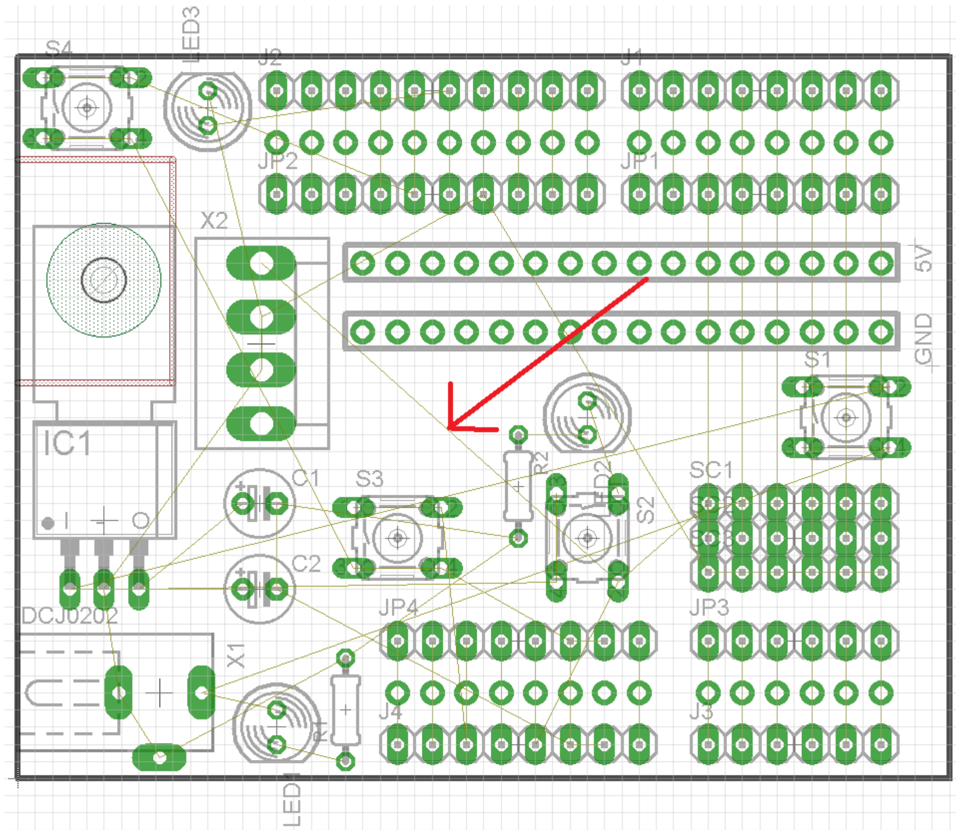


Figure 10. Preliminary placement with Ratsnest

Placement approval is also a necessary step prior to routing. Routing before a client approves a placement could result in wasted hours spent on the design if the client requests moving the components. During this step, Dr. Smith requested to move the X2 (Molex) connector closer to the edge of the board. With the final placement, X2 has been moved next to X1.

3.5.2 Footprint Modification

There were a few footprints from the library that did not conform to the physical dimensions of the component. The schematic symbol was used, but the footprint required some editing. For example, the 1x6 HEADER footprints used for S1, S2, and S3 originally had oblong

pads (Figure 11). The pads were changed to round shape, in order to avoid shorting when the rows are placed side by side at a .100” grid.

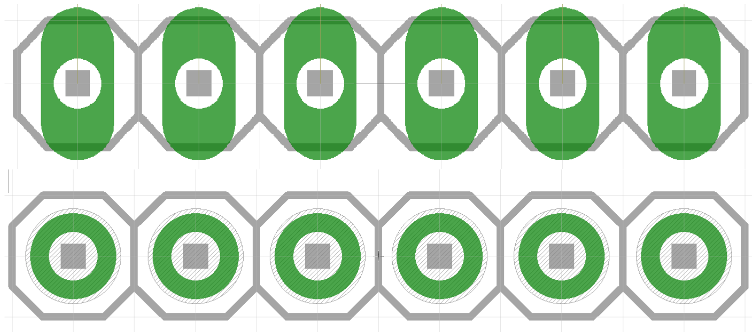


Figure 11. Component footprint modification

The MOLEX footprint also did not match. The footprint was modified to almost a new footprint since the distance between each pin, the pitch, was different.

I started with a connector Molex footprint, using the KK-156-4 library package from the CON-MOLEX library. Compared to the MOLEX’s mechanical drawing’s recommended PCB hole layout, the pitch is 5.08 mm [.200”]. The existing footprint only had roughly 4.12 mm [.160”] approximately .040” off (Figure 12).

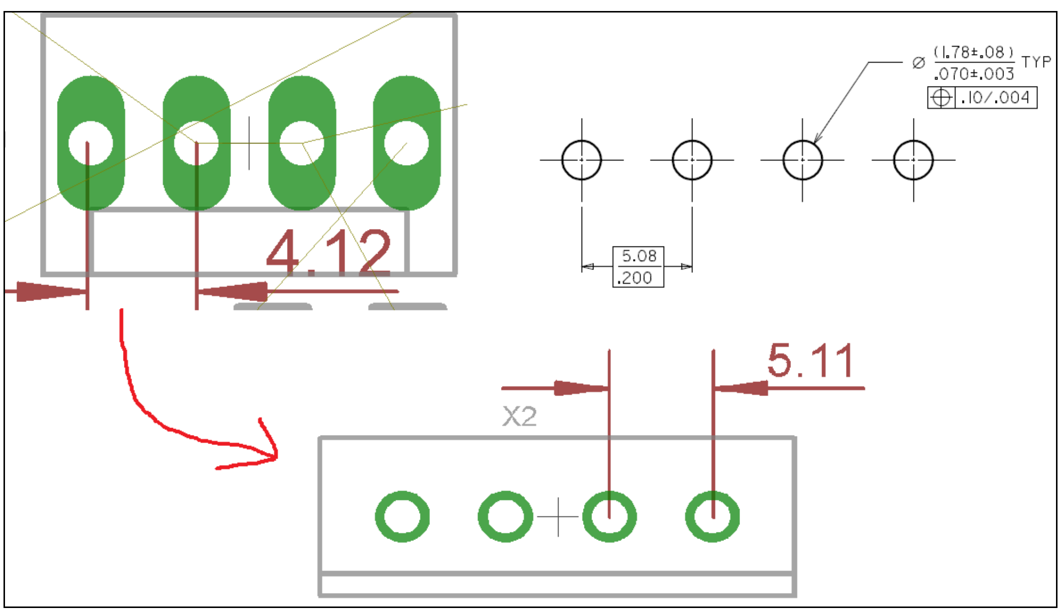


Figure 12. MOLEX footprint modified

I increased the pitch and changed the component body outline to help determine the spaced it will occupy on the board. The outline is essential, especially for large component s like these.

3.5.3 Routing

Routing is drawing the connection in the board layout to create a path between the pads. With an effective schematic, well thought out placement, and ratsnest to give guidance, the routing process did not take too long. Traces were laid out at .010" thick, which is a good line width for most boardshops. The smaller the trace, the more cost spent on the board. This applies also to drill holes. Boardshops have certain capabilities and if the board has features beyond their capabilities, we either need to find a different boardshop, or make changes to the board. This board has no limitations because it's not too dense.

3.6 PCB Clean-up and Design Verification

Due to the many components laid out on this board, it is best to have a good silkscreen (the white markings on the board) to label the pins and components. Some reference designators were changed for clarity, e.g. X2 to MOLEX and switches to RAIL_OFF and SEL_POW.

One of the most important final steps is to run EAGLE's Design Rule Check (DRC). This is an essential check that should never be omitted. In the PCB layout, under Tools > DRC, the window opens up where the designer can specify the rules that the board must meet, like minimum clearance between copper. DRC will also check for the possibility of any shorts.

Should there be any rules not met, a summary is provided, in which the designer can have the chance to make corrections.

3.7 Gerber File Generation

When the board passed DRC, the gerber files are generated. Gerbers are the files, or the artwork, that the boardshop uses to manufacture the boards. The SBP Shield board has 8 gerber files:

.L1 = Top Layer + Pads + Vias
.L1S = Dimension (board outline) + tPlace + tNames (silk)
.L2 = Bottom Layer + Pads + Vias
.L2S = Dimension (board outline) + bPlace + bNames (silk)
.L1M = tStop (mask layer)
.L2M = bStop (mask layer)
.NCD = Drills + Holes

It is good practice to open the gerber files in a gerber file viewer to review the artwork. Doing so can help you see any discrepancy on the design, if any. I used gerbv, "a free/open source gerber viewer" [10]. From this, the gerber plots were printed out to a pdf for review and approval.

3.8 Review and Approval

A final review to the schematic and gerber data is reviewed. This is the time to make any necessary changes to the board. Once everything looks good, the board is considered ready for manufacture.

3.9 Fabrication Drawing

A fabrication drawing is good to have with the gerber files since it has notes for the boardshop on what material to use and summarizes the board stack-up and drill. Unfortunately, in an effort to purchase the boards so we can receive it on time, this drawing was not created. It will be created for the next round of fabrication.

3.10 Final Gerbers and Fabrication Package

The gerber files were compressed in a .zip file. The Rev. A files do not have a fab drawing, yet, but the criteria was either specified in the board house website, email, or phone call. The zip file is sent to board houses for request for quotes (RFQ's). If the boardhouse does not have any inquiries or request for any changes to the board, the same files are used for manufacture. Appendix E shows the gerber plots.

As an added precaution, we can request for the working gerber files from the boardhouse, prior to their manufacture. These working files are our gerber files, transferred to their system. This gives an extra step in preventing any possible oversight. These "work files" are what one would call "what you see is what you get" type of files. Boardshops will not change the artwork we submit to them, so the files should be the same.

4.1 Board House Selection

Once the design is finished, the race to find an affordable board house with a quick turn begins. The EAGLE website has a good list of them, under Services > Board Houses [11]. Due to time sensitivity, I chose board houses that are either in California or in the near states. See Appendix F for the summary.

The most ironic part about this is that the board house that quoted me the highest ($1,020.80 for a 3-day turn), ended up lower than what I expected with the most affordable $150 for 5 boards at a 3-day turn. The only caveat to it was the $101.50 shipping which is necessary for the overnight shipment. Even with the shipping cost, the confirmation to deliver within my target date was enough for me to place the order with Advanced Assembly. Chet Williams was very helpful. He really worked hard in getting me the best price possible. Sometimes, it's beneficial to explain to the contact sales person what these prototype boards are used for and if there is a forecast for future board orders. No commitments were made to order future boards from Advanced Assembly, dealing with the same vendor will help with the re-ordering process.

Since there is no fabrication drawing, the board was manufactured in the most basic way to save on cost. Because it's a prototype, I was not too concerned with the other factors, as long as it's within the scope of the form, fit, and function of the design.

The board was built per the specifications: .062" thickness, FR4 material, 1 oz. copper, HASL plating, green mask LPI, and silkscreen on Top layer.

4.2 Purchasing Components

Having the BOM handy made the component purchase much easier. There is one deviation from the list that David sent me, to what was ordered. I only ordered parts from two vendors: Mouser Electronics and Sparkfun Electronics. I was able to get most of the components from these vendors, except for the R2 and R3, which was originally referenced with the Digikey website. Rather than spend an overpriced shipment for 2 pcs of resistor, I purchased an equivalent 150 ohm resistor at Mouser. It seemed like the more practical way to go.

When purchasing these items for future builds, make sure it's given enough time to ship. The Mouser order almost didn't arrive on time because there was a delay at the shipping center. Fortunately, I received the parts a day prior to submission, in which I was still able to assemble the prototype.

Save on cost, order ahead of time. Shipping cost can be ridiculous when turn time becomes critical.

Upon receipt of the bare boards, I first did a visual inspection to make sure it looks like the gerber files created, which it did.

The next test is what I call the bare board test in which I tested for continuity. This is to make sure the connections are present and there is no broken trace. The worst thing that can happen is to build an assembly, only to find out the board is faulty. This is not only wasteful on time, but on parts as well.

Using a multimeter (Figure 13), each pad is probed and tested. The connection were correct, except for one. It's not related to the component, but rather the power and ground rails. There was no connection on those pads, so I must not have tied it to the ground and power. Checking the gerber plots, it's confirmed that they were omitted. This needs to be corrected for the next board order. Regardless, this oversight does not affect the functionality of the board and is still usable. It can also be amended by soldering a blue wire from a GND pin to one of the pins on the rail. The same goes for the power pin to the power rail.

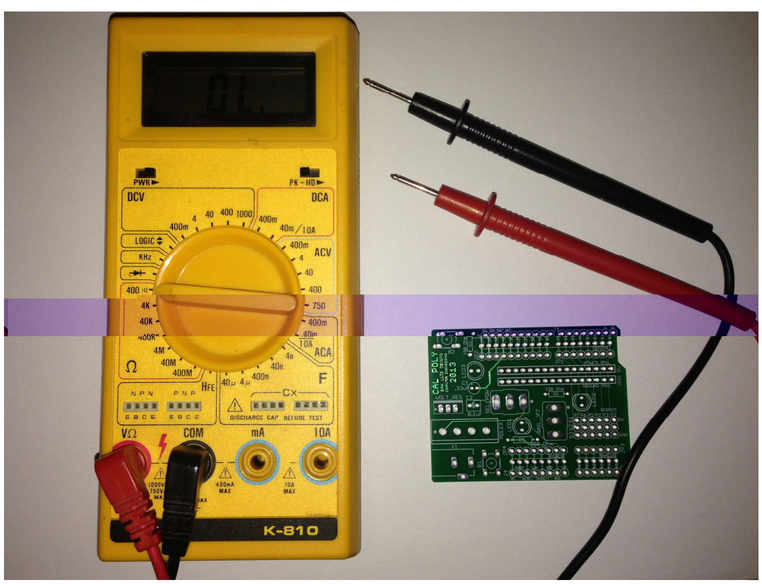


Figure 13. Bare Board Test

The shield is assembled by hand using a standard soldering iron (Figure 14). The components were first fitted prior to soldering.

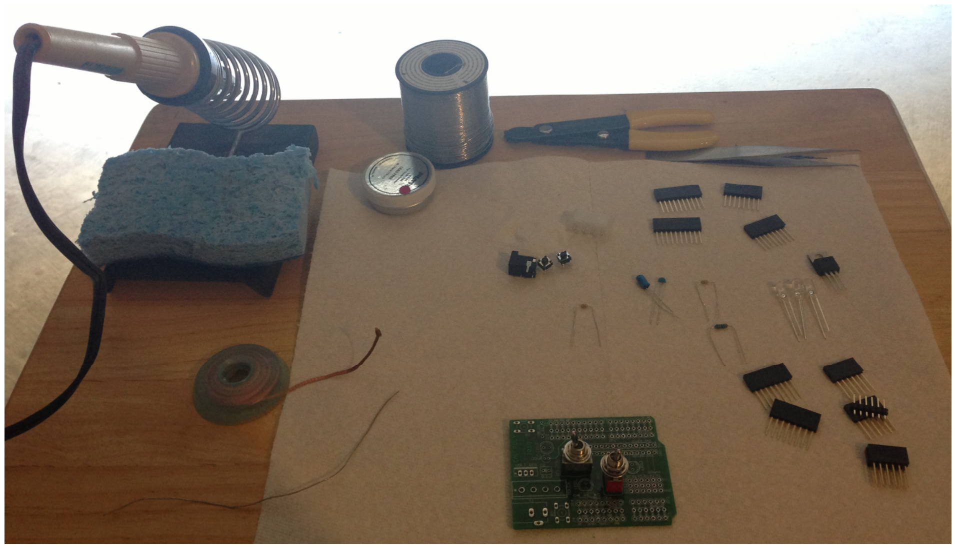


Figure 14. Assembly

There are two incorrect footprints on the board. The RAIL_OFF and SEL_POW had holes that were too small for the leads. Figure 15 shows the correct hole size for the switches.

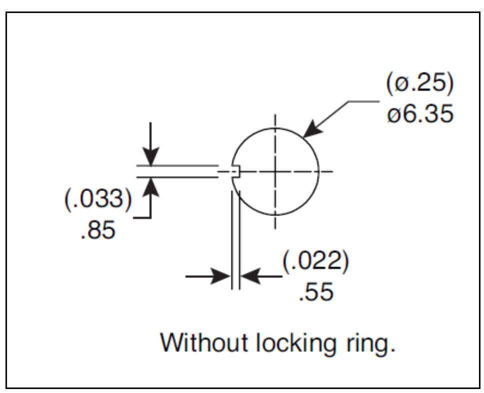


Figure 15. Switch hole size

Unfortunately, I grabbed the wrong information from the NKK Switch datasheet (Figure 16).

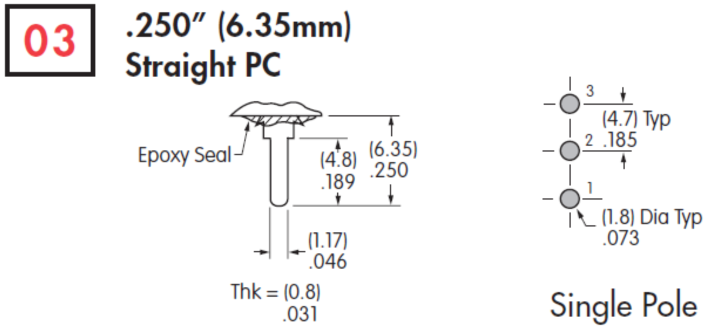


Figure 16. NKK Incorrect Switch Info

This needs to be corrected for the next build. Fortunately, the leads of the switches can be crimped, so that I was still able to fit the leads through the hole for soldering.

The shield mated well with the Arduino board. The clearance about the USB was a good decision since it prevented any pins sticking down from hitting the USB casing. The Arduino power jack did have a slight conflict with the X1 pins directly above it. This can be amended by cutting the leads of X1 as close to the board as possible to avoid any contact below it.

After submission of the prototype to Dr. Smith and David, we found some changes are necessary to improve the spacing on the SBP Shield. See Table 2 for the summary of changes and notes for Revision B release.

The two flaws on the board were unfortunate, but not disastrous. These are also listed in Table 2. I also noticed that the SBP shield did not have any mounting holes. This was not necessary on the SBP Shield since the Arduino board already has holes, which can be used for mounting.

Change #	Description
1	Correct SEL_POW and RAIL_OFF footprints. Need bigger holes Ø.250”
2	Tie power rail to +5V and ground rail to GND
3	Increase annular ring pads on MOLEX
4	Servo headers need changing – TBD
5	Replace female headers on J1, J2, J3, and J4 with male headers. Plastic side to be on bottom side, solder on top of PCB. Same male header pins for S3 may possibly be used.
6	Move RAIL_OFF switch further away from the servo headers
7	Determine cost for lower switches (lower profile). Possible slider switch? -TBD
8	Change silkscreens: a) B1 to RESET b) LED1 to Pin13 c) LED2 to Pwr_Rail d) LED3 to Brd_Pwr

Table 2. Summary of Changes for Revision B

David and I tested the SBP shield in the lab. While the results of its full functionality will have to wait until David can test it thoroughly with the Arduino board, the shield appeared to be running appropriately. The LEDs displayed the expected results, based on when power is plugged in through X1 and MOLEX. LED1 turned on when X1 was plugged in. LED2 turned on and off when RAIL_OFF was switched on and off. And LED3's blinking which, according to David, was generated from the pin 13, with the Arduino's default blinking program.

In conclusion, this was a great project to be involved in. I thought that because I had experience with PCB design, this would be fairly simple. However, because I was removed from my comfort zone and had to deal with a new software, it turned out to be challenging and educational. It wasn't what I was used to, but I learned a lot, especially beyond the process of design. Purchasing and assembly allowed me to open my eyes to the difficult tasks in getting the board fabricated and assembled. Planning is very important because when time became a factor, the cost can hurt the project's overall budget.

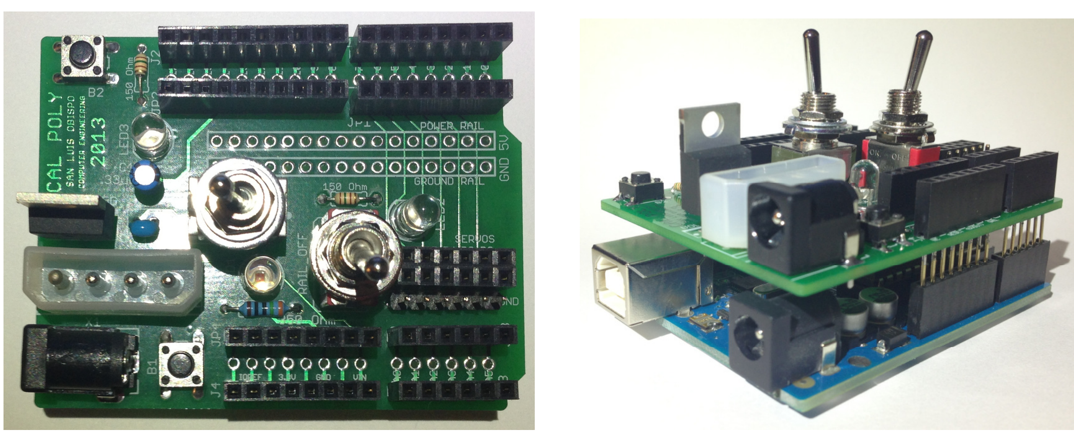


Figure 17. Images of the finished assembly

[1] Burke, David, and Tang, Devin. *CPE 200: The small bear project*. California Polytechnic State University, San Luis Obispo. Winter Quarter, 2013. Report.

[2] Burke, David, and Tang, Devin. *power_control_board.v4.jpg*. California Polytechnic State University, San Luis Obispo. Winter Quarter, 2013. Photograph.

[3] Printed Circuit Board. Retrieved on June 5, 2013, from Wikipedia: http://en.wikipedia.org/wiki/Printed_circuit_board

[4] EAGLE PCB Design Software, http://www.cadsoftusa.com

[5] Arduino, http://www.arduino.cc/

[6] Arduino UNO, http://arduino.cc/en/Main/ArduinoBoardUno

[7] Shields, http://arduino.cc/en/Main/ArduinoShields

[8] DealExtreme, http://dx.com/p/arduino-prototype-shield-mini-breadboard-118040

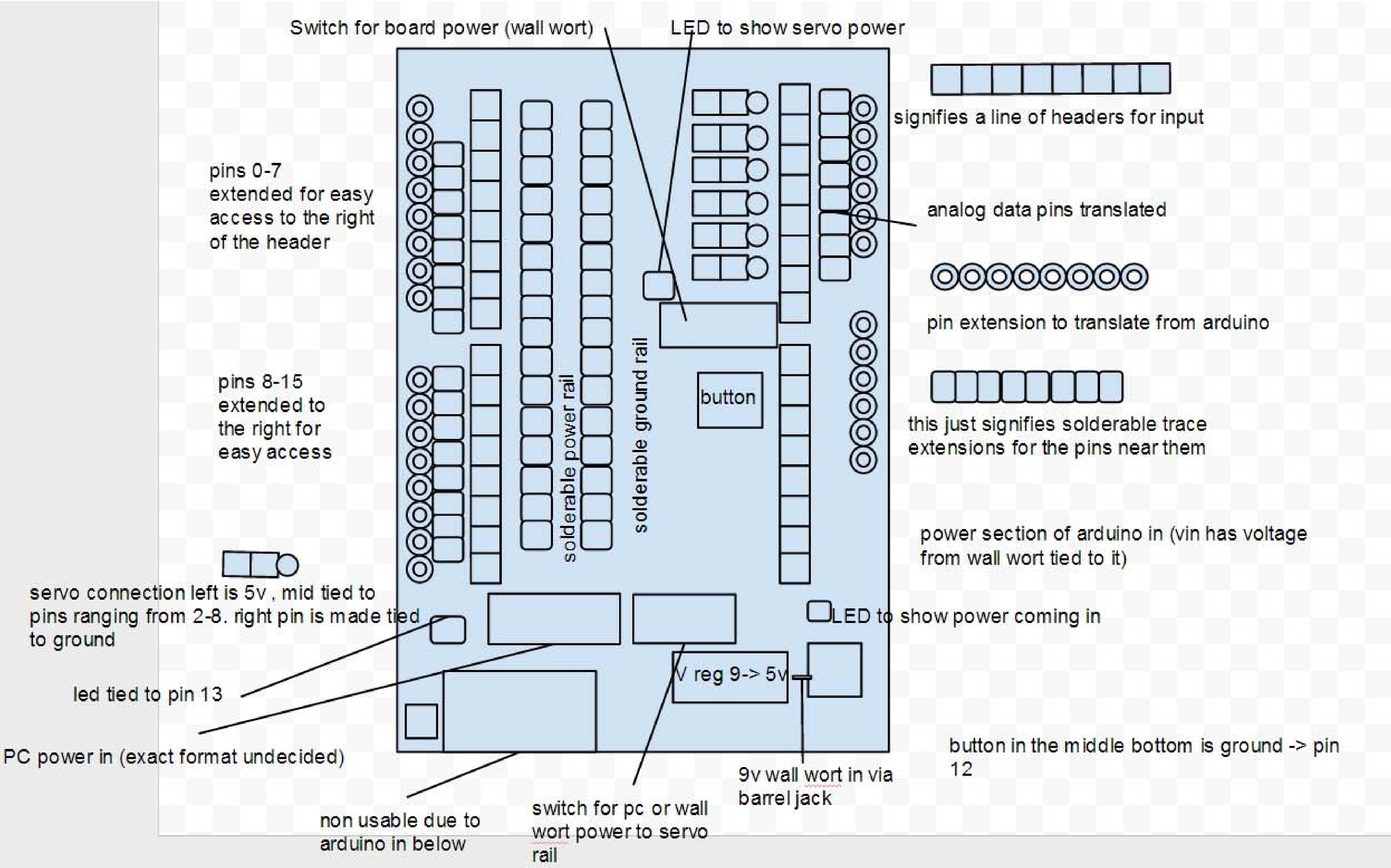
[9] Arduino Shield List, http://shieldlist.org/

[10] gerbv, http://gerbv.sourceforge.net

[11] Board Houses, http://www.cadsoftusa.com/services/board-houses/?language=en

APPENDICES

APPENDIX A: Rough layout of power board

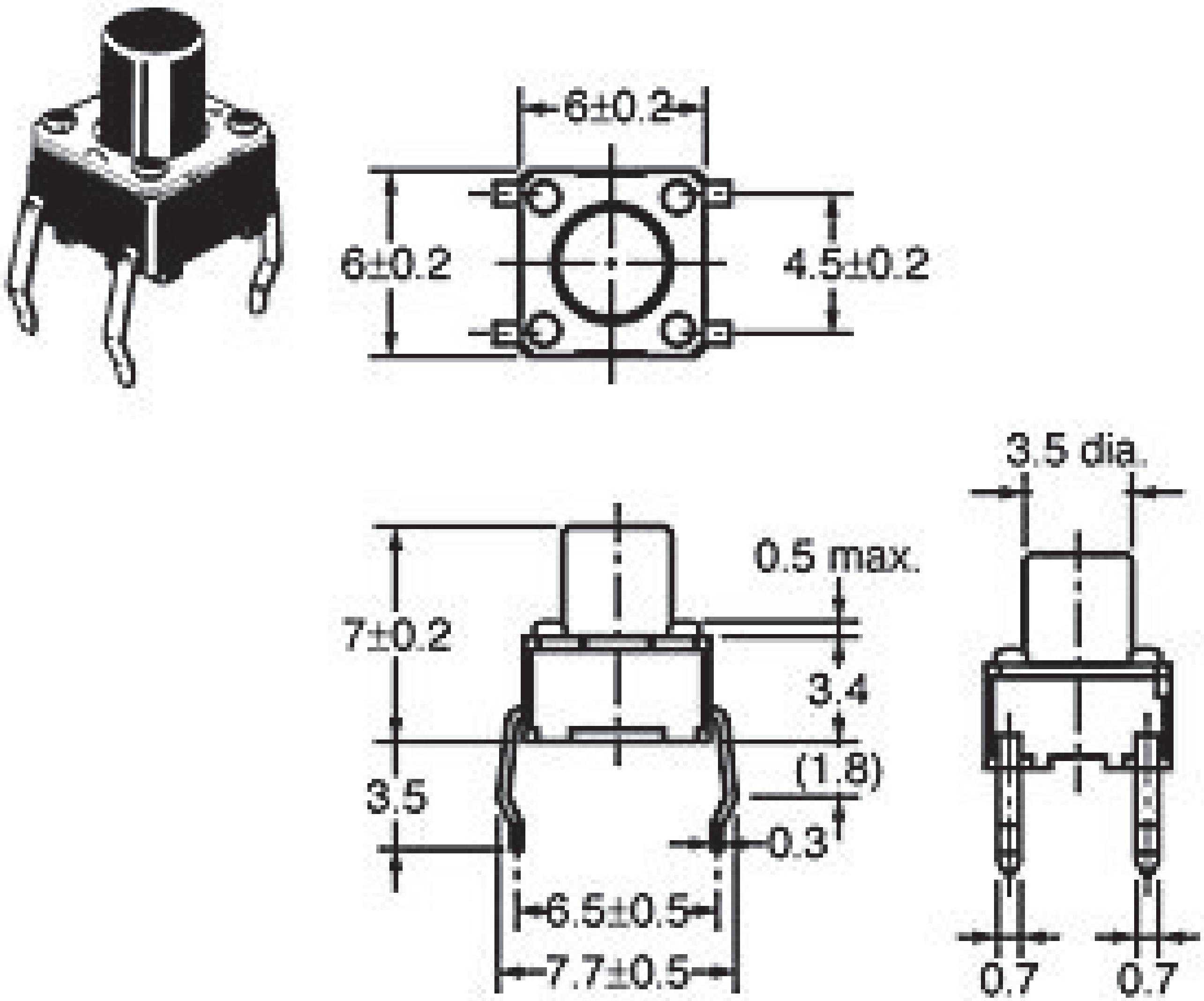


APPENDIX B

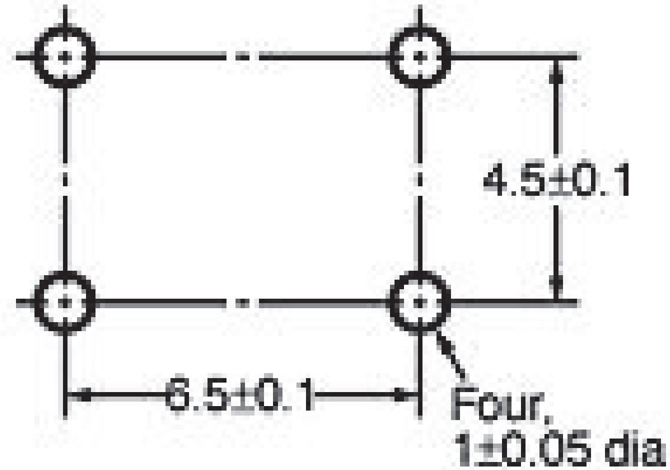
SMALL BEAR PROJECT, REV. A
BILL OF MATERIALS AND COMPONENT COST SHEET
6/5/2013

Item	Reference Designator	QTY	Vendor	Vendor P/N	Manufacturer	Mfg. P/N	Description	Cost (each)	Cost (per assy)
1	B1,B2	2	SparkFun Electronics	COM-00097	SparkFun Electronics	COM-00097	Mini Push Button Switch, up to 50mA	$0.04	$0.07
2	C1	1	Mouser Electronics	140-SEA0R33M1HBK0407	Lelon	SEA0R33M1HBK-0407P	Aluminum Electrolytic Capacitors - Leaded 50V 0.33uF 20% 4X7mm	$0.09	$0.09
3	C2	1	Mouser Electronics	81-RPER71H104K2P1A03	Murata Electronics	RPER71H104K2P1A03B	Multilayer Ceramic Capacitors MLCC - Leaded 0.1uF 50volts X7R 10% 2.5mm L/S	$0.33	$0.33
4	J3, JP3 S1, S2	4	SparkFun Electronics	PRT-09280	4UCON Technology Inc.	18688	Arduino Stackable Female Header, 6-PIN, .100" pitch	$0.50	$2.00
5	J1, JP1 J4, JP4	4	SparkFun Electronics	PRT-09279	4UCON Technology Inc.	18688	Arduino Stackable Female Header, 8-PIN, .100" pitch	$0.50	$2.00
6	J2, JP2	2	SparkFun Electronics	PRT-11376	4UCON Technology Inc.	18688	Arduino Stackable Female Header, 10-PIN, .100" pitch	$0.50	$1.00
7	LED1, LED2, LED3	3	Mouser Electronics	941-C503BAANCY0B0251	Cree, Inc.	C503B-AAN-CY0B0251	Standard LEDs - Through Hole Amber Round LED	$0.14	$0.42
8	MOLEX	1	SparkFun Electronics	TOL-11300	Molex	SDA-8981-4V	Molex Connector - Straight, Vertical Mount, 4-PIN, .200" pitch	$0.95	$0.95
9	R1	1	Mouser Electronics	271-350-RC	Xicon	271-350-RC	Metal Film Resistors - Through Hole 350ohms 1% 50PPM	$0.13	$0.13
10	R2, R3	2	Mouser Electronics	660-CFS1/4CT52R151J	KOA Speer	CFS1/4CT52R151J	Carbon Film Resistors - Through Hole 150 OHM 5% 1/4W size: 1.7 mm Dia. x 3.2 mm L	$0.15	$0.30
11	RAIL_OFF	1	Mouser Electronics	108-0001-EVX	Mountain Switch	108-0001-EVX	Toggle Switches SPST OFF-ON	$3.27	$3.27
12	S3	0.15	SparkFun Electronics	PRT-00116	Pololu Robotics & Electronics, *or equivalent*	965, *or equivalent*	Breakaway Male Header, 1x40-Pin, Straight, .100" pitch	$1.50	$0.23
13	SEL_POW	1	Mouser Electronics	633-M201201	NKK Switches	M2012SS1W01	Toggle Switches ON-NONE-ON SPDT	$2.75	$2.75
14	VOLT_REG	1	Mouser Electronics	595-UA7805CKCS	Texas Instruments	UA7805CKCS	Linear Regulators - Standard Positive V	$0.80	$0.80
15	X1	1	SparkFun Electronics	PRT-10811	4UCON Technology Inc.	18742	DC Barrel Jack Adapter - Breadboard Compatible	$0.95	$0.95
								TOTAL	$13.85

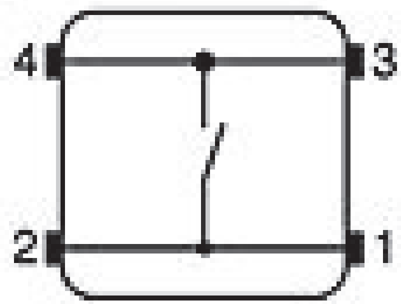
APPENDIX C: DATASHEETS



PCB Mounting (Top View)
(Single-sided PCB, t=1.6)



Terminal Arrangement
/Internal Connections
(Top View)



SEA Series

Features

- 85℃, 2,000 hours assured, standard miniature type with 7mm height for compact circuits
- RoHS Compliance

Sleeve & Marking Color: Blue & Black

Specifications

Items	Performance
Category Temperature Range	-40℃ ~ +85℃
Capacitance Tolerance	±20% (at 120Hz, 20℃)
Leakage Current (at 20℃)	I = 0.01CV or 3 (A) whichever is greater (after 2 minutes) Where, C = rated capacitance in F V = rated DC working voltage in V
Tan (at 120Hz, 20℃)	see table below
Low Temperature Characteristics (at 120Hz)	Impedance ratio shall not exceed the values given in the table below.
Endurance	see table below
Shelf Life Test	Test time: 500 hours; other items are the same as those for the Endurance.
Ripple Current & Frequency Multipliers	see table below

Tan (at 120Hz, 20℃)

Rated Voltage	4	6.3	10	16	25	35	50	63
Tan (max)	0.35	0.23	0.20	0.16	0.14	0.12	0.10	0.10

Low Temperature Characteristics (at 120Hz)

Impedance ratio shall not exceed the values given in the table below.

Rated Voltage		4	6.3	10	16	25	35	50	63
Impedance Ratio	Z(-25℃)/Z(+20℃)	7	4	3	3	2	2	2	2
	Z(-40℃)/Z(+20℃)	14	10	8	6	4	4	4	4

Endurance

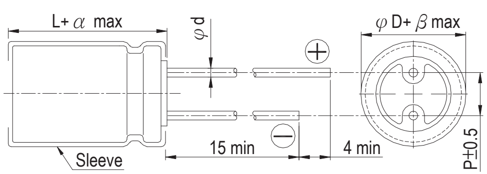
Test Time	2,000 Hrs
Capacitance Change	Within ±20% of initial value
Tan	Less than 200% of specified value
Leakage Current	Within specified value

* The above Specifications shall be satisfied when the capacitors are restored to 20℃ after the rated voltage applied with rated ripple current for 2,000 hours at 85℃.

Ripple Current & Frequency Multipliers

Cap.(F) \ Freq.(Hz)	60 (50)	120	500	1k	10k up
Under 47	0.70	1.00	1.20	1.30	1.45
100 to 1,000	0.80	1.00	1.10	1.15	1.20

Diagram of Dimensions



Lead Spacing and Diameter Unit: mm

ϕD	4	5	6.3	8	10
P	1.5	2.0	2.5	3.5	5.0
ϕd	0.45	0.5			0.6
α	1.0				1.5
β	0.5				

Dimension & Permissible Ripple Current

Dimension: ϕD × L(mm)
Ripple Current: mA/rms at 120 Hz, 85°C

F \ V. DC	Contents	4V (0G)		6.3V (0J)		10V (1A)		16V (1C)		25V (1E)		35V (1V)		50V (1H)		63V (1J)	
		ϕD×L	mA	ϕD×L	mA	ϕD×L	mA	ϕD×L	mA	ϕD×L	mA	ϕD×L	mA	ϕD×L	mA	ϕD×L	mA
1	010													4×7	10	4×7	11
2.2	2R2													4×7	15	4×7	17
3.3	3R3													4×7	18	4×7	21
4.7	4R7											4×7	22	5×7*	23	5×7*	26
10	100							4×7	25	4×7	26	5×7*	30	6.3×7*	34	6.3×7*	40
22	220			4×7	31	4×7	32	5×7*	39	5×7*	41	6.3×7*	47	6.3×7	53	8×7*	70
33	330	4×7	32	4×7	32	4×7	35	5×7	43	6.3×7	53	8×7*	71	8×7*	76	8×7	80
47	470	4×7	38	4×7	38	5×7*	47	6.3×7*	59	6.3×7	65	8×7*	83	8×7	85	8×7	95
100	101	5×7	61	6.3×7*	75	6.3×7	80	6.3×7	90	8×7	125	8×7	115	8×9	130	10×9	170
220	221	6.3×7	90	6.3×7	99	8×7	140	8×7	146	8×9	190	10×9	215				
330	331	8×7	129	8×7	156	8×7	165	8×9	185	10×9	265						
470	471	8×7	154	8×7	175	8×9	215	10×9	255								
1,000	102	8×9	200	10×9	205												

Note: Case size in mark of " * " is available to product down size.

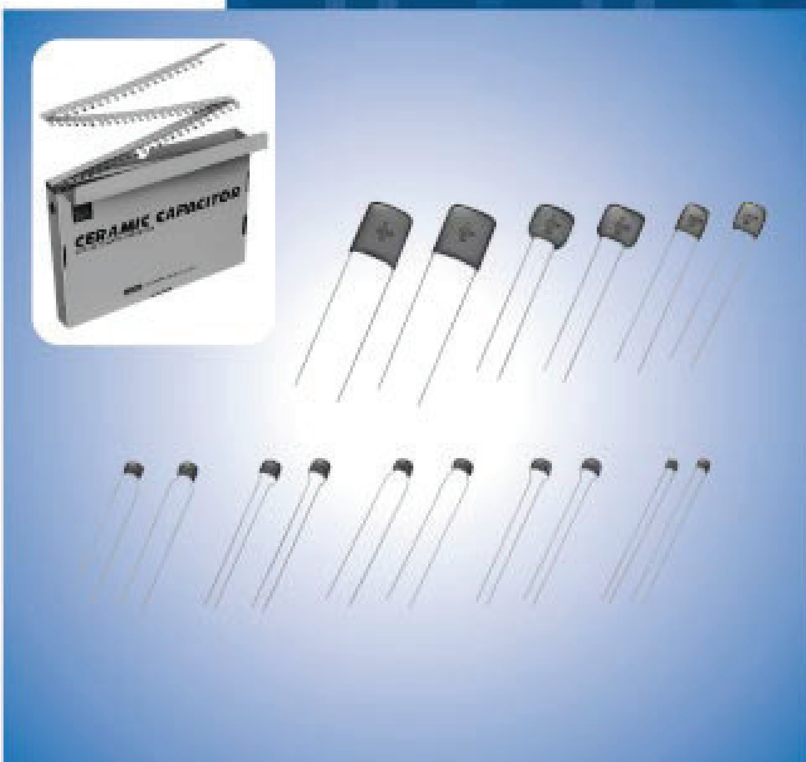
Part Numbering System

SEA series	470 F	±20%	6.3V	Bulk Package	Gas Type	8ϕ×7L	Pb-free and PET coating case
SEA	**471**	**M**	**0J**	**BK**	**-**	**0807**	
Series	Capacitance	Capacitance Tolerance	Rated Voltage	Lead Configuration & Package	Rubber Type	Case Size	Lead Wire and Coating Type

Note: For more details, please refer to "Part Numbering System (Radial Type)" on page 10.

⚠Note • Please read rating and ⚠CAUTION (for storage, operating, rating, soldering, mounting and handling) in this catalog to prevent smoking and/or burning, etc.
• This catalog has only typical specifications because there is no space for detailed specifications. Therefore, please review our product specifications or consult the approval sheet for product specifications before ordering.

Radial Lead Type Monolithic Ceramic Capacitors



Innovator in Electronics

Murata Manufacturing Co., Ltd.

Cat.No.C49E-21

Part Numbering

Radial Lead Type Monolithic Ceramic Capacitors

(Part Number) RP E R7 1H 104 K 2 M1 A03 A
❶ ❷ ❸ ❹ ❺ ❻ ❼ ❽ ❾ ❿

❶Product ID

❷Series/Terminal

Product ID	Series/Terminal	
RP	**E**	Radial Lead Type Monolithic Ceramic Capacitors (DC25V-DC100V)
RH	**E/D**	Radial Lead Type Monolithic Ceramic Capacitors 150°C max. (for Automotive) (DC50V-DC100V)
RD	**E**	Radial Lead Type Monolithic Ceramic Capacitors (For Commercial Use Only) (DC25V-DC630V)

❸Temperature Characteristics

Code	Temperature Characteristics	Reference Temperature	Temperature Range	Capacitance Change or Temperature Coefficient	Operating Temperature Range
5C	C0G*	25°C	25 to 125°C	0±30ppm/°C	-55 to 125°C
5G	X8G*	25°C	25 to 150°C	0±30ppm/°C	-55 to 150°C
C7	X7S	25°C	-55 to 125°C	±22%	-55 to 125°C
D7	X7T	25°C	-55 to 125°C	+22, -33%	-55 to 125°C
F1	F	20°C	-25 to 85°C	+30, -80%	-25 to 85°C
F5	Y5V	25°C	-30 to 85°C	+22, -82%	-30 to 85°C
L8	X8L	25°C	-55 to 125°C	±15%	-55 to 150°C
			125 to 150°C	+15, -40%	
R7	X7R	25°C	-55 to 125°C	±15%	-55 to 125°C

*Please refer to table for Capacitance change under reference temperature.

• Capacitance change from each temperature

Char.	Nominal Values (ppm/°C) *1	Capacitance Change from 25°C (%)					
		-55°C		-30°C		-10°C	
		Max.	Min.	Max.	Min.	Max.	Min.
C0G	0±30	0.58	-0.24	0.40	-0.17	0.25	-0.11
X8G							

*1: Nominal values denote the temperature coefficient within a range of 25 to 125°C.

❹Rated Voltage

Code	Rated Voltage
1E	DC25V
1H	DC50V
2A	DC100V
2E	DC250V
2W	DC450V
2J	DC630V

❺Capacitance

Expressed by three-digit alphanumerics. The unit is pico-farad (pF). The first and second figures are significant digits, and the third figure expresses the number of zeros that follow the two numbers.
If there is a decimal point, it is expressed by the capital letter "**R**." In this case, all figures are significant digits.

❻Capacitance Tolerance

Code	Capacitance Tolerance	Temperature Characteristics	Capacitance Step
C	±0.25pF	C0G	≦5pF : 1pF Step
D	±0.5pF		6 to 9pF : 1pF Step
J	±5%	C0G/X8G	≧10 : E12 Series
K	±10%	X7S/X7T/X7R/X8L	E6 Series
M	±20%	X7S/X7T/X7R/X8L	E3 Series
Z	+80%, -20%	F/Y5V	E3 Series

Continued on the following page. ↗

Continued from the preceding page.

❼Dimensions (LxW)

Code	Dimensions (LxW)
0	4.0X3.5mm or 5.0X3.5mm (Depends on Part Number List)
1	4.0X3.5mm or 4.5X3.5mm or 5.0X3.5mm (Depends on Part Number List)
2	5.0X3.5mm or 5.5X4.0mm or 5.7X4.5mm (Depends on Part Number List)
3	5.0X4.5mm or 5.5X5.0mm or 6.0X5.5mm (Depends on Part Number List)
5	7.5X7.5mm *
6	10.0X10.0mm
7	12.5X12.5mm
8	7.5X5.5mm
U	7.7X12.5mm *
W	5.5X7.5mm

* DC630V: W+0.5mm

❽Lead Style

Code	Lead Style	Lead Spacing
A2	Straight Long	2.5mm
B1	Straight Long	5.0mm
C1	Straight Long	10.0mm
DB	Straight Taping	2.5mm
E1/E2	Straight Taping	5.0mm
K1	Inside Crimp	5.0mm
M1/M2	Inside Crimp Taping	5.0mm
P1	Outside Crimp	2.5mm
S1/S2	Outside Crimp Taping	2.5mm

Lead distance between reference and bottom planes.
M1, S1: H_0 = 16.0±0.5mm
M2, S2: H_0 = 20.0±0.5mm
E1: H = 17.5±0.5mm
E2: H = 20.0±0.5mm

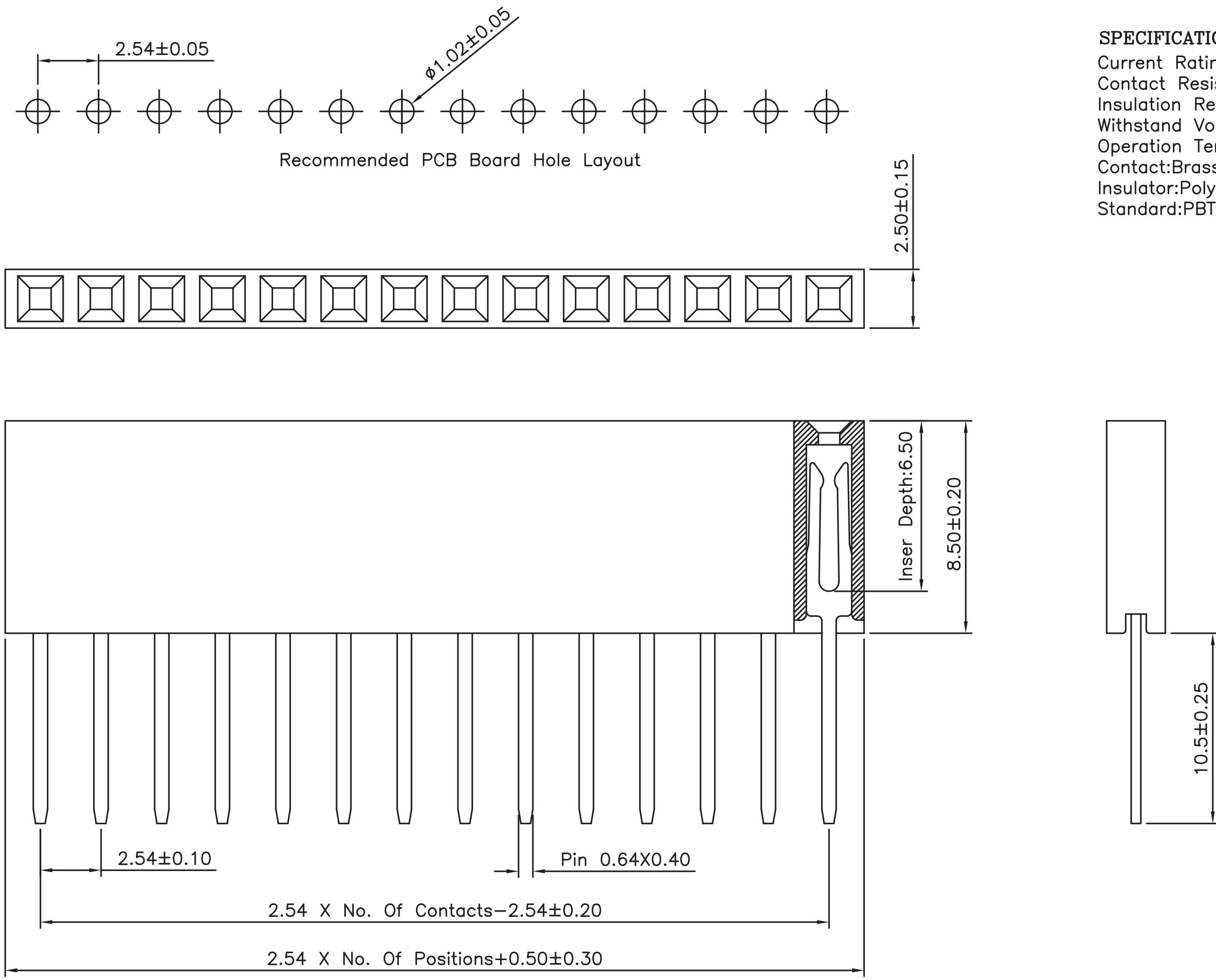
❾Individual Specification Code

Expressed by three-digit alphanumerics

❿Packaging

Code	Packaging
A	Ammo Pack
B	Bulk

File No.	432	SHEET	1/1



SPECIFICATIONS

Current Rating:3Amp
Contact Resistance:20 Milliohms Max.
Insulation Resistance:1000 Megohms Min.
Withstand Voltage:1000V AC/Minute
Operation Temperature:−40°C To +105°C
Contact:Brass,Gold Plated Over Nickel
Insulator:Polyester(UL94V−0)
Standard:PBT+30%G.F

4UCON 元化興業股份有限公司
4UCON TECHNOLOGY INC.

18688	APPROVED BY	MEI
	CHECKED BY	HZF
	DRAWING BY	PEGA
SCALE: NONE	REV.	A0

UNLESS OTHERWISE SPECIFIED TOLERANCES ARE
.X ±0.40
.XX ±0.25
.XXX ±0.15
ANG. ±3°
UNIT mm

NO.	REV.	REVISIONS	CHK	DATE
	A0	NEW RELEASE	HZF	02/08/2007

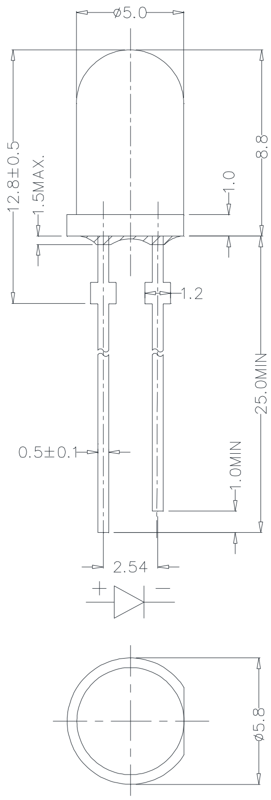
MECHANICAL DIMENSIONS

All dimensions are in mm. Tolerance is ±0.25 mm unless otherwise noted.

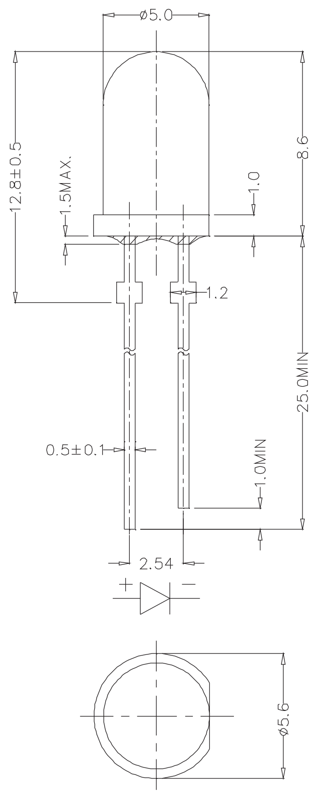
An epoxy meniscus may extend about 1.5 mm down the leads.

Burr around bottom of epoxy may be 0.5 mm max.

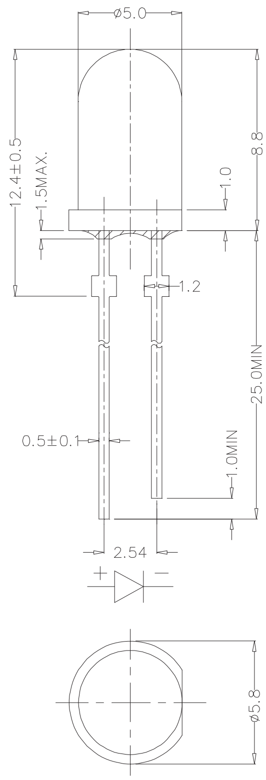
C503B-RAS/AAS:



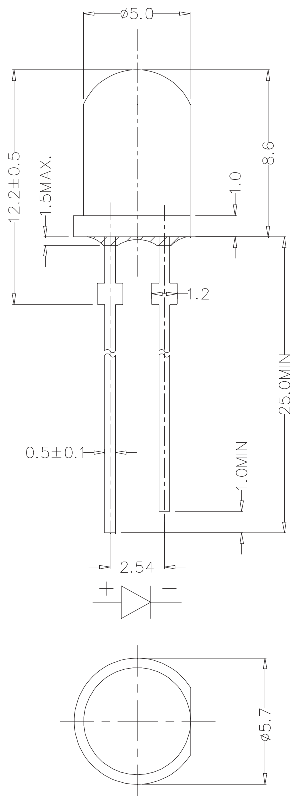
C503B-RBS/ABS:



C503B-RCS:



C503B-ACS:



NOTES

RoHS Compliance

The levels of environmentally sensitive, persistent biologically toxic (PBT), persistent organic pollutants (POP), or otherwise restricted materials in this product are below the maximum concentration values (also referred to as the threshold limits) permitted for such substances, or are used in an exempted application, in accordance with EU Directive 2002/95/EC on the restriction of the use of certain hazardous substances in electrical and electronic equipment (RoHS), as amended through April 21, 2006.

Vision Advisory Claim

Users should be cautioned not to stare at the light of this LED product. The bright light can damage the eye.

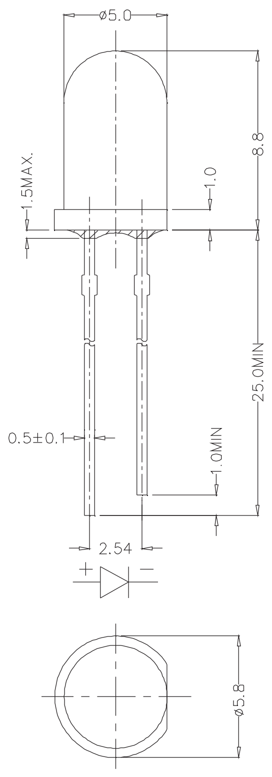
MECHANICAL DIMENSIONS

All dimensions are in mm. Tolerance is ±0.25 mm unless otherwise noted.

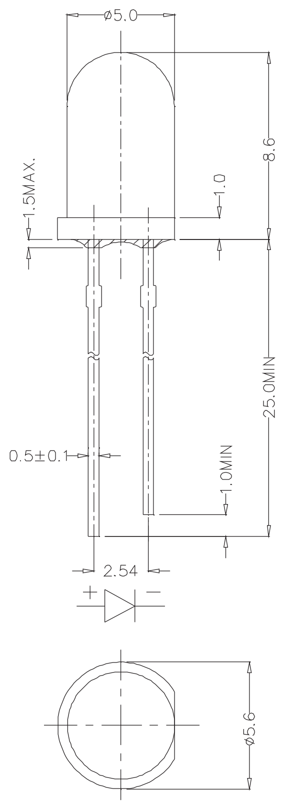
An epoxy meniscus may extend about 1.5 mm down the leads.

Burr around bottom of epoxy may be 0.5 mm max.

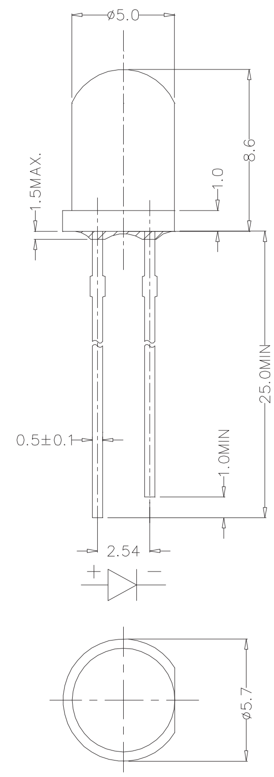
C503B-RAN/AAN/RCN:



C503B-RBN/ABN:



C503B-ACN:



NOTES

RoHS Compliance

The levels of environmentally sensitive, persistent biologically toxic (PBT), persistent organic pollutants (POP), or otherwise restricted materials in this product are below the maximum concentration values (also referred to as the threshold limits) permitted for such substances, or are used in an exempted application, in accordance with EU Directive 2002/95/EC on the restriction of the use of certain hazardous substances in electrical and electronic equipment (RoHS), as amended through April 21, 2006.

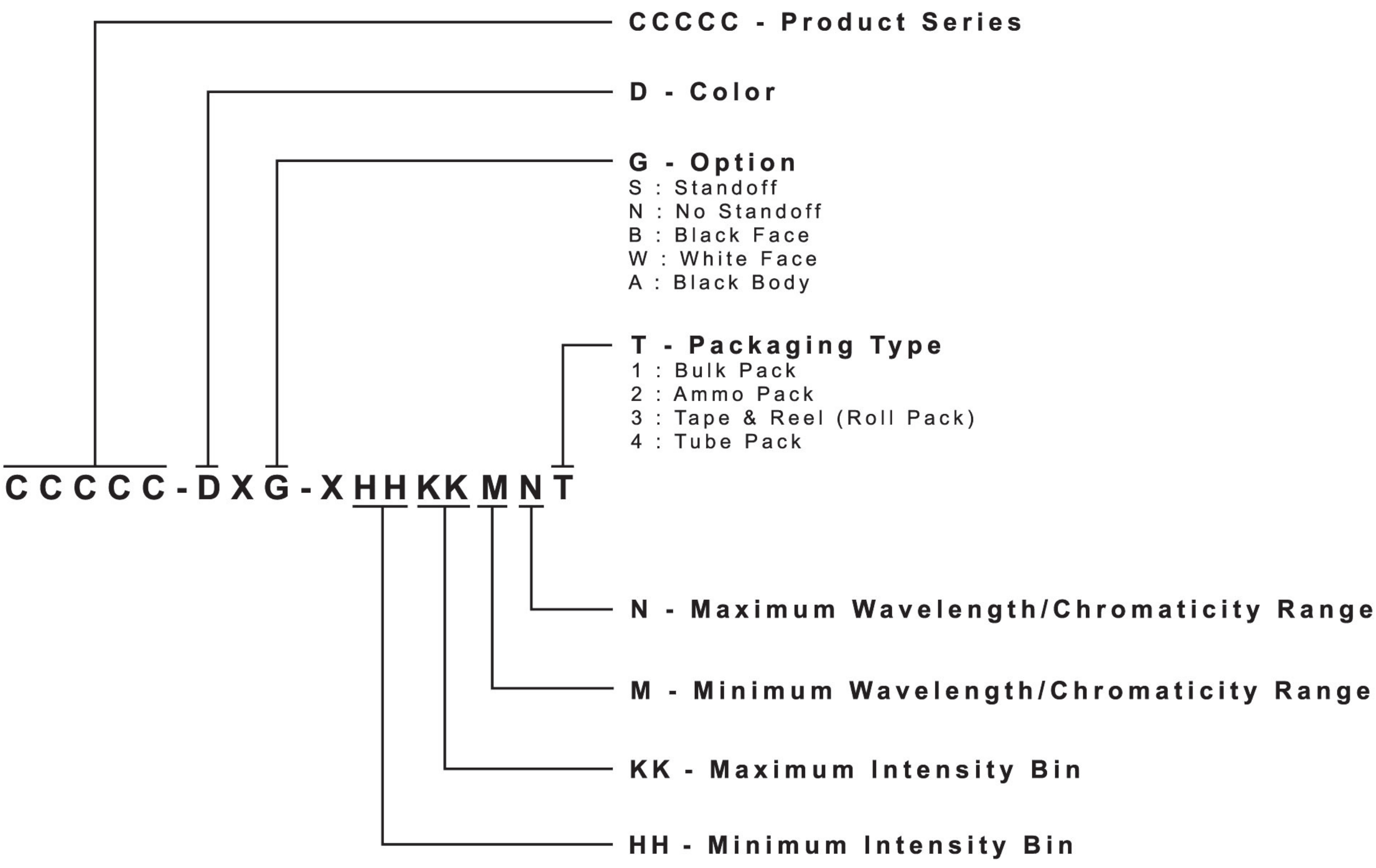
Vision Advisory Claim

Users should be cautioned not to stare at the light of this LED product. The bright light can damage the eye.

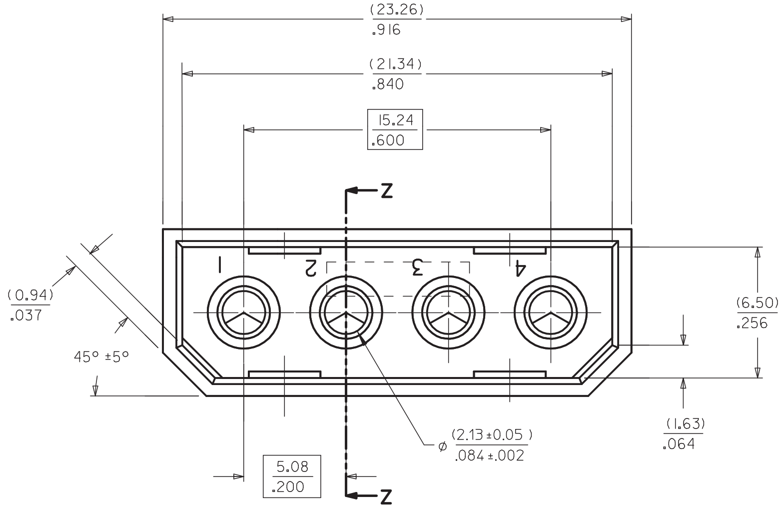
KIT NUMBER SYSTEM

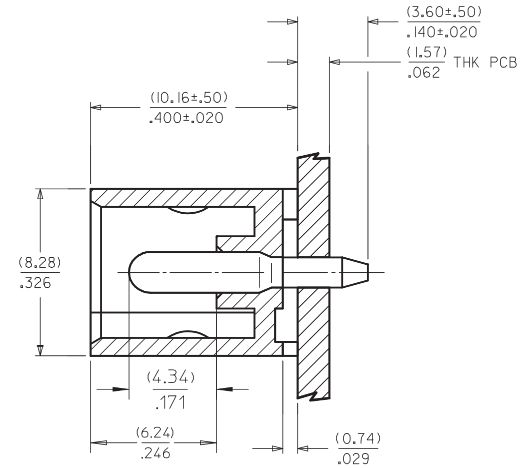
All dimensions in mm.Cree LED lamps are tested and sorted into performance bins. A bin is specified by ranges of color, forward voltage, and brightness. Sorted LEDs are packaged for shipping in various convenient options. Please refer to the "Cree LED Lamp Packaging Standard" document for more information about shipping and packaging options.

Cree LEDs are sold by order codes in combinations of bins called kits. Order codes are configured in the following manner:



MAT'L NO.	ENG NO.
15-24-4449	A-8981-4V-LF





Section Z-Z

NOTES:

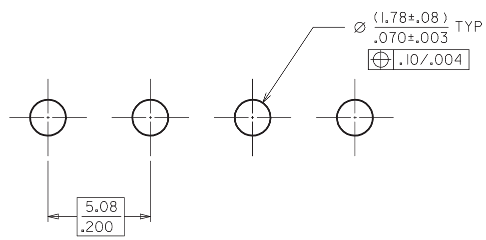
1. PART TO MATE WITH HOUSING 8981-4P AND TERMINAL 8980-3*.
2. PRODUCT SPEC. PS -8981-4V* APPLIES
3. PARTS TO BE PACKED IN MOISTURE BARRIER BAG.

MATERIAL :

HOUSING : NYLON 6/6, UNFILLED, 94V-2, COLOR - NATURAL (OFF-WHITE).

TERMINAL : PHOSPHOR BRONZE, C51000

PLATING : POSTPLATE (0.00254)/.000100 MIN. TIN OVER (0.00127)/.000050 MIN. NICKEL OVERALL



RECOMMENDED P C B HOLE LAYOUT

REV	DESCRIPTION		
L3	ADD NOTE 3.	EC NO: DRWN:SKANG CH'KD: APPR:	2011/12/07

QUALITY SYMBOLS: ▼=0 ; Ⓒ=0

GENERAL TOLERANCES (UNLESS SPECIFIED)	mm	INCH
4 PLACES	± ---	± ---
3 PLACES	± ---	±.008
2 PLACES	±0.20	± ---
1 PLACE	± ---	± ---
ANGULAR ± 3 °		

DRAFT WHERE APPLICABLE MUST REMAIN WITHIN DIMENSIONS

DIMENSION STYLE	SCALE	DESIGN UNITS	
MM/IN	NTS	METRIC	THIRD ANGLE PROJECTION

	BY	DATE
DRAWN BY	TAY	1989/05/26
CHECKED BY	NK	1989/05/26
APPROVED BY	AB	1989/05/26

TITLE: HEADER, 4 CKTS VERTICAL MOUNT

molex MOLEX INCORPORATED

MATERIAL NO.: SEE TABLE

DOCUMENT NO.: SDA-8981-4V

SHEET NO.: 1 OF 1

SIZE: A3

Metal Film Fixed Resistors (RoHS Compliant) MF-RC Series

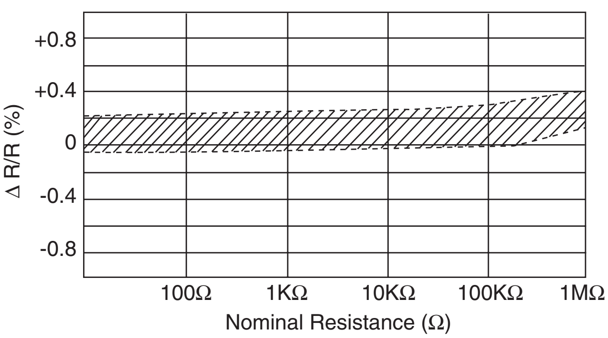
FEATURES

- Temperature Range: -55°C ~ +155°C
- ±1% tolerance
- Epoxy coated miniature resistors
- Nickel chromium ceramic substrate
- Alloy coated leads
- Welded end caps
- EIA standard color coding

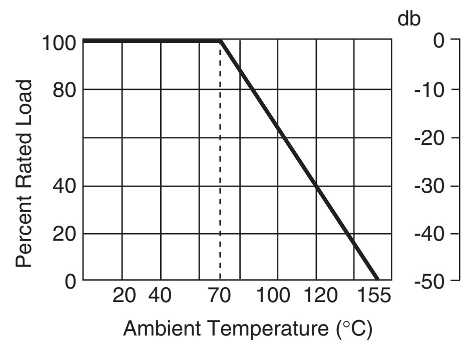
LEAD-FREE

Environmental Commitment

LOAD LIFE



DERATING CURVE



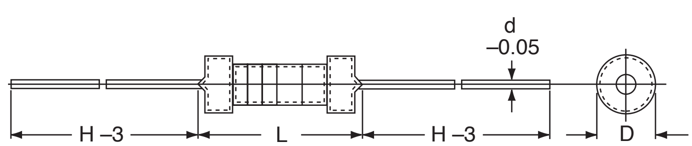
PART NUMBERING SYSTEM

Code:	Package:
	Bulk
REEL	Tape and Reel
AP	Ammo Pack

2 7 0 – 1 0 0 / R E E L - R C

Series — Ohms — Package Code — Suffix (RoHS Compliant)

SERIES, WATTAGE, SIZE, VOLTAGE, DIMENSIONS, AND AVAILABLE PACKAGING

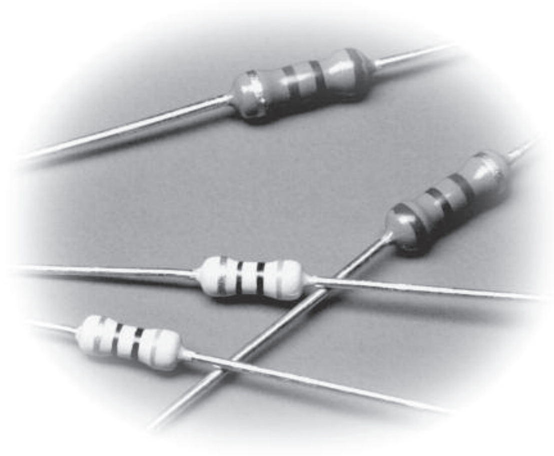


Series	Watts	Standard Range of Values (Ω)	Voltage (max.)		Dimensions (mm)				Standard Quantities Available		
			W.V.	O.V.	L max.	D max.	H	d	Bulk	Tape and Reel	Ammo Pack
270	1/8	1~10M	200	400	3.5	1.85	28	0.45	1,000	5,000	2,000
271	1/4	1~10M	250	500	6.8	2.5	28	0.54	1,000	5,000	1,000
273	1/2	1~10M	350	700	10	3.5	28	0.54	1,000	3,000	1,000

CF

carbon film leaded resistors

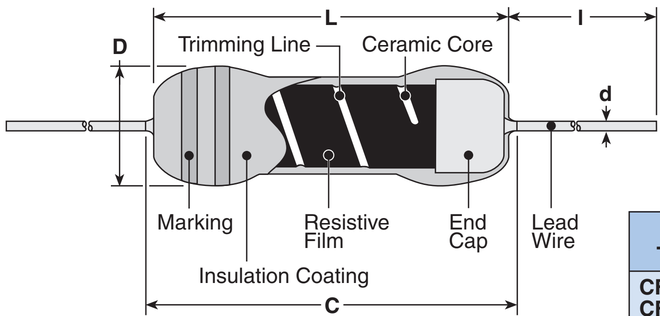
EU RoHS COMPLIANT



features

- Flameproof coating is available (specify “CFP”)
- Reduced body size (specify “CFS/CFPS”)
- Suitable for automatic machine insertion
- Marking: Venetian red with color-coded bands on CF
  Green body color with color-coded bands on CFP
  Ivory body color with color-coded bands on CFS1/4
- Products with lead-free terminations meet EU RoHS and China RoHS requirements

dimensions and construction



Type	Dimensions inches (mm) L	C (max.)	D	d (nom.)	l** Standard	l** Long
CFS1/4 CFPS1/4	.126±.008 (3.2±0.2)	.134 (3.4)	.067±.008 (1.7±0.2)	.018 (0.45)	.551 Min.* (14.0 Min.)	.787 Min.*** (20.0 Min.)
CF1/4 CFP1/4	.240±0.2 (6.1±0.5)	.280 (7.1)	.092±.012 (2.3±0.3)	.024 (0.6)	.787 Min. (20.0 Min.)	—
CFS1/2 CFPS1/2	.248±.02 (6.3±0.5)	.280 (7.1)	.112±.012 (2.85±0.3)	.024 (0.6)		

* Forming code S is applied for bulk type.
** Lead length changes depending on taping and forming type.
*** Long type is custom-made

ordering information

New Part #

CF	1/4	C	T52	R	103	J
Type	**Power Rating**	**Termination Material**	**Taping and Forming**	**Packaging**	**Nominal Resistance**	**Tolerance**
CF CFP	S1/4: 0.25W 1/4: 0.25W S1/2: 0.5W	C: SnCu	Axial: T26, T52, L52 Radial: VT, MT, MHT, VTP, VTE U Forming: U, UCL M Forming: M5, M10, M12.5 L Forming: L10, L12.5	A: Ammo R: Reel	2 significant figures + 1 multiplier “R” indicates decimal on value <10Ω	G: ±2% J: ±5%

For further information on packaging, please refer to Appendix C.

 12/18/12

KOA SPEER ELECTRONICS, INC.

CF

carbon film leaded resistors

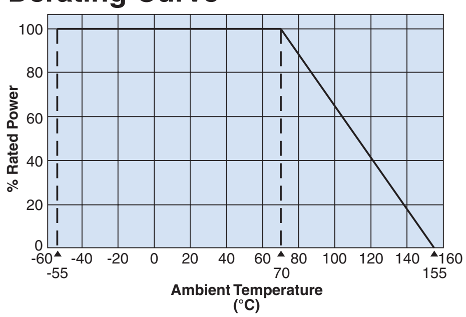
applications and ratings

<table>
<tr><th rowspan="2">Part Designation</th><th rowspan="2">Power Rating @ 70°C</th><th rowspan="2">Minimum Dielectric Withstanding Voltage</th><th colspan="4">T.C.R. (ppm/°C)</th><th rowspan="2">Resistance Range E-24 (G±2%)</th><th rowspan="2">Resistance Range E-24 (J±5%)</th><th rowspan="2">Absolute Maximum Working Voltage</th><th rowspan="2">Absolute Maximum Overload Voltage</th></tr>
<tr><th>+350 to -450</th><th>0 to -700</th><th>0 to -1000</th><th>0 to -1300</th></tr>
<tr><td>CFS1/4</td><td rowspan="4">0.25W</td><td rowspan="2">300V</td><td rowspan="2">2.2Ω - 47kΩ</td><td rowspan="2">51kΩ - 100kΩ</td><td rowspan="2">110kΩ - 330kΩ</td><td rowspan="2">360kΩ - 1MΩ</td><td>10Ω - 330kΩ</td><td>2.2Ω - 1MΩ</td><td rowspan="2">250V</td><td rowspan="2">500V</td></tr>
<tr><td>CFPS1/4</td><td>10Ω - 100kΩ</td><td>2.2Ω - 1MΩ</td></tr>
<tr><td>CF1/4</td><td rowspan="2">500V</td><td rowspan="2">2.2Ω - 100kΩ</td><td rowspan="2">110kΩ - 330kΩ</td><td rowspan="2">360kΩ - 1MΩ</td><td>1.1MΩ - 5.1MΩ</td><td rowspan="4">10Ω - 1MΩ</td><td>2.2Ω - 5.1MΩ</td><td rowspan="2">300V</td><td rowspan="2">600V</td></tr>
<tr><td>CFP1/4</td><td>—</td><td>2.2Ω - 1MΩ</td></tr>
<tr><td>CFS1/2</td><td rowspan="2">0.50W</td><td rowspan="2">700V</td><td>1.0Ω - 91kΩ</td><td rowspan="2">100kΩ - 1MΩ</td><td>1.1MΩ - 2.2MΩ</td><td>2.4MΩ - 5.1MΩ</td><td>1.0Ω - 5.1MΩ</td><td rowspan="2">350V</td><td rowspan="2">700V</td></tr>
<tr><td>CFPS1/2</td><td>2.2Ω - 91kΩ</td><td>—</td><td>—</td><td>2.2Ω - 1MΩ</td></tr>
</table>

Operating temperature: -55°C ~ +155°C

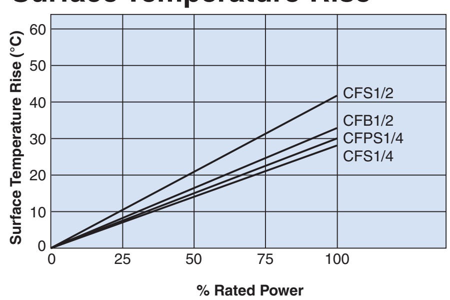
environmental applications

Derating Curve

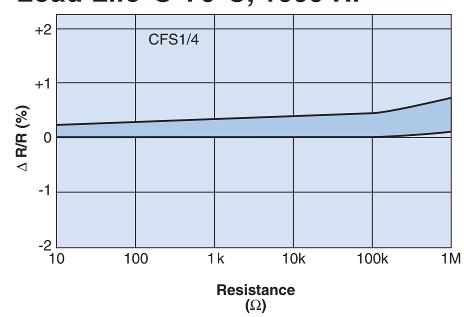


For resistors operated at an ambient temperature of 70°C or above, a power rating shall be derated in accordance with the above derating curve.

Surface Temperature Rise



Load Life @ 70°C, 1000 Hr

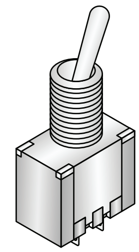


Performance Characteristics

Parameter	Requirement Δ R ±(% + 0.05Ω) Limit	Typical	Test Method
Resistance	Within specified tolerance	—	Measuring points are at 10mm ±1mm from the end cap.
T.C.R.	Within specified T.C.R.	—	Room temperature +100°C
Overload (Short time)	±1%	±0.5%	Rated voltage x 2.5 or max. overload voltage for 5 seconds, whichever is lower
Resistance to Solder Heat	±1%	±0.5%	260°C ±5°C, 10 seconds ± 1 second
Terminal Strength	No lead-coming off and loose terminals	—	Twist 360°C, 5 times
Rapid Change of Temperature	±1%	±0.5%	-55°C (30 minutes), +125°C (30 minutes), 5 cycles
Moisture Resistance	±5%	±2.5%	40°C ± 2°C, 90 - 95% RH, 1000 hours, 1.5 hr ON, 0.5 hr OFF cycle
Endurance at 70°C	±3%	±1.5%	70°C ± 2°C, 1000 hours, 1.5 hr ON, 0.5 hr OFF cycle
Resistance to Solvent (CFS & CFPS only)	No abnormality in appearance. Marking shall be easily legible.	—	Ultrasonic washing with Isopropyl alcohol for 2 minutes. Power: 0.3W/cm², f: 28kHz, temp: 35°C±5°C
Flame Retardant (CFS & CFPS only)	No evidence of flaming or self-flaming	—	Flame test: The test flame shall be applied and removed for each 15 seconds respectively to repeat the cycle 5 times. Overload flame retardant: Power (AC) corresponding to 2, 4, 8, 16 and 32 times the power rating shall be applied for each 1 minute until disconnection occurs. However the applied voltage shall not exceed 4 times the maximum operating voltage.

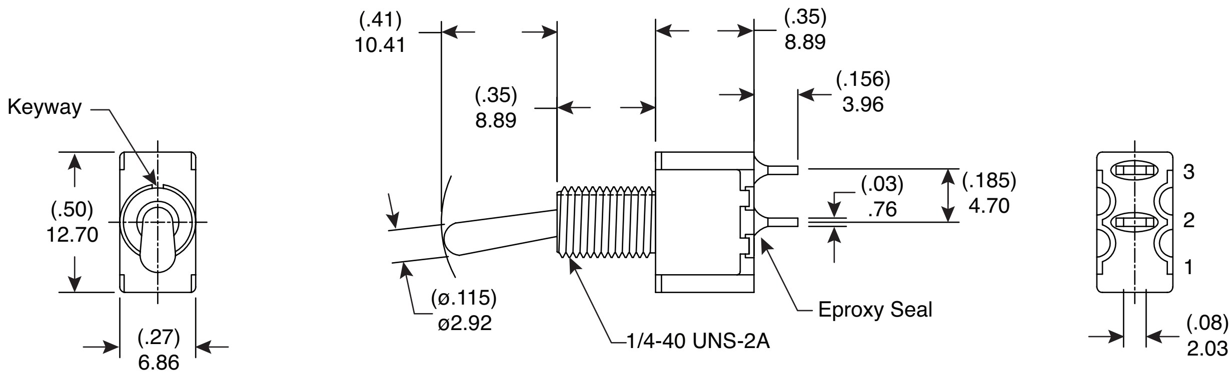
5/10/13

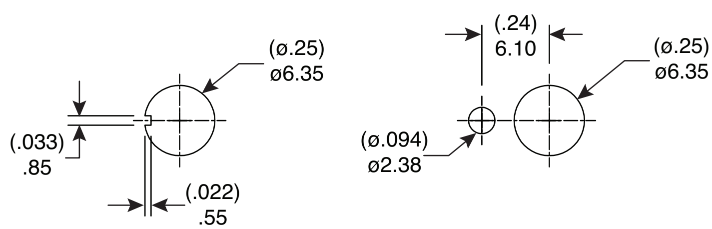
Dimensions: mm (in.)



POS. 1	POS. 2
ON	OFF
2-3	OPEN
1 2 Comm. 3	

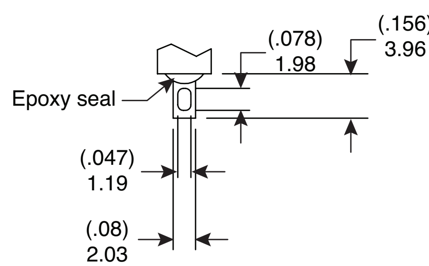
SWITCH FUNCTION





Without locking ring. With standard locking ring.

Panel Mounting



Solder Lug

Electrical Specifications:

- Rating: 2A / 250VAC, 5A / 120 VAC or 28 VDC
- Contact Resistance: 10mΩ max. initial @ 2-4VDC 100mA
- Insulation Resistance: 1,000MΩ min.
- Dieletric Strength: 1,000 V RMS@sea level

Mechanical Specifications:

- Mechanical Life: 40,000 make-and-break cycles
- Operating Temperature: -30°C to 85°C
- Type: SPST
- Circuit: On-Off

Materials:

- Case: Diallyl phthalate (DAP)
- Actuator: Brass, chrome plated
- Bushing: Brass, nickel plated
- Housing: Stainless Steel
- Terminal/Contact: Silver or gold plated

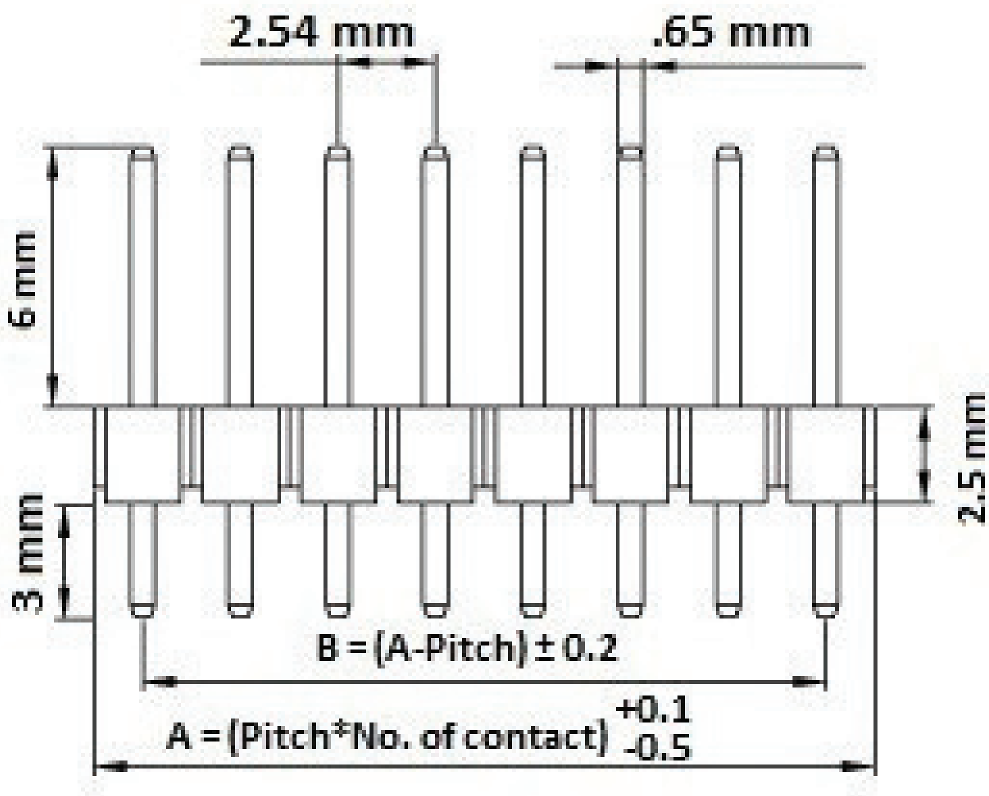
Note:

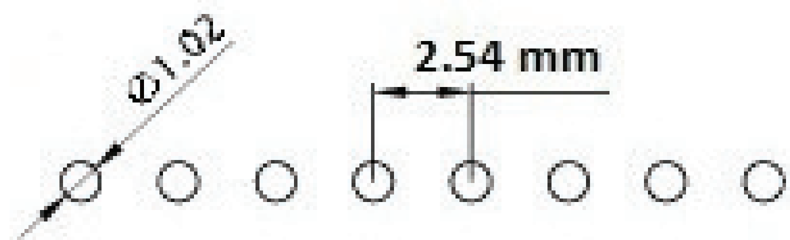
- RoHS compliant by exemption

Miniature Toggle Switch

108-0001-EVX

Mountain Switch


2.54 mm
.65 mm
6 mm
2.5 mm
3 mm
B = (A-Pitch) ± 0.2
A = (Pitch*No. of contact) +0.1 -0.5


⌀1.02
2.54 mm

PCB LAYOUT

CONTACT MATERIALS & RATINGS

W	Silver over Silver	Power Level	6A @ 125V AC & 3A @ 250V AC
G	Gold over Brass or Copper	Logic Level	6A @ 125V AC & 3A @ 250V AC

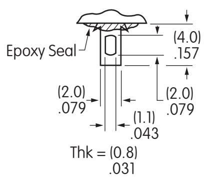
Note: See Supplement section to find complete explanation of operating range.

A	Gold over Silver	Power Level or Logic Level	6A @ 125V AC or 0.4VA maximum @ 28V AC/DC maximum

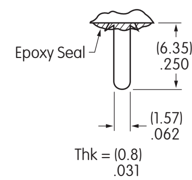
Note: This dual rated option is suitable when two or more identical switches are used in logic and in power circuits within the same application. See Supplement section to find complete explanation of dual rating and operating range.

TERMINALS

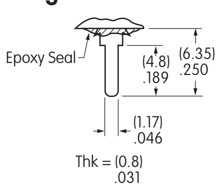
01 Solder Lug

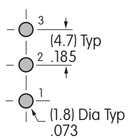


02 .062″ (1.57mm) Wide Quick Connect

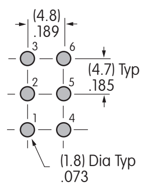


03 .250″ (6.35mm) Straight PC

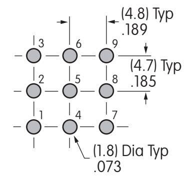




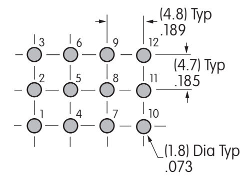
Single Pole



Double Pole



Three Pole



Four Pole

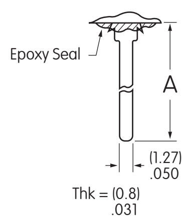
05 .425″ (10.8mm) Wirewrap or Extended PC

06 .750″ (19.05mm) Wirewrap or Extended PC

07 .964″ (24.5mm) Wirewrap or Extended PC

08 1.062″ (27.0mm) Wirewrap or Extended PC

If using as extended PC terminal, refer to the above footprints.



Dimension A = terminal lengths as shown beside the terminal codes at the left.

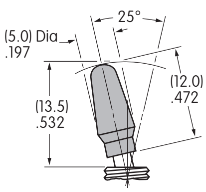
OPTIONAL CAPS & CAP COLORS

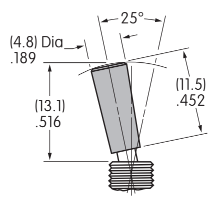
B **AT415 Lever Cap for S Bat Toggle**
Material: Polyethylene

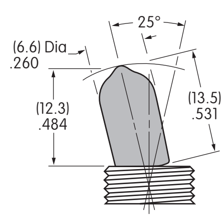
C **AT444 Conical Cap for S Bat Toggle**
Material: Polyethylene

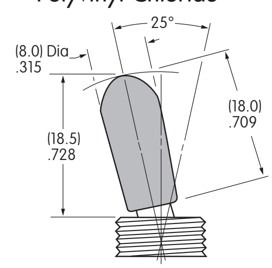
R **AT434 Lever Cap for B Toggle**
Material: Polyvinyl Chloride

V **AT406 Lever Cap for B2 Toggle**
Material: Polyvinyl Chloride









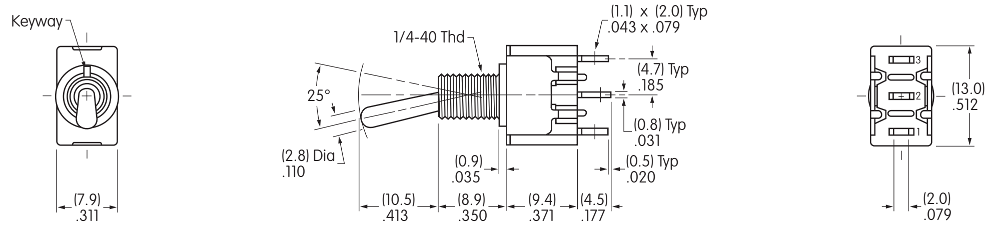
Cap Colors Available: **A** Black **B** White **C** Red **E** Yellow **F** Green **G** Blue

Toggles A
Rockers
Pushbuttons
Illuminated PB
Programmable
Keylocks
Rotaries
Slides
Tactiles
Tilt
Touch
Indicators
Accessories
Supplement

TYPICAL SWITCH DIMENSIONS

Solder Lug

Single Pole

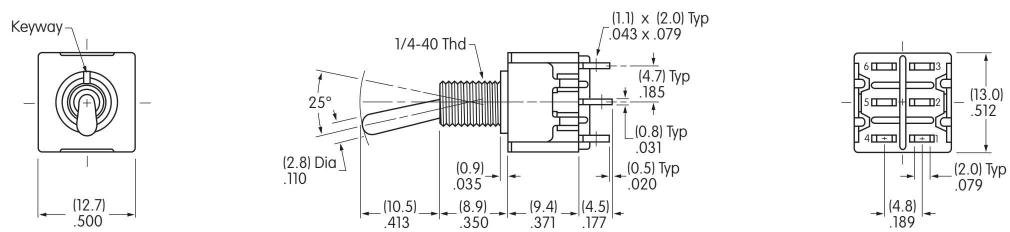


M2012SS1W01

M2011 model does not have terminal 1.

Solder Lug

Double Pole

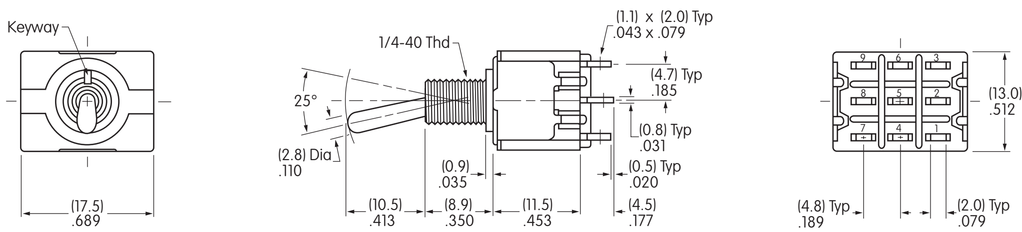


M2022SS1W01

M2021 model does not have terminals 1 & 4.

Solder Lug

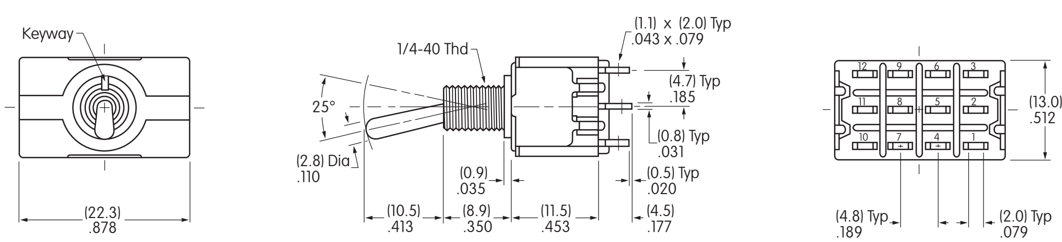
Three Pole



M2032SS1W01

Solder Lug

Four Pole



M2042SS1W01

uA7824 Electrical Characteristics (continued)

at specified virtual junction temperature, V_I = 33 V, I_O = 500 mA (unless otherwise noted)

PARAMETER	TEST CONDITIONS	T_J (1)	µA7824C MIN	µA7824C TYP	µA7824C MAX	UNIT
Output voltage regulation	I_O = 5 mA to 1.5 A	25°C		12	480	mV
	I_O = 250 mA to 750 mA			4	240	
Output resistance	f = 1 kHz	0°C to 125°C		0.028		Ω
Temperature coefficient of output voltage	I_O = 5 mA	0°C to 125°C		–1.5		mV/°C
Output noise voltage	f = 10 Hz to 100 kHz	25°C		170		µV
Dropout voltage	I_O = 1 A	25°C		2		V
Bias current		25°C		4.6	8	mA
Bias current change	V_I = 27 V to 38 V	0°C to 125°C			1	mA
	I_O = 5 mA to 1 A				0.5	
Short-circuit output current		25°C		150		mA
Peak output current		25°C		2.1		A

APPLICATION INFORMATION

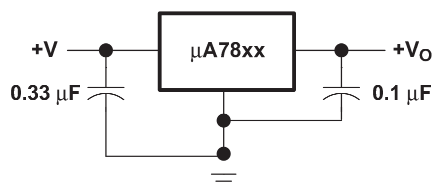


Figure 2. Fixed-Output Regulator

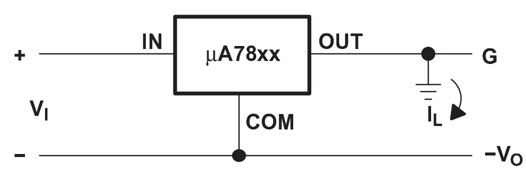
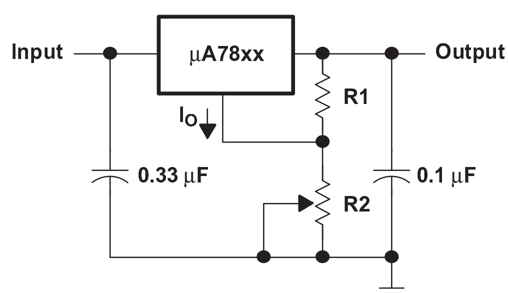


Figure 3. Positive Regulator in Negative Configuration (V_I Must Float)

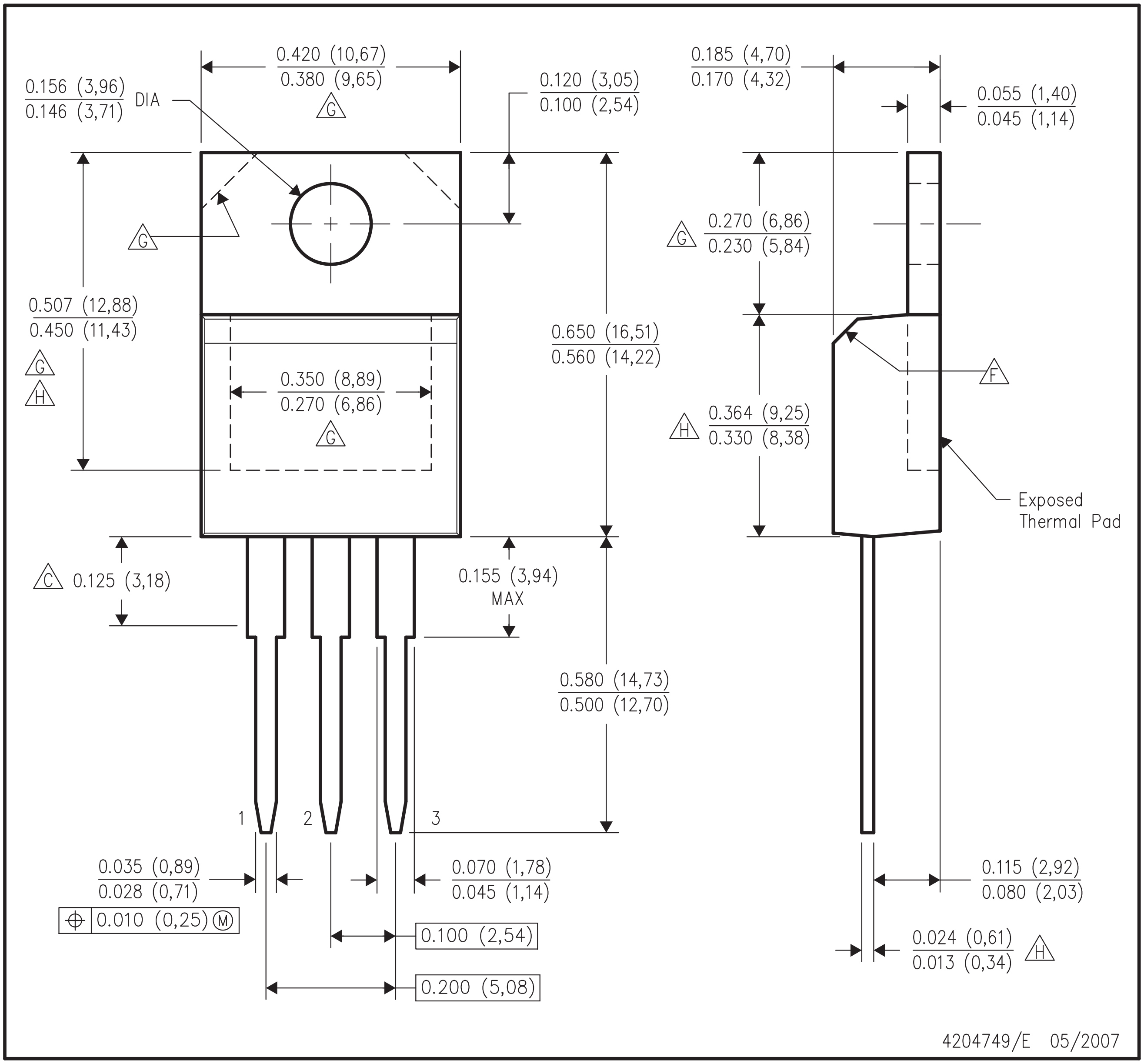


A: The following formula is used when V_{xx} is the nominal output voltage (output to common) of the fixed regulator:

$$V_O = V_{xx} + \left(\frac{V_{xx}}{R1} + I_Q\right)R2$$

Figure 4. Adjustable-Output Regulator

KCS (R-PSFM-T3) PLASTIC FLANGE-MOUNT PACKAGE

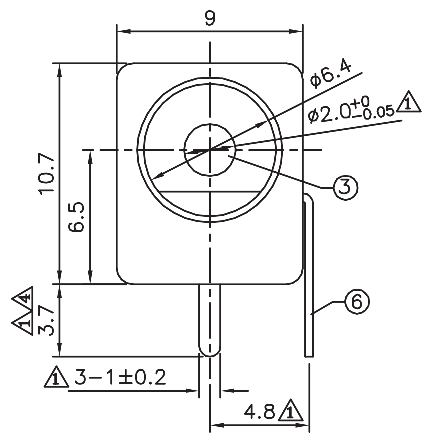


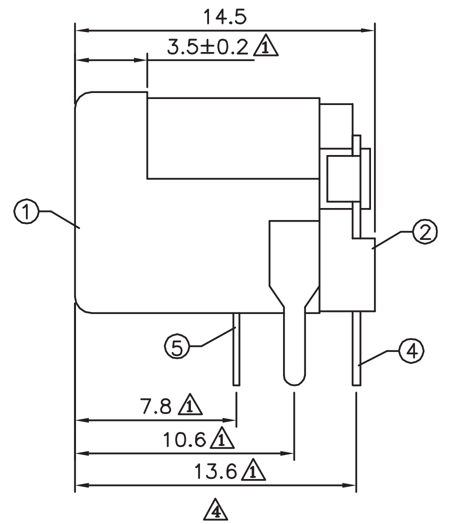
NOTES:
A. All linear dimensions are in inches (millimeters).
B. This drawing is subject to change without notice.
C. Lead dimensions are not controlled within this area.
D. All lead dimensions apply before solder dip.
E. The center lead is in electrical contact with the mounting tab.
F. The chamfer is optional.
G. Thermal pad contour optional within these dimensions.
H. Falls within JEDEC TO-220 variation AB, except minimum lead thickness, minimum exposed pad length, and maximum body length.

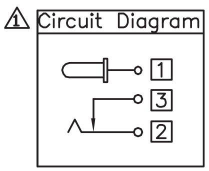
File No.	439	SHEET	1/1

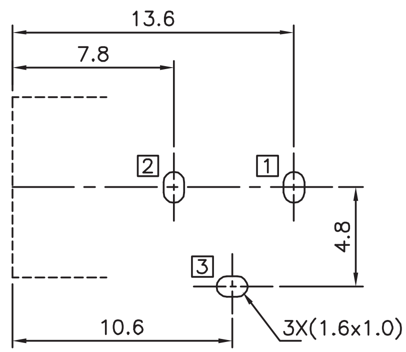
SPECIFICATION

Rating:DC 20V 4A
Contact Resistance:30 Milliohms Max.
Insulation Resistance:100 Megaohms Min. (DC 500V)
Withstand Voltage:500V AC
Insertion And Extraction Force:3~30N
Life Test:5000 Cycles
Operation Temperature Range:−20°C~+70°C
Preservative Temperature Range:−40°C~+85°C
RoHS Compliant

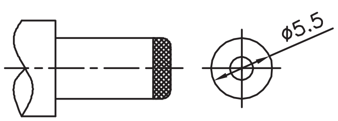








Recommended P.C.B. Layout(Top View)



DC Mate Plug

No.	Description		Q'ty	Material	Remark
6	Shunt Terminal	3	1	Brass	Tin Plated
5	Tip Spring	2	1	Phosphor Bronze	Tin Plated
4	Center Pin Terminal	1	1	Brass	Tin Plated
3	Center Pin	1	1	Brass	CuSn Plated
2	Cover		1	PBT UL94V−0	Black
1	Housing		1	PBT UL94V−0	Black

4UCON 元化興業股份有限公司
4UCON TECHNOLOGY INC.

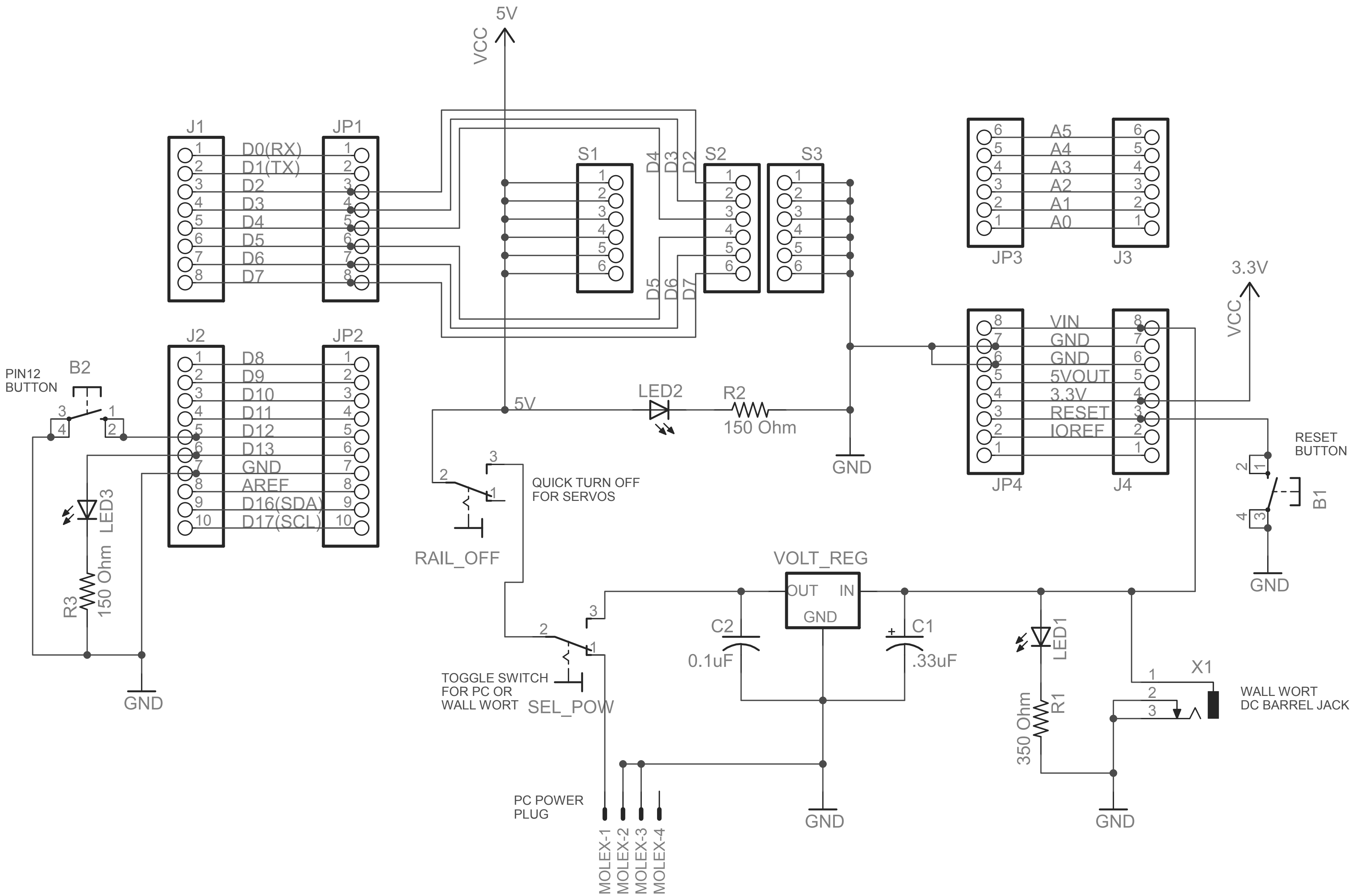
DRAWING No.	18742
SCALE	NONE
APPROVED BY	EMILY
CHECKED BY	HZF
DRAWING BY	WSS
REV.	A4

UNLESS OTHERWISE SPECIFIED TOLERANCES ARE
X. ±0.3
.X ±0.3
ANG. ±2°
UNIT mm

NO.	REV.	REVISIONS	CHK	DATE
4	A4	Modify Data	HZF	03/16/2011
3	A3	Modify Data	HZF	12/02/2010
2	A2	Modify P/N Code	HZF	05/03/2010
1	A1	Modify Data	HZF	04/14/2010
0	A0	NEW RELEASE	HZF	05/01/2007

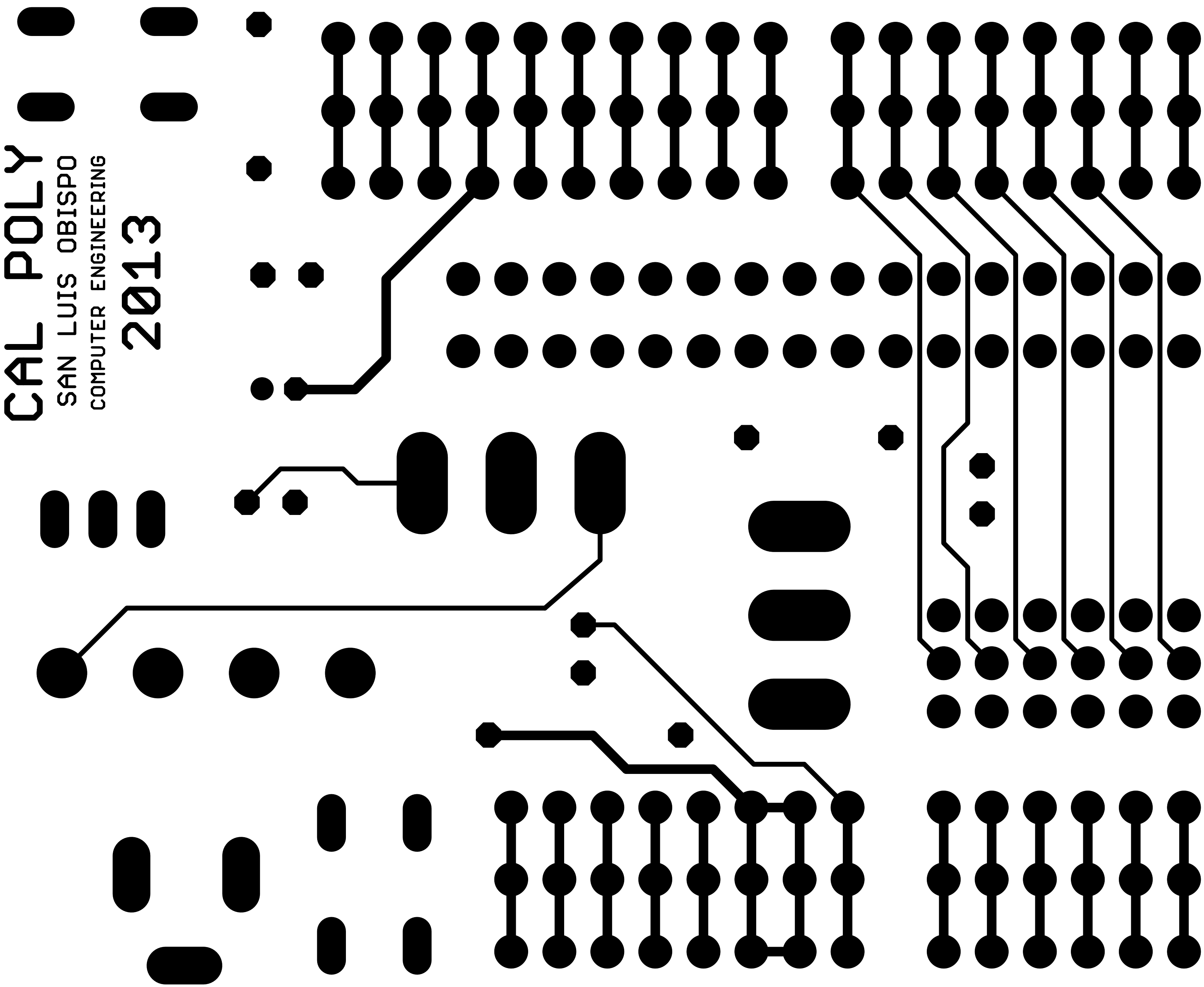
This page intentionally left blank.

SMALL BEAR PROJECT, REV. A
06/05/2013

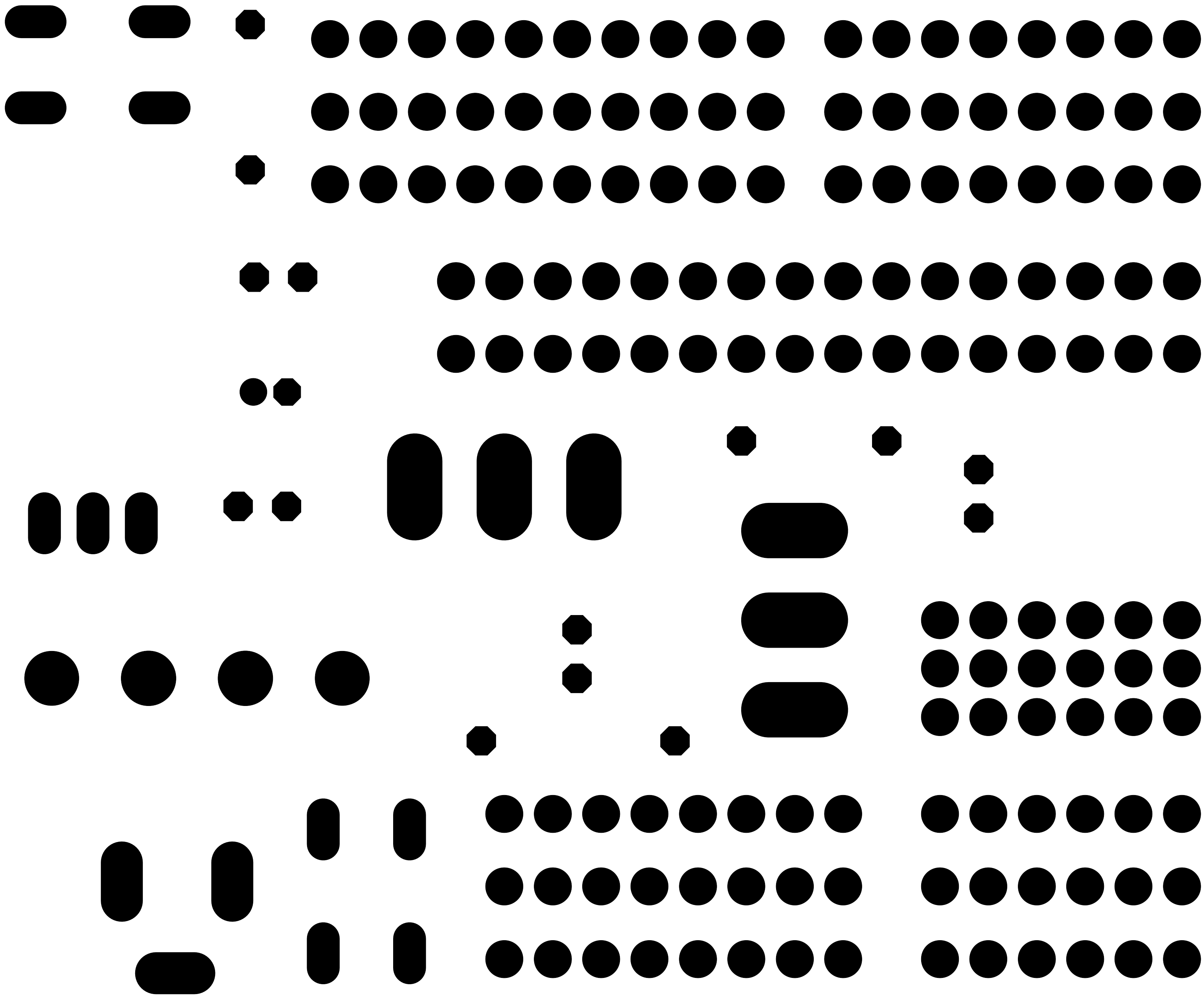


APPENDIX E: GERBER PLOTS

SmallBearProject_RevA.L1

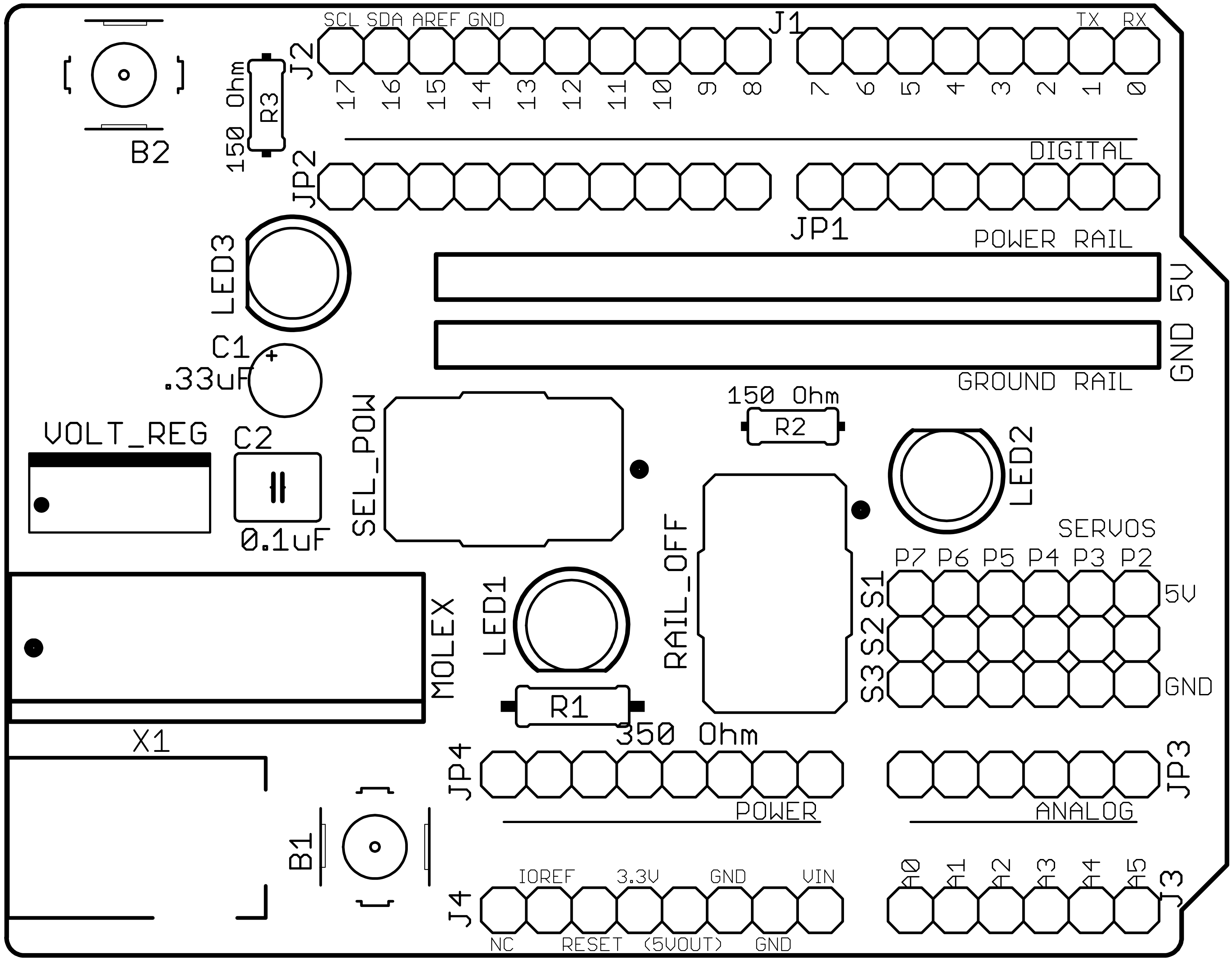


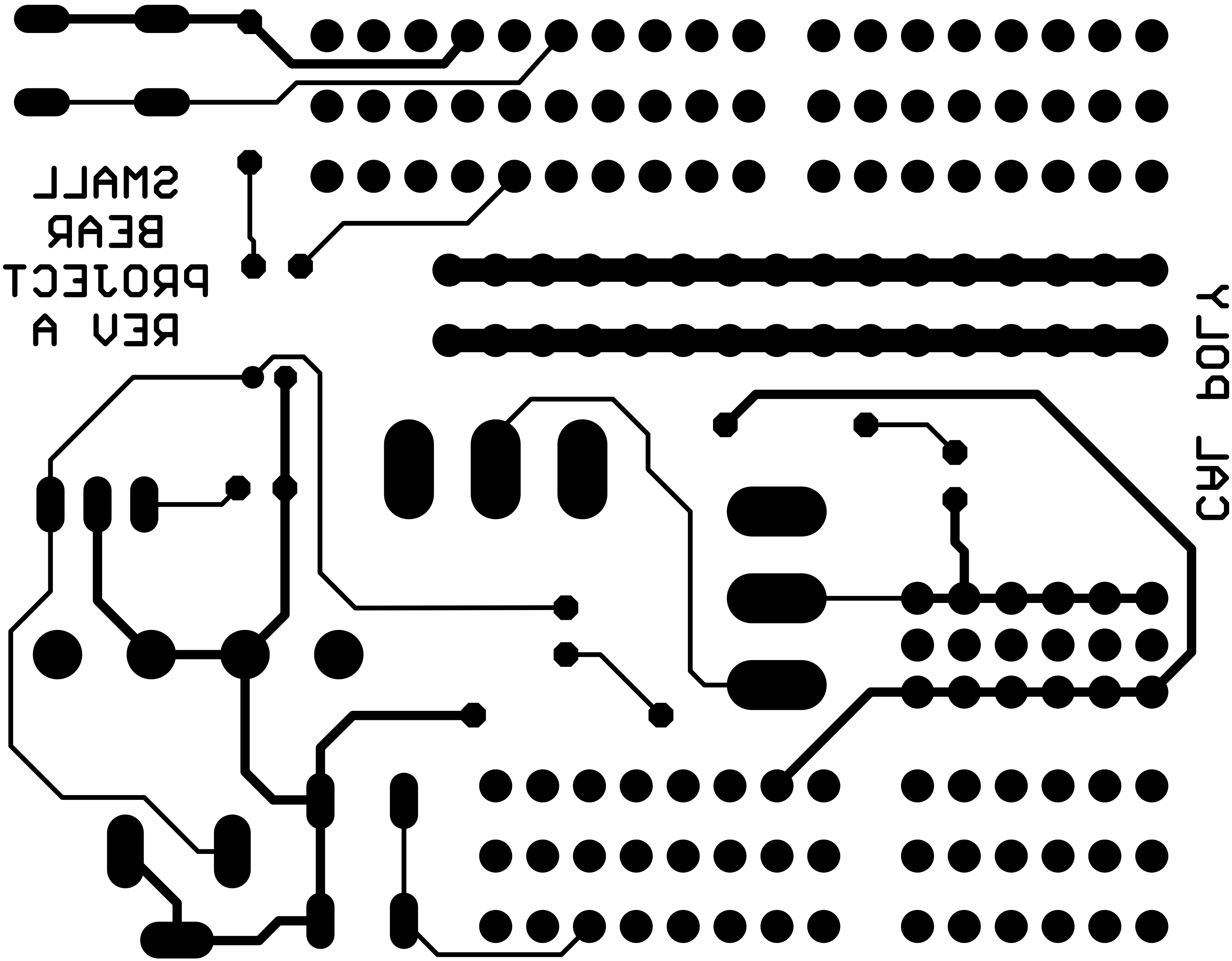
SmallBearProject_RevA.L1M



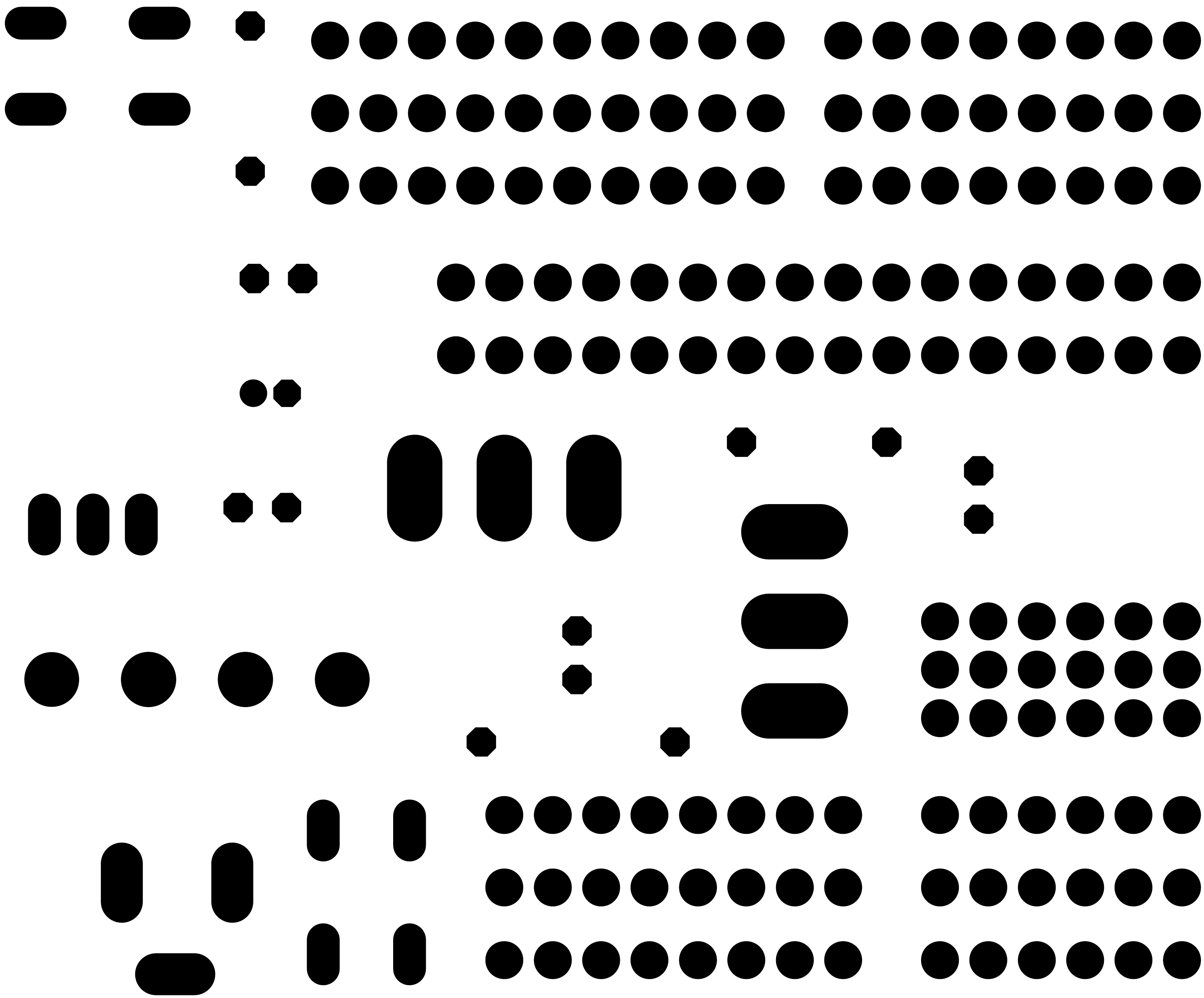
SmallBearProject_RevA.L1S

SMALL BEAR PROJECT, REV. A
06/05/2013

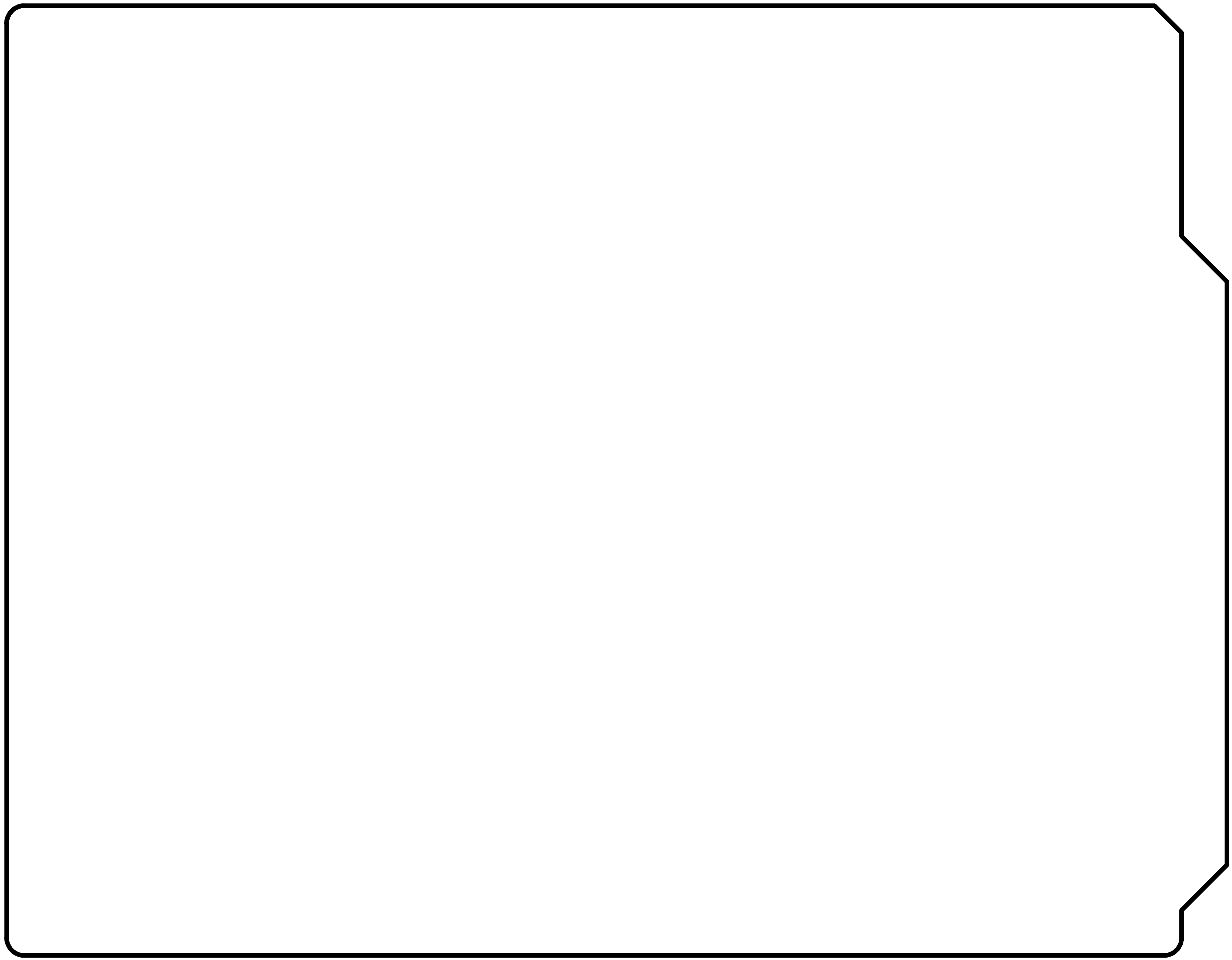



SMALL
BEAR
PROJECT
REV A
CAL POLY

SmallBearProject_RevA.L2M



SmallBearProject_RevA.L2S



This page intentionally left blank.

APPENDIX F - BOARD HOUSE QUOTES								
Board House	**QTY**	**Price Each**	**NRE**	**TOTAL**	**Lead Time**	**Dock Date**	**Notes**	**E-mail Contact**
Custom Circuit Boards	5	$90.00 $71.20	--	$450.00 $356.00	3-days 5-days		RFQ sent 6/4 via website	Tom White <tw@customcircuitboards.com>
Custom Circuit Boards	10	$46.80 $37.50	--	$468.00 $375.00	3-days 5-days		RFQ sent 6/4 via website	Tom White <tw@customcircuitboards.com>
Custom Circuit Boards	~~25~~	$22.00 $18.00	--	$550.00 $450.00	3-days 5-days		RFQ sent 6/4 via website	Tom White <tw@customcircuitboards.com>
OnBoard Circuits	5			$675 $610	3-days 2-days		RFQ sent 6/4 via website	Yvonne Banovich <yvonne@onboardcircuits.com>
Photo Fabricators Inc.							RFQ sent 6/4 via website email (broken link, gerber no upload)	
Fineline Circuits & Technology	5			$ 500 $ 400	3-days 6-days		RFQ sent 6/4 via website (no gbrs)	R. Bajaria <rbajaria@finelinecircuits.com>
Printed Circuit Solutions, Inc. (Select Circuits)							RFQ sent 6/4 via website	
American Circuit Technology	5			$450 $600	5-days 3-days	6/11/13 6/7/13	RFQ sent 6/4 via email attach (quotes@actpcb.com)	Andy Butani <andy@actpcb.com>
PCB Fab Express	5	$31.04	Included	$155.20	3-days	Starting Wed, 6/10	RFQ online	
Sierra Proto Express	5	$126.07	Included	$630.35	3-days		RFQ online	
Advanced Assembly	5	--	--	$1,020.80 $964.20	3-days 5-days		RFQ sent 6/4 via website upload; 6/4 nope,	Chet Williams <chet@aapcb.com>
Advanced Assembly	5	--	--	$245 + $101.50 shipping	5-days	6/11/2013	6/5 new quote, Chet is willing to work with	Chet Williams <chet@aapcb.com>
Advanced Assembly	5	--	--	$150 + $101.50 shipping	3-days	6/11/2013		Chet Williams <chet@aapcb.com>
Electronic Interconnect	5	$131.67	Included	$658.35	2-day		RFQ online	

ANALYSIS OF THE SENIOR PROJECT DESIGN

Project Title: Small Bear Project Shield
Student: Janice A. Gelacio
Advisor: Dr. Hugh Smith

Quarter/Year Submitted: Spring 2013
Date: 14 June 2013

- **Summary of Functional Requirements**
  The Small Bear Project Shield is an upgrade of the Power Control Board used in the Small Bear Project class, which utilizes the printed circuit board technology for a smaller structure. It has a connector that receives a 9V power through a wall wart, converted to 5V with a voltage regulator. It also has a connector to plug in a computer power supply. The power source is selected with a switch. The assembled board is used to power a stuffed animal bear that is built and programmed in the class.

- **Primary Constraints**
  The main challenge was to learn the EAGLE software. Maneuvering through the schematic, board, and library editor consumed a majority of the time allotted for the design. Another challenge was purchasing the boards and components. Time and cost were the limiting factor. Time was very limited upon approval of the design. Finding the right boardhouse that can manufacture the board with a quick turn at a low cost proved to be challenging. The same goes with ordering the components.

- **Economic**
  Original estimated cost for board and component parts: approx. $300
  Actual final cost of board and component parts: $280.35 for 5 pcbs, components, and shipping
  Bill of materials for component parts is included in the report.
  Additional costs: $34.99 Arduino and $4.95 Prototype Shield used for reference.
  All software used are freeware.
  Original estimated development time: one quarter
  Actual development time: one quarter for the final assembly build. Time for testing the shield with the Arduino is variable.

- **If manufactured on a commercial basis**: Not Applicable

- **Environmental**
  Components are attached on the board by means soldering. Solder contains tin and lead. Lead is a chemical known to be toxic. Any scrapped components can also have an environmental impact if not disposed of accordingly.

- **Manufacturability**
  Purchasing: It's a challenge to find the right boardshop that can produce a good quality board at a low price, especially on the first run. For future builds, availability of components will be a challenge.
  Assembly: Soldering the components can be difficult for anyone who has not soldered before.

- **Sustainability**
  Maintaining: The shield contains components that are electrostatic-sensitive device (ESD). Proper care for storage and handling should be followed accordingly to avoid damage.
  Project impacts the sustainable use of resources: uses many components for the assembly
  Upgrades: Smaller component can be used as an alternative in order to keep the height to a minimum.
  Issues with upgrading: Changing components can require a redesign of the component footprint

- **Ethical**
  The only ethical issue relating to the design is disposal. Electronic equipment should be disposed of appropriately in order to avoid toxic waste to the environment.

- **Health and Safety**
  During assembly, soldering produces fumes that can cause risk to one's health. Students assembling the boards should be in an environment with good ventilation. Students shall wear protective eyewear. Soldering irons is hot and proper soldering etiquette should be followed.

- **Social and Political**: Not applicable.

- **Developmental**
  Discovering the open-source software, especially the EAGLE software, expanded my vision for future hobby projects. The availability of resources on the EAGLE website was very valuable. And learning beyond the process of design, extending to purchasing and manufacturing, increased my appreciation for those other departments.