Over-Current Protection Reference Design & Study

ECE 480 Design Team 5

Final Report

Kenji Aono

Stephen England

Ryan Laderach

Joshua Myers

Facilitator: Dr. Wen Li

Sponsor Representative: Mr. Pete Semig

Report Submitted: April 26, 2011

Executive Summary

The purpose of this project is to design, test, and document multiple real-world over-current protection application circuits. The designs will focus on placing Texas Instrument (TI) components on a printed circuit board to demonstrate the application of theory. The project represents a complete engineering design cycle, from receiving specifications to final documentation of findings, and qualifies for participation in the Texas Instruments Analog Design Competition.

Two applications of over-current protection have been provided by TI. The first application is to measure the current being drawn by a tablet PC and shutdown current to the load if an excess amount is drawn. The second application is to measure the current being draw by a cell phone to monitor the power consumption and, therefore, understand how battery life is affected.

In addition to the two applications, TI requested various studies to display how PCB traces affect the final output of current sensing ICs. Design Team 5 successfully completed four separate study cases, determined by recommendations from TI.

Acknowledgements

Design Team 5 would like to thank our facilitator, Dr. Wen Li, for all of her help and support throughout the course of this project. Design Team 5 would also like to thank Mr. Pete Semig, the sponsor representative from Texas Instruments, for all the help with understanding the impact of the project as well as providing the team with any necessary materials. Design Team 5 would like to thank Gregg Mulder, Bryan Wright and Roxanne Peacock for helping the team acquire parts and equipment from the ECE shop and with orders to other suppliers. Design Team 5 would like to thank Raoul Ouedraogo and Junyan Tang for extending their time, space, and equipment to teach the team how to fabricate a PCB using chemical etching. Finally, Design Team 5 would like to thank CadSoft for proving a free copy of EAGLE for educational use.

Table Of Contents

Chapter 1: Introduction and Project Background

1.1 Introduction

The use of overcurrent protection (OCP) is a common practice in designing electrical circuits, and several common methods currently exist, such as; circuit breakers, fuses, and ground fault circuit interrupts. However, these traditional methods do not meet the design criteria for portable, low-power devices that may be sensitive to even the slightest (dozens of mA) level of overcurrent. These devices are especially sensitive to overcurrent, and could become inoperable with even the smallest amount of overcurrent. Although OCP devices are designed to restrict the excessive flow of current, most of the traditional methods cannot effectively detect the low current levels in portable electronics in order to remove power before circuit components are damaged. The solution to this problem is to use integrated circuit systems, which can detect current levels in the micro amp range, and make logical decisions to control a switch that can respond several orders of magnitude faster than mechanical systems. In addition to the fast system response time, the integrated systems will not suffer from wear and can be used to protect a circuit multiple times without service, unlike a fuse.

Texas Instruments (TI) is a company that specializes in various electrical systems, and is in need of a reference design for electrical OCP systems. TI has requested that two different applications be designed and implemented for a tablet PC and cellphone OCP system. For the tablet PC application, as soon as any excess current draw is detected, power to the system is removed and the system is shutdown. In the cellphone application, the load current of the cellphone will be accurately monitored, in order to report power consumption of the system. For both designs, select TI components from a large portfolio will be utilized, including the MSP430 microcontroller, operational amplifiers, and current shunt monitors.

1.2 Current Sensing Background Information

Several current sensing methods are in existence today: R_{ds} MOSFET sensing, Hall-effect, current transformers, resistive shunts, and other more exotic methods. Choosing an appropriate method for the given design criteria requires some knowledge on how each procedure may affect system performance.

R_{ds} MOSFET sensing uses the characteristic impedance between the drain and source of the MOSFET (if used as a switch) in a power supply and the voltage at the switch to infer the current flowing through the circuit. This method is not useful in solving the giving design requirements because it requires a MOSFET switch, which would not exist when running the current sensing application using a battery. [1]

Hall-effect current sensing is a common solution, which does not suffer from insertion losses that reduce useful battery life. The main concept is to focus a flux field using a toroid such that current flow through a conductor placed within the toroid induces a magnetic field. By observing the generated magnetic flux density, one may figure the amount of current flow. However, this method is undesirable because of the increased PCB real estate usage and increased cost for the components required by this process. [2]

Current transformers, like Hall-effect current sensing, allow for circuit isolation and zero insertion losses. This method also has zero offset voltage and requires no external power. The concept is similar to how a transformer operates, that is; running current through an inductor (winding) allows for coupling to another inductor, with induced current flow proportional to the number of turns in each inductor. Also like the Hall-effect method, current transformers are expensive and require large amounts of PCB real estate — perhaps most debilitating is that this method requires AC current and the design specifications call for current sensing on batteries, which are inherently DC. [2]

Resistive shunts require placing a resistance in series with the power source and the load, and measuring the voltage drop across said resistance to calculate the current flowing through the circuit. This method is low cost, although it suffers from insertion loss and may require signal amplification of the measured voltage. Two distinct uses of the resistive shunt exist; low side current sensing and high side current sensing. Low side current sensing places the resistive shunt between the load and the ground, which allows for easy implementation of current sensing using nothing but an operational amplifier. This also allows the current sensing system to avoid causing insertion loss. However, it also adds impedance to the load's line to ground, which may affect system performance and will not detect faults in the circuit until the current has already reached the load. High side current sensing, on the other hand, places its resistive shunt between the power source and the load, which does not add a disturbance to ground and allows faults to be sensed prior to the current reaching the load, and necessary action to be triggered in time. Both methods can be designed for fast, precise, and accurate operation, factors important to meeting the design requirements. Due to low cost, low PCB real estate requirements, and performance capabilities, this method is chosen for this project. [3]

Chapter 2: Exploring Solutions and Selecting Approach

2.1 Design Specifications

The following will show the exact requirements requested by Texas Instruments (TI) for each application.

2.1.1 Application One

The purpose of this application is to create a reference design that will demonstrate the use of components offered by TI in an emergency shutdown situation in a tablet-PC. The over-current protection (OCP) system will shut off power to the system if the current drawn reaches 1 A. The system should be as low power as possible, small size, low cost, and at highest priority a fast speed of shutoff. This project was to be completed assuming the tablet battery is a 3.6 V, 6.75 A-hr Li-Ion battery.

2.1.2 Application Two

The purpose of this application is to create a reference design that will accurately monitor the current delivered to a cell phone using components from TI. The supply voltage will be a 3V battery and the load will require 2.7 V-3.3 V. Accuracy from 7 mA to 192.5 mA is necessary for successfully completing the project. An LCD will be used to display the current being drawn from the system, and will be driven by the MSP430 microcontroller.

2.2 FAST Diagrams

For application one, the FAST diagram shows that many components are necessary in the design to reach the primary function. The list of components for this project are very expansive and making the correct selections made this process very difficult, so only the more general components of the overall design solution are shown. Many of the components selected fit into the design specifications provided by the customer, which made some decisions clearer than others. From these options, more general parts were selected, which is shown in detail in chapter 3. The FAST diagram for application one is shown in Figure 1.

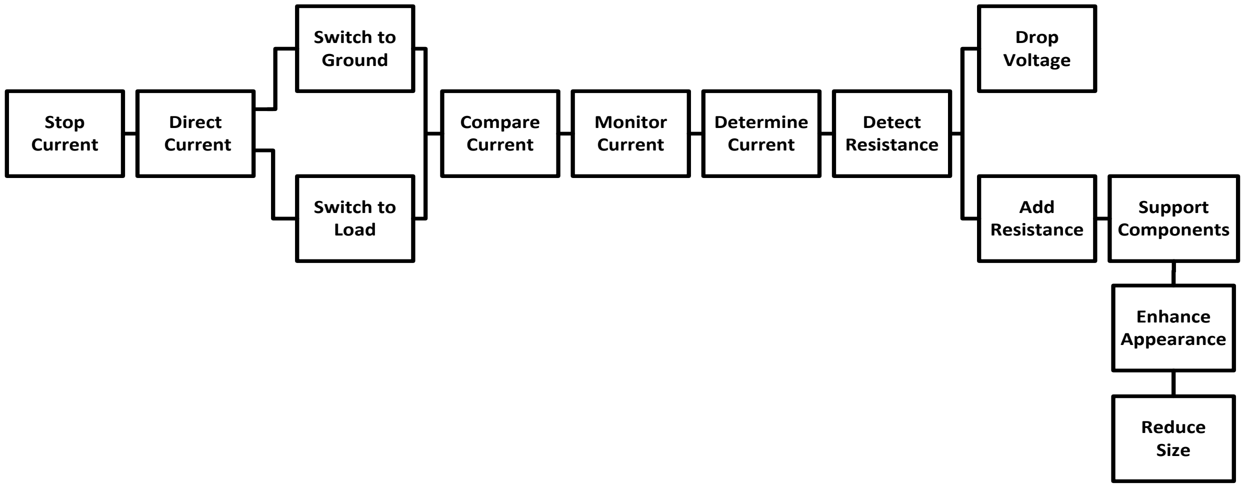


Figure 1: Application One FAST Diagram

For application two, the FAST diagram is much less complex due to the limited number of components needed to achieve the design solution. This makes component selection in the upcoming house of qualities unnecessary. The FAST diagram for application two is shown in Figure 2.

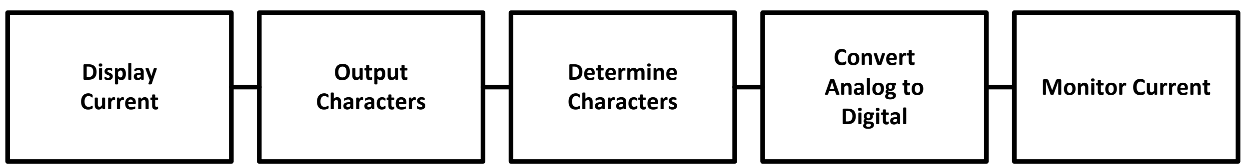


Figure 2: Application Two FAST Diagram

2.3 House of Quality

The house of quality created for this project was specifically created for application one, because application two did not require any additional parts selection beyond the design specification provided by the sponsor. The house of quality is used as a solution matrix, allowing the design team to find the easiest path to the final design solution. The completed house of quality is shown below in Figure 1.

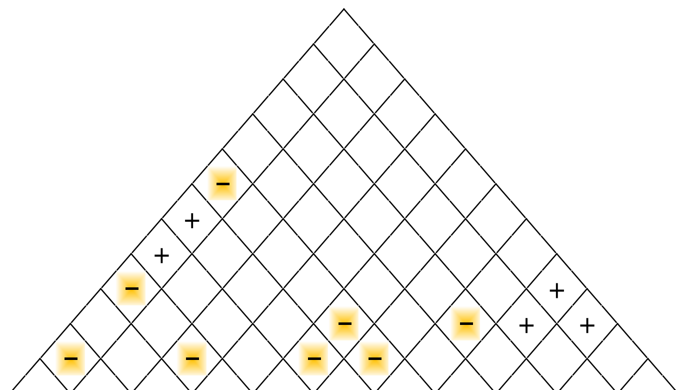
QFD: House of Quality

Project: Texas Instruments Over Current Protection
Revision: 001
Date: 25 Apr. 2011

Correlations	
Positive	+
Negative	−
No Correlation	

Relationships	
Strong	●
Moderate	○
Weak	▽

Direction of Improvement	
Maximize	▲
Target	◇
Minimize	▼



Row #	Weight Chart	Relative Weight	Customer Importance	Maximum Relationship	Customer Requirements (Explicit and Implicit) / Functional Requirements	Current Shunt Monitor	Op Amp	Voltage Reference	Voltage Divider	Professional PCB	Self Made PCB	Proto-board	Load Switch IC	MOSFET Switch	Latch	Flip Flop
					Column #	1	2	3	4	5	6	7	8	9	10	11
					Direction of Improvement	▲	▼	▲	▼	▼	▲	▼	▼	◇	◇	◇
1		21%	4	9	Low Cost	●	●	●	●	▽	●	○	○	○	●	●
2		26%	5	9	Low Power	●	●	●			○	▽	○	○		
3		16%	3	9	Fast Speed								▽	●		
4		16%	3	9	Small Size	●	○	●	○	●	●	▽	●	○	○	○
5		11%	2	9	Accurate	●	▽			●	○	▽				
6		11%	2	9	Working prototype										●	●
					Max Relationship	9	9	9	9	9	9	3	9	9	9	9
					Technical Importance Rating	663.16	484.21	568.42	236.84	257.89	442.11	115.79	300	331.58	331.58	331.58
					Relative Weight	16%	12%	14%	6%	6%	11%	3%	7%	8%	8%	8%
					Weight Chart											

Figure 3: House of Quality

2.4 Gantt Chart and Project Schedule

The Gantt chart is a very useful tool in maintaining a proactive schedule to reach the goal of this project, which is to deliver a working product. The resources section of the chart was utilized for the technical aspects of the course; presentations and assignments are not included in the resources due to them being off the critical path. The initial Gantt chart for our project is shown in Figure 4.

ID	Task Name	Duration	Start	Finish	Predecessors	Resource Names
1	**ECE 480 Design Project**	**78 days**	**Fri 1/14/11**	**Mon 5/2/11**		
2	**Design Team Meeting**	**73 days**	**Thu 1/20/11**	**Thu 4/28/11**		
17	Initial Meeting with Sponsor	0 days	Mon 1/24/11	Mon 1/24/11		Josh,Kenji,Ryan,Steve
18	Six Sigma Homework	3 days	Fri 1/28/11	Mon 1/31/11		Josh[25%],Kenji[25%],Ryan[25%],Steve[25%]
19	Preproposal	5 days	Wed 2/2/11	Mon 2/7/11		Josh[25%],Kenji[25%],Ryan[25%],Steve[25%]
20	Initial Website Design	4 days	Tue 2/8/11	Fri 2/11/11		Josh[25%],Kenji[25%]
21	~~Meeting with Sponsor~~	~~0 days~~	~~Wed 2/2/11~~	~~Wed 2/2/11~~		~~Josh,Kenji,Ryan,Steve~~
22	Meeting with Sponsor	0 days	Fri 2/11/11	Fri 2/11/11		Josh[25%],Kenji[25%],Ryan[25%],Steve[25%]
23	Part List	0 days	Tue 2/8/11	Tue 2/8/11		Kenji,Steve,Josh[25%],Ryan
24	Order Parts	0 days	Mon 2/14/11	Mon 2/14/11	23	Kenji[25%],Steve[25%]
25	**Application One**	**27.13 days**	**Mon 2/7/11**	**Tue 3/15/11**		
26	Test PCB Design and Fabrication	5 days	Mon 2/14/11	Thu 2/17/11	27	Josh[25%],Kenji[25%],Ryan[25%],Steve[25%]
27	Simulations for App 1	3 days	Mon 2/7/11	Wed 2/9/11		Steve[25%]
28	Build Prototype App 1	5 days	Fri 2/18/11	Thu 2/24/11	26	Steve[25%]
29	Build Test Fixture For App 1	5 days	Fri 2/11/11	Fri 2/18/11		Josh[25%]
30	Testing for Prototype App 1	16 days	Thu 2/24/11	Wed 3/2/11	28,29	Kenji[25%],Ryan[25%]
31	Adjustments to Prototype	20 days	Wed 3/2/11	Tue 3/8/11	30	Steve[25%]
32	PCB Design App 1	20 days	Tue 3/8/11	Tue 3/15/11	31	Josh[25%],Steve
33	**Application Two**	**40 days**	**Mon 2/14/11**	**Wed 4/6/11**		
40	Proposal Presentation	0 days	Mon 3/14/11	Mon 3/14/11		Josh[25%],Kenji[25%],Ryan[25%],Steve[25%]
41	Test Final PCBs	10 days	Fri 3/25/11	Wed 4/6/11	32,39	Josh[25%],Ryan[25%]
42	Technical Lecture Preparation	6 days	Wed 3/23/11	Thu 3/31/11		Josh[25%],Kenji[25%],Ryan[25%],Steve[25%]
43	Technical Lecture	0 days	Fri 4/1/11	Fri 4/1/11	42	Josh,Kenji,Ryan,Steve
44	PCB & Design Changes	12 days	Thu 4/7/11	Thu 4/21/11	41	Josh[50%],Kenji[50%],Ryan[50%],Steve[50%]
45	Compile Data for Study	24 days	Thu 3/24/11	Tue 4/26/11		Josh[50%],Kenji[50%],Ryan[50%],Steve[50%]
46	Design Day	0 days	Fri 4/29/11	Fri 4/29/11		Josh,Kenji,Ryan,Steve
47	Final Submission	0 days	Mon 5/2/11	Mon 5/2/11		

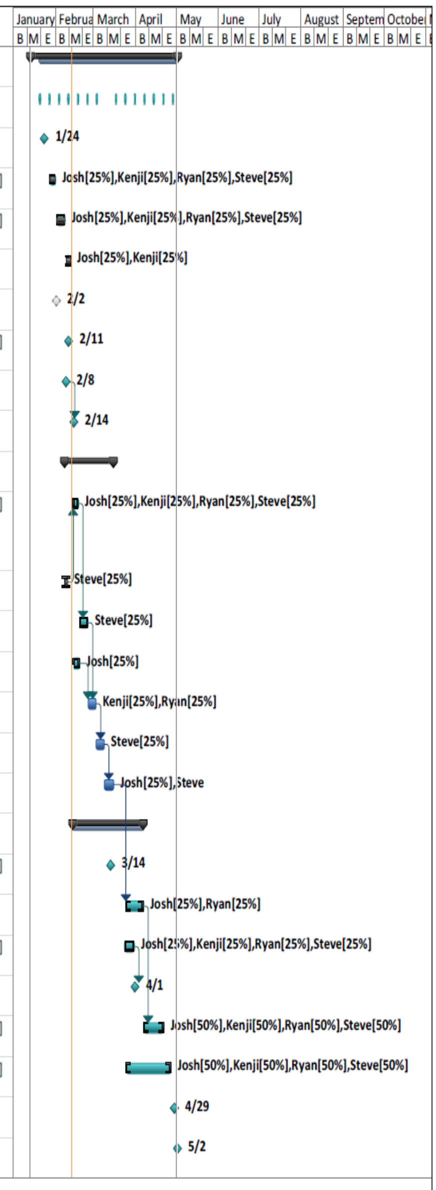


Figure 4: Initial Gantt Chart

Due to scheduling conflicts and possibly a slightly too optimistic schedule, Design Team 5 had a difficult time following the schedule of the initial Gantt chart. However, the Gantt chart was revised to meet the final schedule that was followed. The final Gantt chart that was followed through the end of this project is shown in Figure 5.

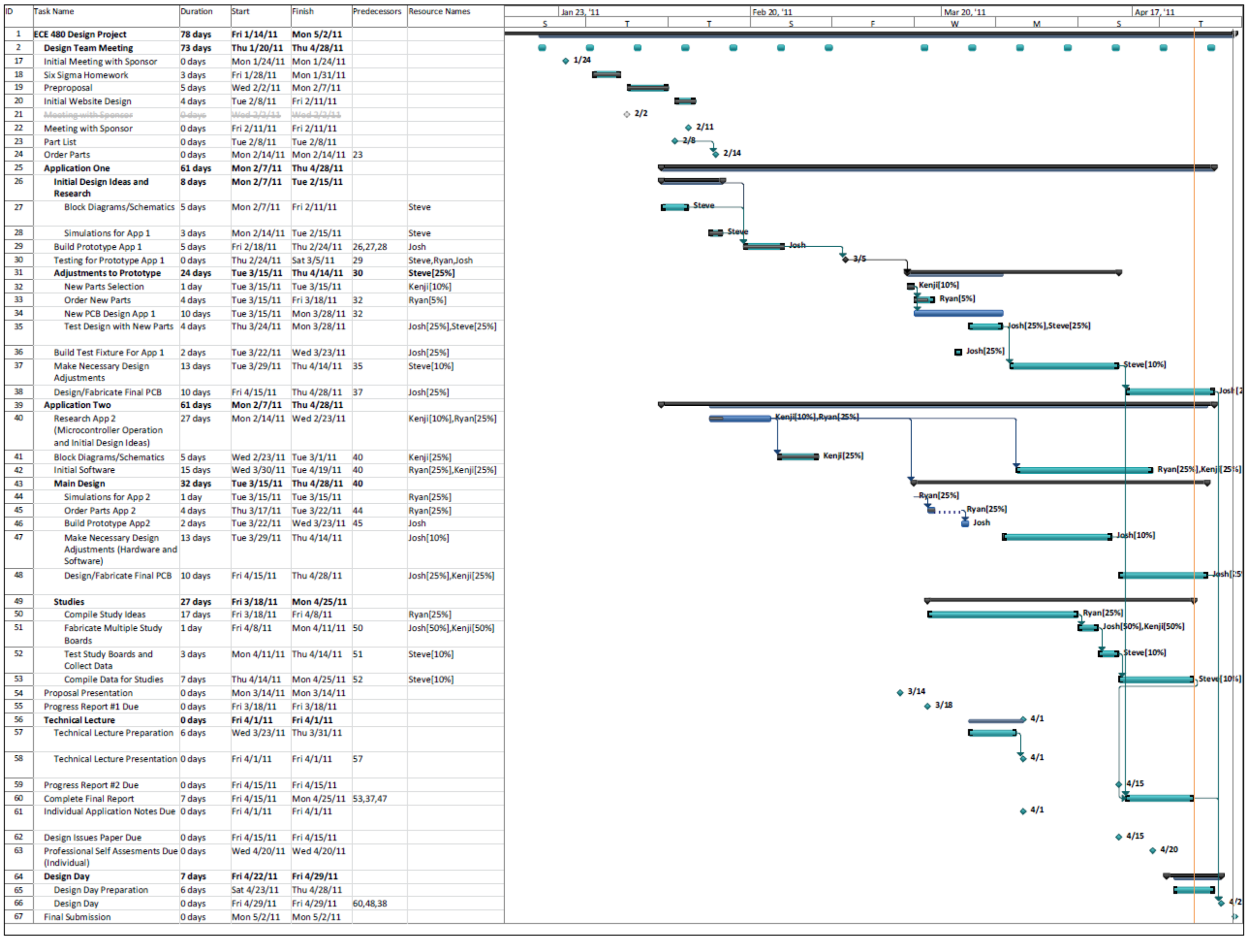


Figure 5: Final Gantt Chart

2.5 Initial Budget

Since the project is split into two separate applications, the budget will be shown for each application separately. Although application one uses more parts, the cost is lower than application two. This is due to the high price of the LCD and the instrumental amplifier INA333. The initial budget, with an estimated cost for PCB fabrication, is shown in Table 1.

Table 1: Initial Budget

Item	Projected Cost Per Unit	Current Cost
INA138 Current Shunt Monitor	$3.77	Sampled
INA333 Instrumentation Amplifier	$5.40	Sampled
ADS1113 Analog to Digital Converter	$6.30	Sampled
Professional PCB Fabrication	$250.00 (total estimate)	TBD
TLV3491 Comparator	$0.56	Sampled
Current Shunt Resistors	$1.27	$3.81
TPS2033	$1.61	Sampled
MSP430 Target Board/Development Tools	$149.00	Sampled
Total Cost	$417.75	TBD
Remaining Budget	$82.25	TBD

2.6 Design Solution and Justification

Using the house of quality as a solution selection matrix, the design solution for application one uses components to match customer specifications, such as; a current shunt monitor, voltage reference, MOSFET switches, and PCBs that will be made by members of the group. This includes the lowest cost without sacrificing other customer requirements. The design solution for application two includes a MSP430 microprocessor and an instrumental amplifier which is a more accurate current shunt monitor. The MSP430 is a 2xx series that will decrease the cost substantially. Both solutions will use a resistor with a low resistance in the 10-25 mΩ range. A third solution to application one will also be explored attempting to replace some of the components with the faster, cheaper MSP430 microcontroller to reduce the number of parts.

This solution to application one is unique in that every avenue of the customer's specifications will be delivered upon while attempting to explore both digital and analog solutions to application one. Application two is less unique because the only solution that drives and LCD and monitors current draw is that of a microcontroller. These solutions are likely to be successful for a number of reasons. The analog solution to application one is well documented and most processes of the solution can be simulated using TI-TINA Spice, which gives us the possibility to discover some errors before a final prototype is fabricated. This solution is also likely to work because the design is based off of well-known analog circuit theory, which has been used for many years. Application two is simply running an LCD from a microcontroller and using an instrumentation amplifier to amplify a current signal. This concept is simple, and has documented solutions by other engineers, but the solution is non-trivial because of the application requires precise measurements. The team's initial simulations and prototyping have also shown that these solutions will be viable, and deliverable to the customer.

Chapter 3: Technical Description of Project

3.1 Application One Design

The purpose of the first application is to use resistive shunt current sensing for over-current protection in a tablet PC. If current being drawn to the load reaches 1A, connection to the battery will be cut, preventing damages to the device.

Design Team 5 attempted to accomplish the desired operation using hardware components exclusively. Basically, the operation of the circuit is to sense current using a current shunt monitor connected using high side topology to output a voltage signal based on the current being drawn to the load. A reference voltage would be created that is a lower voltage than the battery to compare the output of the current shunt monitor to. A comparator would then be used to compare the output of the current shunt monitor to the voltage reference. The current shunt monitor would be set up to output a value close to the voltage reference at the switching current, which is 1 A in this case. When the voltage output of the current shunt monitor exceeds that of the reference, the comparator will change its output voltage and trigger necessary action. The switch will then cut power to the load when it receives input from the comparator. The following is a block diagram of the basic idea of our project:

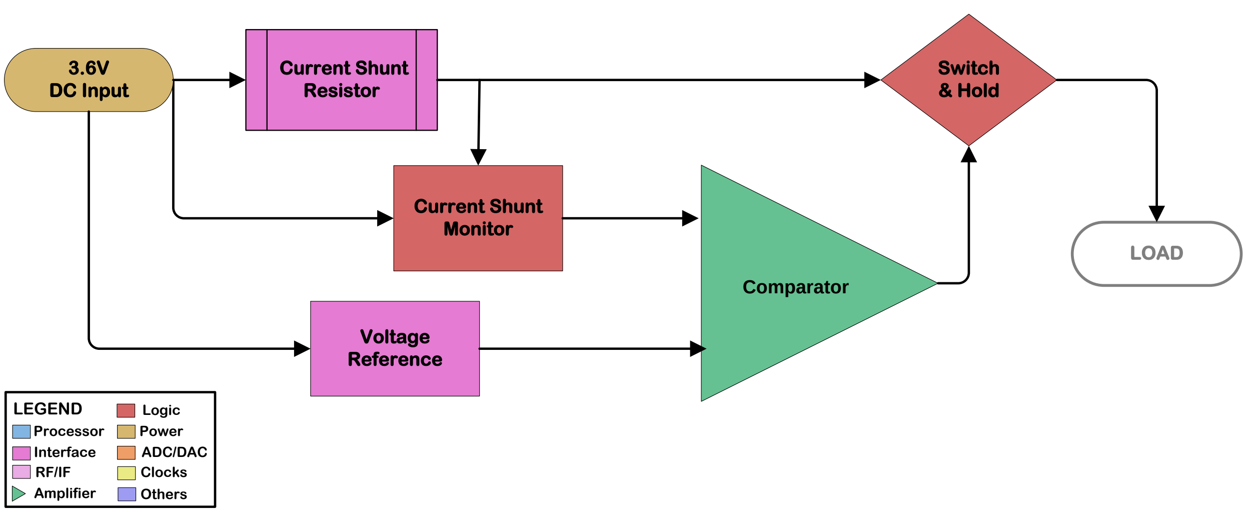


Figure 6: Application One Functional Block Diagram

In order to do this, a current shunt monitor, comparator, and switch were known to be needed from the beginning. As testing was completed, parts were changed and added as needed. The design can basically be split into three basic sections: current sensing, comparing and decision making, and switching.

3.1.1 Current Sensing

There are basically two components required for the current sensing part of the design, the current shunt monitor and the current shunt resistor. The current shunt resistor must be very small to reduce insertion loss but it also must be large enough to account for the finite gain of the current shunt monitor. Design team 5 considered many of the current shunt monitors in Texas Instrument's portfolio. Many specifications had to be considered, especially those that were related to the specifications given by the sponsor. The following is a table that includes information about various current shunt monitors and the differences between them.

Table 2: Current Shunt Monitors Selection

Part Number	Output Signal	Quiescent Current	Size	Cost	Accuracy (V_{IO})	Additional Note
INA138	Current $V_o = \frac{I_S R_S R_L}{5\ k\Omega}$	25 µA	SOT-23 (Footprint: 3.05mm X 3.0mm)	$2.63 + External Resistor Required	1 mV	
INA195	Voltage 100 V/V Gain	250 µA	SOT-23	$3.33	2 mV	High Bandwidth
INA202	Voltage 100 V/V Gain	1.35 mA	SOIC (Footprint: 3.91mm X 4.9mm)	$1.71-Cost of external comparator	2.5 mV	Includes internal comparator
INA214	Voltage 100 V/V Gain	65 µA	SC-70 (Footprint: 2.6mm X 2.4mm)	$2.63	60 µV	May be used in bi-directional measurements
INA216A3	Voltage 100 V/V Gain	25 µA	DSBGA (Footprint: 0.8mm X 0.8mm)	$1.75	75 µV	

The top three current shunt monitors considered were the INA138, INA214, and the INA216A3. The INA138 consumes the least amount of power, due to the very low quiescent current. It also is very flexible since it is a current output device so the output gain can be changed based on the external resistor added. The INA216A3 also draws the least amount of power and has the smallest size and cost. The input offset voltage is also much better than the INA138.

However, even though small size was a priority provided by Texas Instruments, this part was too small to test with the equipment provided in the ECE labs. Therefore, the part selected for the final design is the INA214. This part consumes slightly more power than the other parts considered, but it is a very small size yet large enough to solder by hand, low cost, and significantly more accurate than many of the other current shunt monitors offered by Texas Instruments.

Matching the current shunt monitor's gain to the value of the current shunt resistor is also crucial to avoid clipping of the output of the shunt monitor and skewing the final results. If the resistor is too large, the gain of the monitor will cause the output to clip. If the resistor is too small, the drop across the resistor will be difficult for the current shunt monitor to get an accurate reading due to the offset voltage and common mode gain restraints. The following TINA-TI spice simulations show the different effects of the output caused by shunt resistor as well as the gain change in the INA138 current output case (shown in Figure 35 & Figure 36).

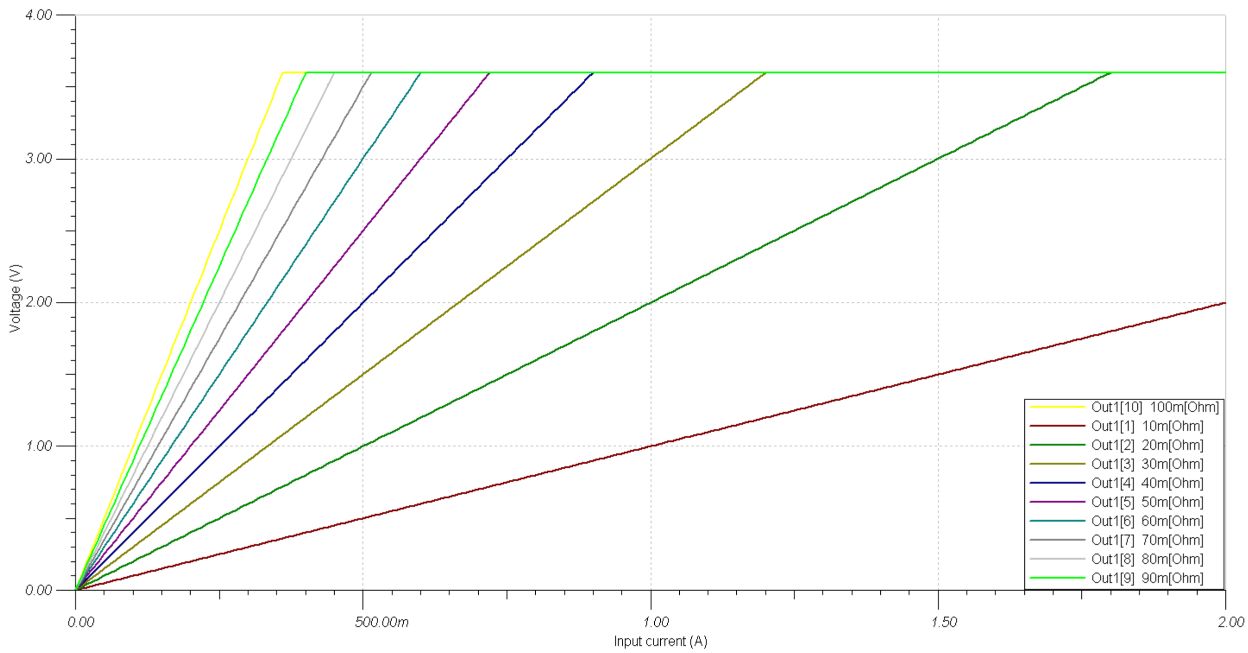


Figure 7: Simulation - Sweep of the Current and Shunt Resistance

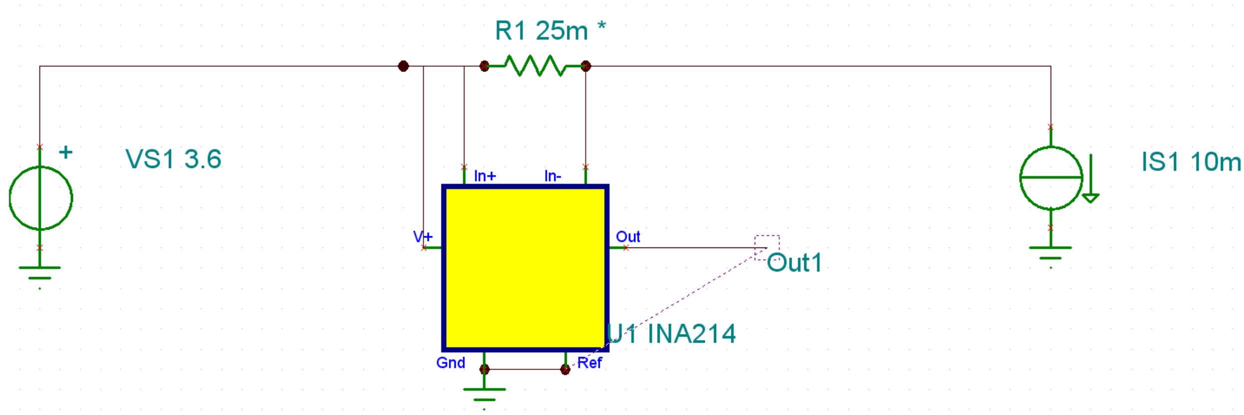


Figure 8: TINA-TI Schematic Used for Figure 7

Using an INA214, which has a gain of 100 V/V, shows that if the current shunt resistor is higher than 70 mΩ, the shunt monitor will hit the rails at 1 Aif running on the 3.6 V given in the customer specifications. Team 5 decided to use a 25 mΩ resistor as the sense resistor for most of the design process.

3.1.2 Comparing

Within the comparing section of the system design, there are two basic parts, creating a reference to compare against and using an IC that compares the current shunt monitor output to the voltage reference. In the original design, a voltage divider using two resistors was used to divide the battery voltage by two and compare that to the output of an INA138, which output 1.8 V at 1 A. However, it was determined that the voltage divider is not the optimal way to create a reference since it draws current and, therefore, consumes power. This does not meet the specifications provided by the sponsor. The voltage divider would also cause problems when the battery is not running at full power. The reference would decrease, lowering the reference voltage the current shunt monitor output is being compared to. This could possibly cause the system to cut power to the tablet-PC at an unexpected time. To avoid this issue, the voltage divider was changed to a TI REF3325 voltage reference, which takes in a voltage within the range of 2.7 V to 5.5 V and outputs a voltage of 2.5 V. This can be compared, using a single comparator, to a current shunt monitor output, which would be 2.5 V with a gain of 100 at 1 A with a 25 mΩ shunt resistor. The voltage reference consumes much less power than the voltage divider, and is much smaller, coming in SC-70 package.

When the current shunt monitor is outputting higher than the reference, more than 1A is being drawn and the comparator can enable and disable the switch as needed. The output of the comparator can either be a logic high (15 V) or low (0 V). Design Team 5 considered two comparators throughout the design process, the TI TLV3491 and the TI TLV3501. The following table shows the specifications of the two comparators:

Table 3: Comparator Selection

Part Number	Output	Size	Output Delay	Rise/Fall Times	Quiescent Current
TLV3491	Push-Pull	SOT-23	13.5 µs	100 ns	1.8 µA
TLV3501	Rail-to-Rail	SOT-23	4.5 ns	1.5 ns	3.2 mA

The TLV3491 draws much less power than the TLV3501, however, the TLV3501 is significantly faster. In the designs using an analog comparator, the TLV3501 is used. Speed was determined to be the more important factor when choosing a comparator for this application so the power could be switched before too much current is drawn to the load and damage is done to the tablet PC.

The following simulation results display how the comparing part of the circuit can be implemented with the output of the current shunt monitor. The current being drawn is varied with a current generator and the comparator is connected to output a logic low if there is an overcurrent condition. The next section on switching design will explain why this is not necessarily the final conditions used for the comparator.

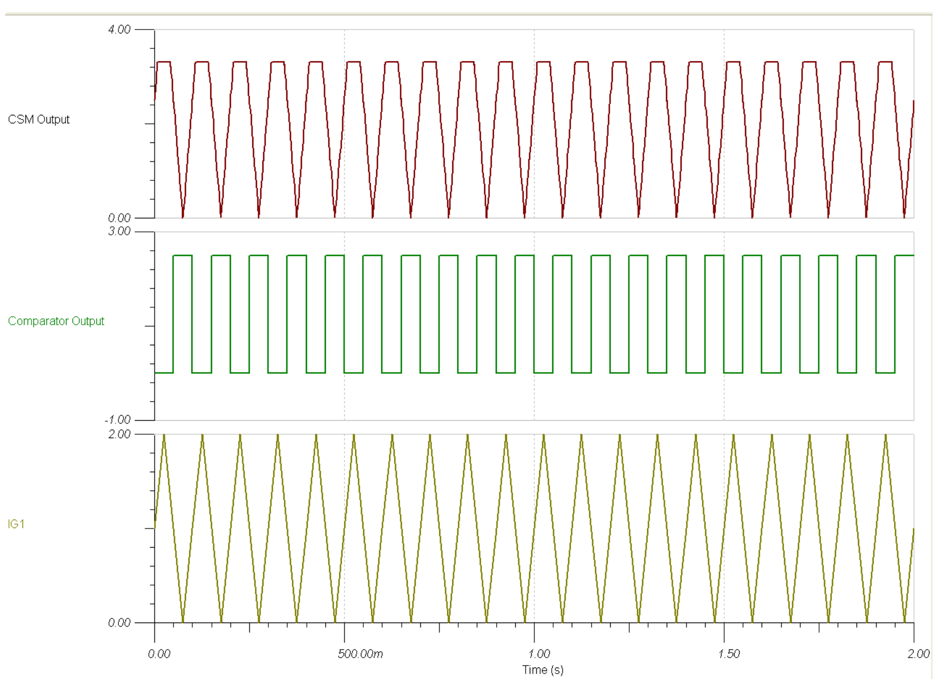


Figure 9: Output from Comparator, Current Drawn, and Current Shunt Monitor Output

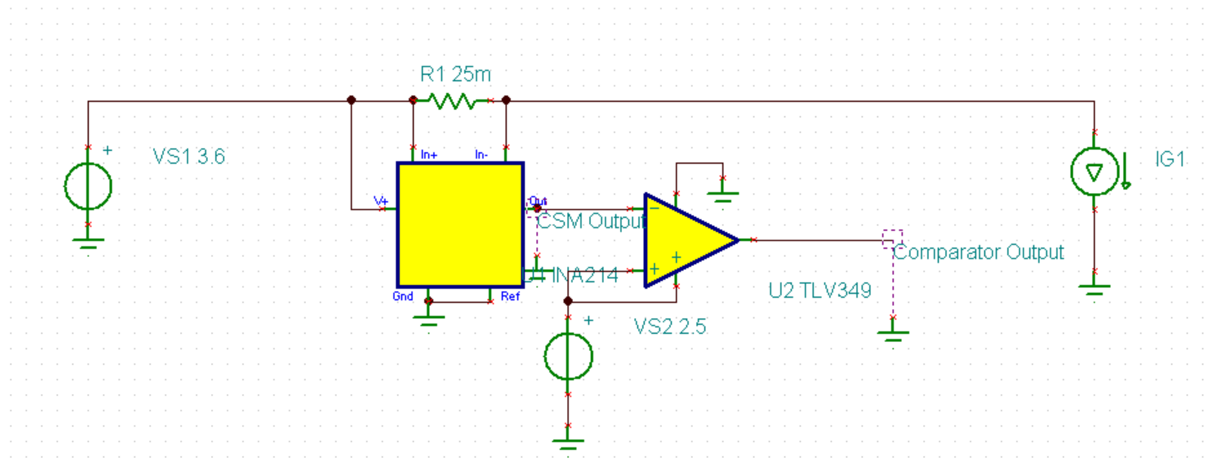


Figure 10: TINA-TI Spice Schematic Used to Produce Figure 9

As one may see, as current goes above 1 A the comparator outputs 0 V, if it is less than 1 A the comparator outputs 2.5 V. This is the expected operation for the current shunt monitor into the comparator.

3.1.3 Switching

This part of the project proved to be the most difficult. Many factors had to be considered when choosing a switch; especially speed and voltage drop across, since a high side switch is being used. There also turned out to be issues in the previous sections that prevented the circuit from switching as expected.

Selecting a switch was the first challenge encountered in the switching block. Design Team 5 considered three basic ideas: a single MOSFET, a load switch IC with internal logic and a MOSFET, and two MOSFETs connected in a load switch topology. The following table shows the parts considered for the switch. Note that for the FDS8858, the P-Channel will be on the high side of the load and both will be switched in a turn-off condition. The connection of the two MOSFETs will be shown later.

Table 4: Switch Selection Table

Part Number	Type	Switching Delay	Fall Time	R_{ON}
TI TPS1100	P-Channel MOSFET	13 ns	2 ns	476 mΩ
TI TPS2033	Power Distribution Load Switch	15 ms	3 ms	33 mΩ
TI TPS22907	Load Switch	40 µs	116 µs	44 mΩ
Fairchild Semi FDS8858 (N-Channel)	Dual N/P Channel MOSFET	19 ns	3 ns	15.2 mΩ
FDS8858 (P-Channel)	Dual N/P Channel MOSFET	33 ns	16 ns	26.5 mΩ

The first switch tested was the TPS1100 P-channel MOSFET. Upon testing, it was determined that a single MOSFET would not work for this application. The MOSFET is unable to switch from 3.6 V when on to 0 V when off. This is most likely due to the current having no path to ground through the switch. The output would switch when a logic low was applied to the gate, but would only drop from 3.6 V to close to 3.3 V. The other design blocks also needed to be changed for this part, since it would be a low enabled switch.

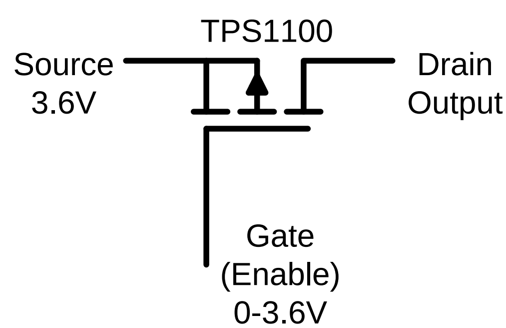


Figure 11: Test Connection for TPS1100 MOSFET

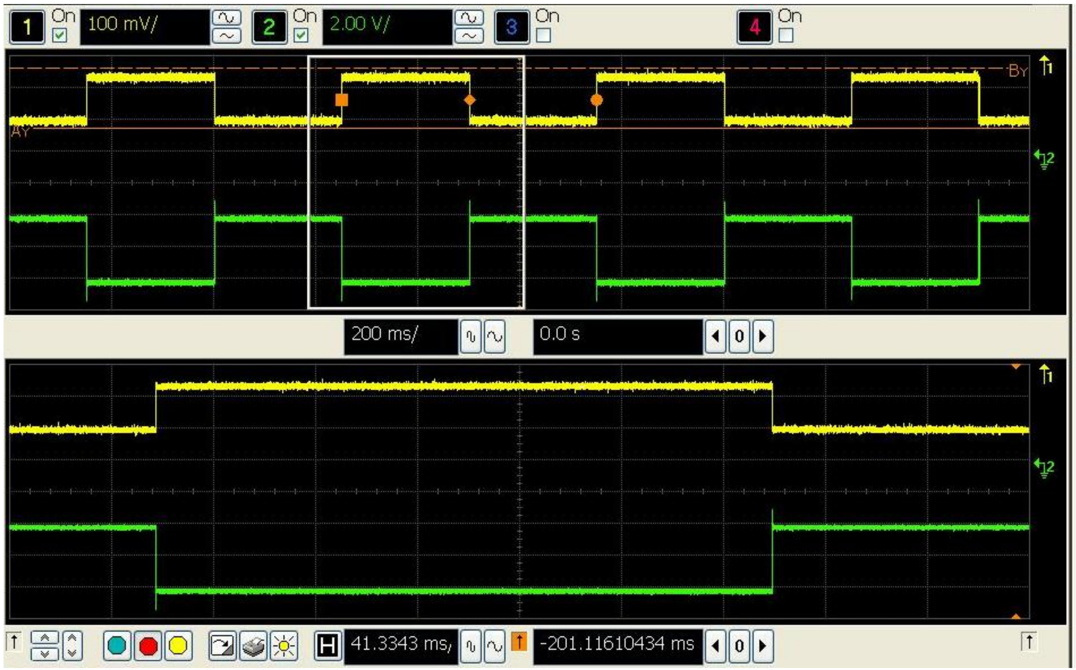


Figure 12: Test Results of TPS1100. Green: VSG Yellow: VD

Next, the load switch IC was considered. The first prototype used the TPS2033 power distribution switch and worked close to expected. However, as shown in the above table, this part is very slow and does not work to the specifications given by Texas Instruments. The TPS22907 turned out to be too small for the team to be able to use and only had a maximum current rating of 1 A. It was also significantly slower than other options available. The slow switching speed may be caused because there is extra logic included within the chip that is not needed in this specific application. The picture below displays the internal block diagram of the TPS2033 power distribution load switch.

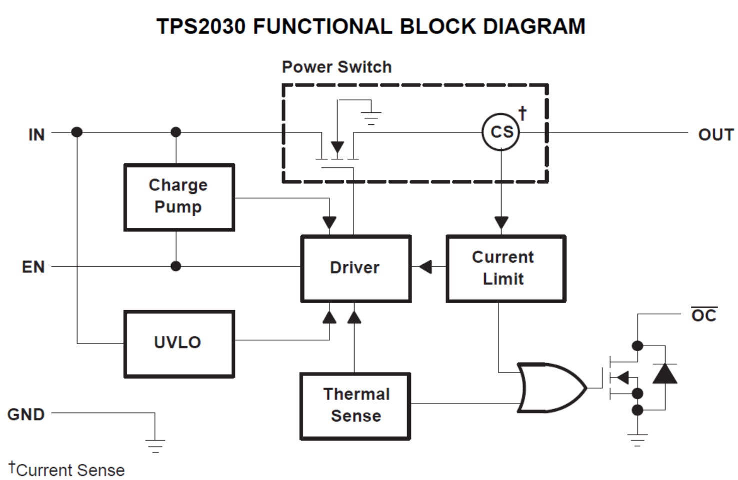


Figure 13: TPS203X Series Block Diagram (http://focus.ti.com/lit/ds/symlink/tps2033.pdf)

In order to increase the speed at which the circuit turns off, other options were explored using only MOSFETs, which appears to be the fastest option. During this research, it was discovered that a load switch can be made using two MOSFETs, one P-channel and one N-channel, using the following topology:

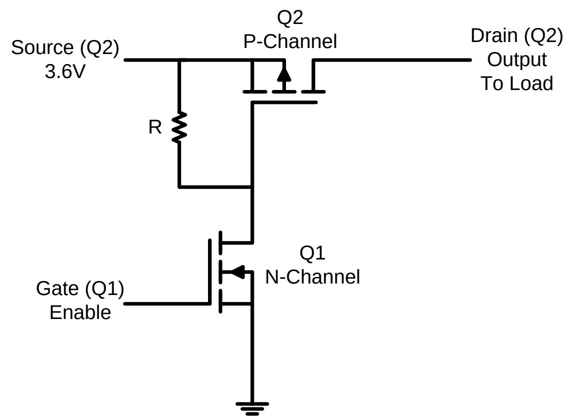


Figure 14: MOSFET Load Switch

This basic idea worked as expected and was able to drop the voltage from 3.6 V to 0 V at the output. In order to minimize parts and cost, the FDS8858 from Fairchild semiconductor seemed to be a good option, since it includes both a P and N Channel MOSFET and is much faster than the ICs available with the above load switch schematic enclosed[1]. The following displays the test results of the FDS8858:

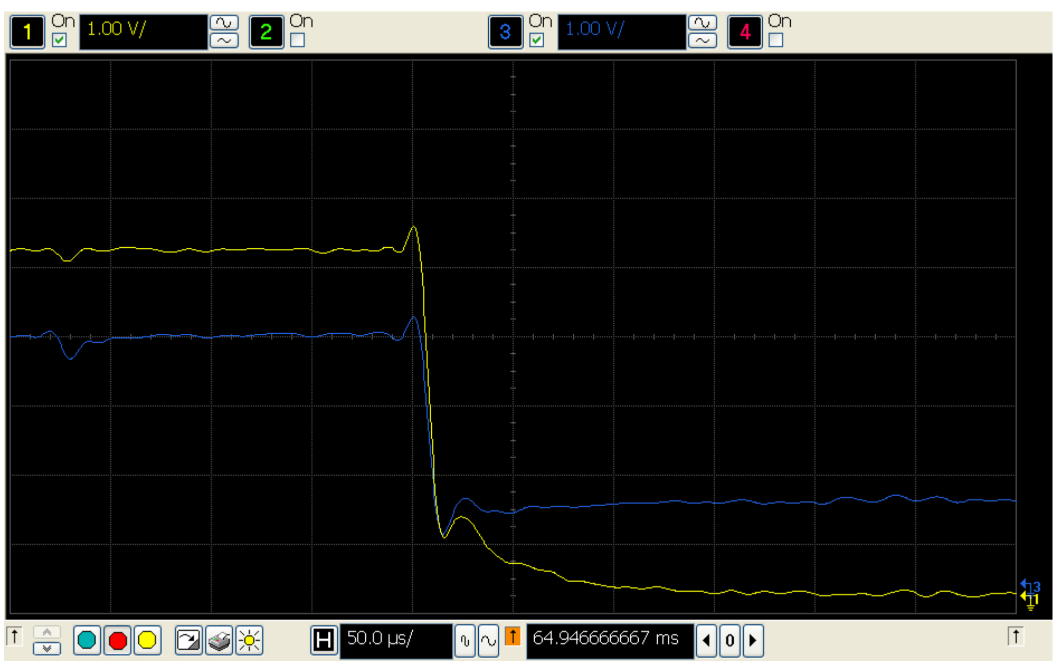


Figure 15: Lab Test of the FDS8858 Dual MOSFET. Yellow: V_D (G1) Blue: V_S (G2)

These results show that the switch does output 3.6 V when a high voltage is applied to the N-Channel gate and 0 V when a low voltage is applied. This also shows that the switch is very fast compared to TPS2033, which will be shown in the next section.

The next challenge was realized after initial testing of the first design, which will be displayed in chapter 4.To hold the switch open after an over-current condition is reached, another part was required. With only the current shunt monitor and comparator controlling the switch, the output will just oscillate. To do this, Design Team 5 decided to use a D-type flip-flop to hold the signal of the output when it switches from high to low. This idea worked; however, there were many challenges. The basic idea of how to use the flip-flop is shown below:

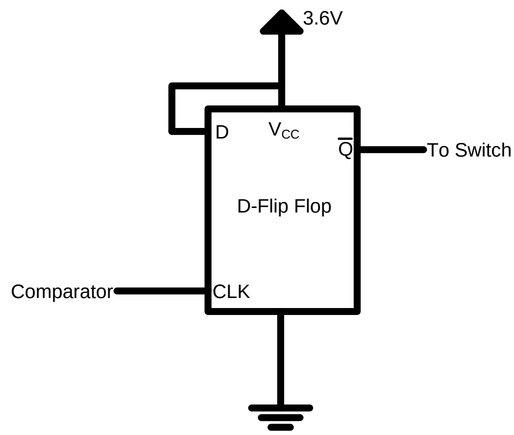


Figure 16: D-Flip Flop Configuration

The major issue with the D-Flip Flop is the initial state. The flip flops used in the design attempts had no set initial condition. It was determined that in order to set the flip-flops initial state, a part with a clear pin needs to be used, and a low pulse needs to be sent to that pin upon start up. If this is not done, the initial state of the flip flop may be low and therefore the circuit will never enable the switch to close, thus leaving the power to the load always off. The pulse can be done using the tablet-PCs microcontroller if this design is to be used. Design team 5 currently has a flip flop that has a high initial state and also is considering using a non-inverting flip flop with an external inverter. There is also a way to pulse the flip-flop using a 555 timer, but that will add multiple parts to this system, increasing cost and size of the system.

3.1.4 Final Design

With all of this considered, the final design using all hardware components is shown in the following schematic:

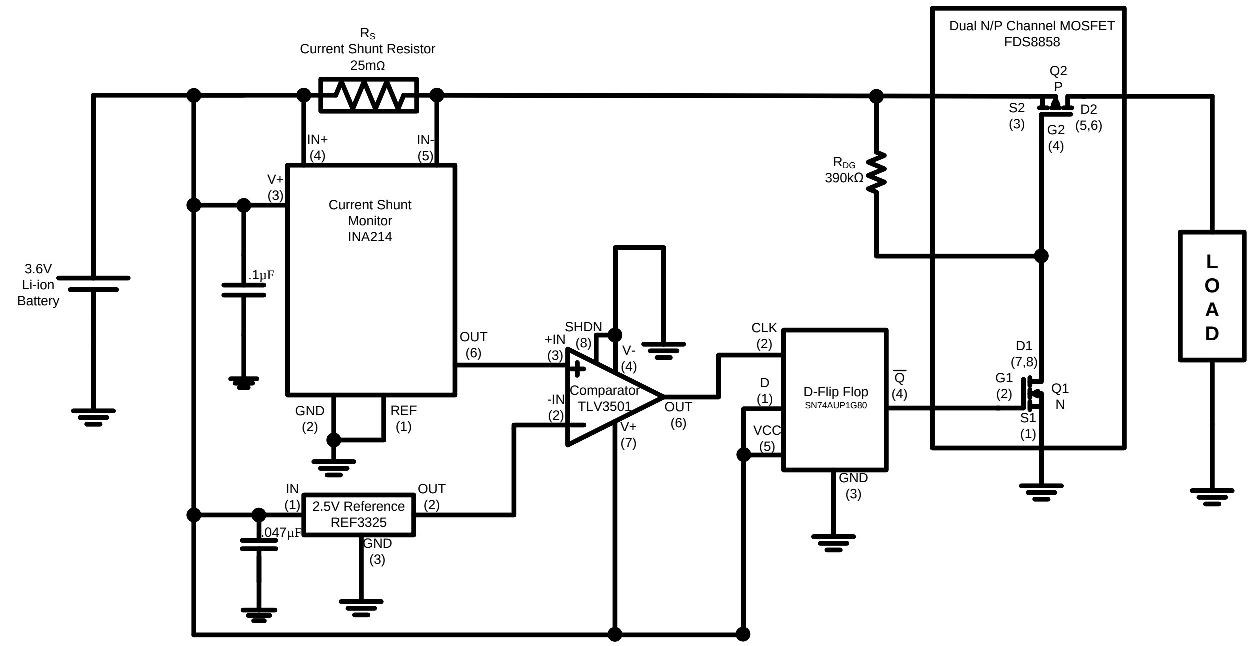


Figure 17: Final Design, Exclusively Using Hardware Components

With the parts selected, this system should theoretically shut off in hundreds of nano-seconds, have small power consumption, and fit on a board smaller than 2in X 2in.

3.1.5 Alternative Design Suggestion

The same design idea and procedure can be used implementing an MSP430 microcontroller. This would replace the comparator, voltage reference, and the flip flop. The output of the current shunt monitor output would be input into the MSP430. Software would be used to compare the voltage signal of the monitor and compare it to a set reference. If it is above a certain sense threshold, the switch will be disabled and held in that state. One issue with changing this is the resistor had to be dropped to 15 mΩ since only a 1.5 V reference could be created. The MSP430F2013 microcontroller was used for this design. General input/output pin 0 was connected to the output of the current shunt monitor and general input/output pin 2 was used to control the switch. This is shown in the following schematic:

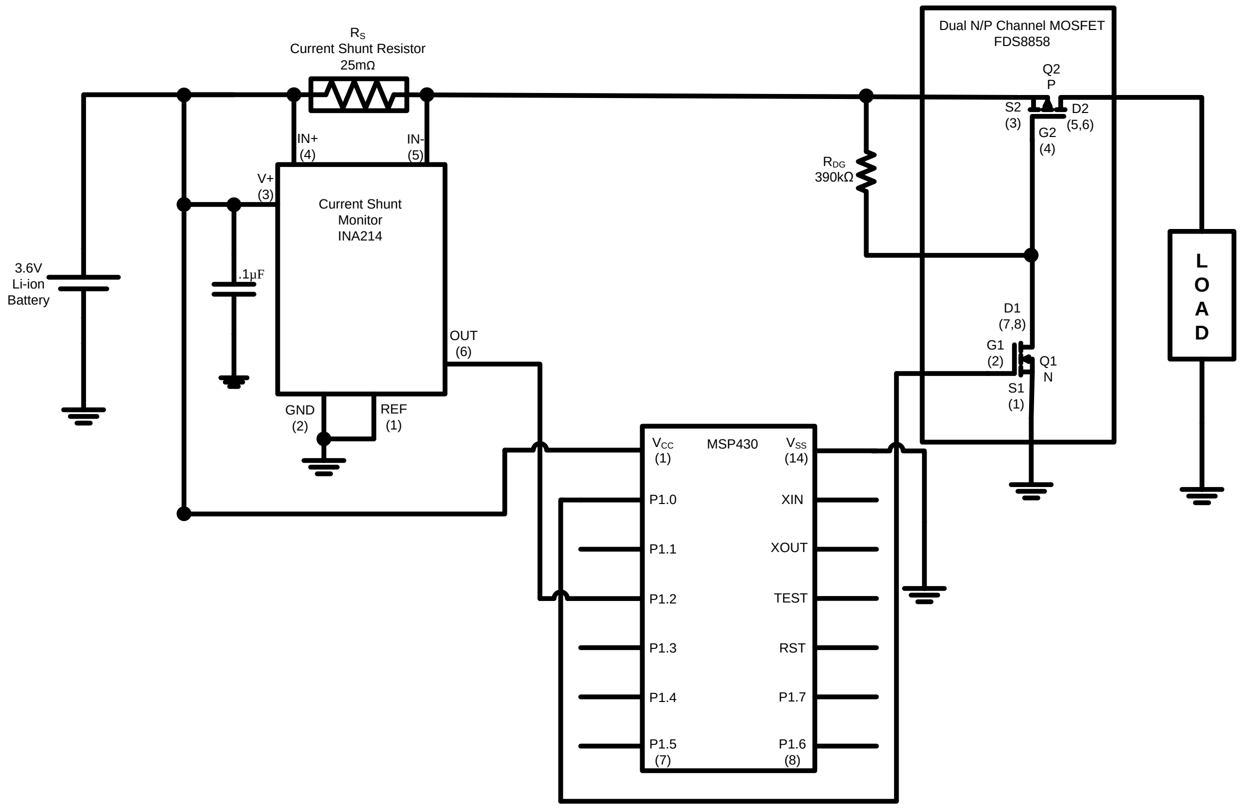


Figure 13: Final design using an MSP430

This design will include fewer components, lowering cost and system size as well as minimizing power consumption with an ultra-low power MSP430 microcontroller. Shutoff speed is determined based on the physical limitations of the part and the efficiency of the software.

3.1.6 Application One Design Challenges

The primary problem during the design process was testing the small components. Since most of these were high precision or high speed analog components, only surface mount parts were available. Therefore, in order to test parts and additions to the design system, soldering them on a PCB was absolutely necessary. This turned out to be quite difficult, since professionally fabricating boards consumes time and currency, which is not conducive to the small budget and short time frame given for this project. It was also quite a challenge to solder pins that were as small as some of the surface mount parts are. Team 5 corrected resolved this issue by chemically etching boards on campus for testing purposes. This will be described in later sections explaining the role of PCBs in this project.

3.2 Application Two Design

The purpose of this application is to monitor the current being drawn by a cell phone to accurately determine the power being consumed for during operation. Unlike application one, application two does not require over-current protection circuitry. The block diagram for the proposed circuit is shown in Figure 18. The basic method of sensing current is the same as in application one, which is to use a current shunt resistor and measure the voltage drop across the resistor to infer the amount of current flow from the power source to the load. In the block diagram, the "Variable Load" is representative of the cell phone, which is expected to have a current draw between 7 mA and 192.5 mA. The main circuit priorities are to 1) minimize system impact, 2) maintain required accuracy, and 3) minimize circuit size.

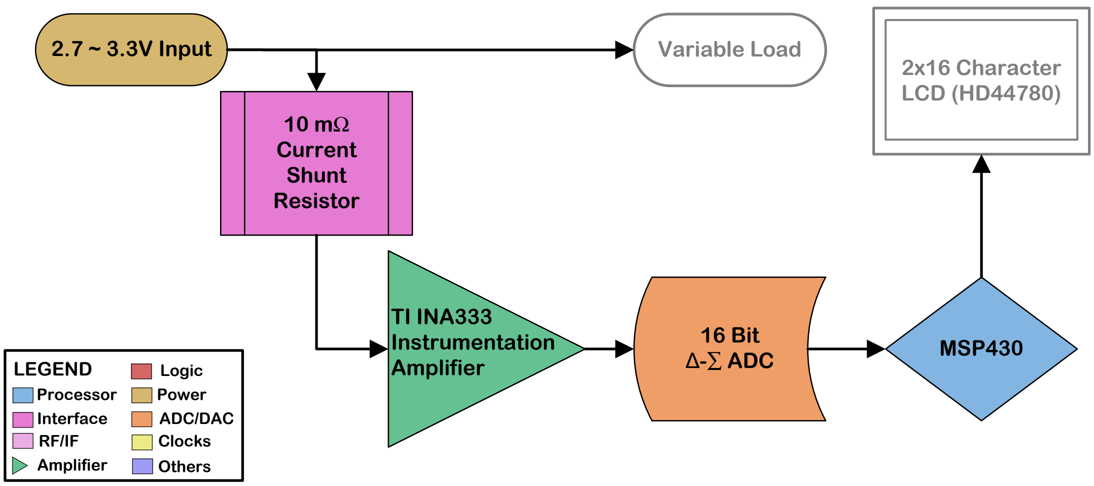


Figure 18: Block Diagram for Application Two

3.2.1 Current Sensing

To meet the first circuit priority of minimizing system impact, this application uses a 10 mΩ resistor in series with the battery and cell phone. To simulate the battery, a DC power supply was used; a potentiometer that allows for varying the current draw is used to model a cell phone load. One potential problem with using the INA138 or other similar current shunt monitors is that the input offset voltage may be too large to get a reliable reading when the cell phone draws 7 mA, which would equate to a 70 µV voltage drop ($V = I \cdot R$). To maintain the required accuracy, without sacrificing system impact, a circuit besides a current shunt monitor was selected from TI's portfolio. The selected part was the INA333, a micro-power zero-drift, rail-to-rail out instrumentation amplifier. The part was selected for its exceptionally low power consumption while offering a precise and accurate method for amplifying the small voltage drop expected across the current shunt resister. The maximum offset voltage for this particular part is 25 µV, and is typically less, which allows for measuring of the 70 µV voltage drop without the offset voltage introducing too much error. Quiescent current draw of the INA333 is rated at 75 µA, which is helpful in meeting our design criteria. It is also offered in the MSOP-8 package, which will allow for a small circuit layout.[2]

The INA333 has inputs for an external resistor to set the gain provided by the circuit, which is determined using the formula: G $= 1 + (100k\Omega/R_G)$. Since the expected voltage drop across the current shunt resistor, when the cell phone load is drawing 192.5 mA, is 1.925 mV and the expected internal reference of the MSP430 will allow for full-scale measurements up to 300 mV, a gain of 100 is desired. For this purpose, the R_G is selected as 1 kΩ. Although the MSP430 allows for higher internal voltage references, or even the use of an external one, this design is focusing on minimizing the system impact. This means that a minimal number of parts should be utilized, as well as ensuring the circuit's low power operation.

3.2.2 Software and Interface

As shown in Figure 18, there is a MSP430 microcontroller that interfaces to a 16x2 LCD in the suggested design. The main task of the MSP430 is to take in the analog output of the INA333 through an analog-to-digital converter (ADC) and display the levels on the LCD. To accomplish this task, Design Team 5 designed and implemented software for the microcontroller. The programming language chosen was C, so all members of the group could read and understand what the code is attempting to accomplish, and the MSP430 was debugged using a MSP-TS430PW14 (MSP430 14-Pin Target Board) and the MSP-FET430UIF (USB Debugging Interface). The team opted for the Code Composer Studio integrated development environment (IDE), since it offered a similar interface with members of the team who are familiar with the Eclipse IDE.

The actual code used for the MSP430 included two main sections, one that reads in the analog value, and another which takes the value and prints to an external LCD. The main components of the circuit were described in section 3.1.1 Current Sensing, one part that was not mentioned is the 16 bit Δ-∑ Analog to Digital Converter (SDADC). Although the initial plan was to include an ADS 1113, 16 bit ADC with integrated multiplexor, oscillator, and reference, it turns out that certain MSP430 microcontrollers have this capability built in. To further reduce the cost of this system, the ADS 1113 from the proposal design is removed and replaced by the MSP430-F2013, which has an internal SDADC. Therefore, for the section of code that reads in the analog value, there is no need to use I^2C or similar communication techniques to interface to an external ADC, which makes the code easier to read and understand. The onboard SDADC of the MSP430-F2013 was initialized to read inputs on P1.0 using the internal voltage reference. The register containing that value was updated several times a second, too fast for the human eye to capture the information if it was updated on the LCD. To overcome this limitation, the LCD was programmed to update after 16 ADC updates, which keeps the value on the LCD long enough for a human observer to interpret, and also allows the value to be averaged over 16 samples, which increases stability of the readings.

The LCD portion of the code is designed to take the 16 averaged ADC results, and display it on the 16x2 HD44870 LCD. Because the MSP430-F2013 has a limited number of pins that can be accessed after setting up the ADC portion, Design Team 5 decided to interface with the LCD using the 4-bit programming mode instead of 8-bit programming mode, which requires half the number of connections, but requires more careful timing and programming consideration. The 4-bit and 8-bit programming modes are methods of interfacing with a HD44870 LCD, the details of which can be found in the datasheet, available online. Although time-consuming to develop, the software used in this application does not contain any unique or novel techniques, thus it is omitted from paper. For those interested in inspecting the code, it is available at Design Team 5's website: www.egr.msu.edu/classes/ece480/capstone/spring11/group05/documents.html.

3.2.3 Final Design

The final design for application two is shown in Figure 19, and contains five parts: the INA333 instrumentation amplifier, a 10 mΩ shunt resistor, a 1 kΩ resistor to set the gain, a MSP430-F2013 microcontroller, and a genetic 16x2 LCD with a HD44780 interface.

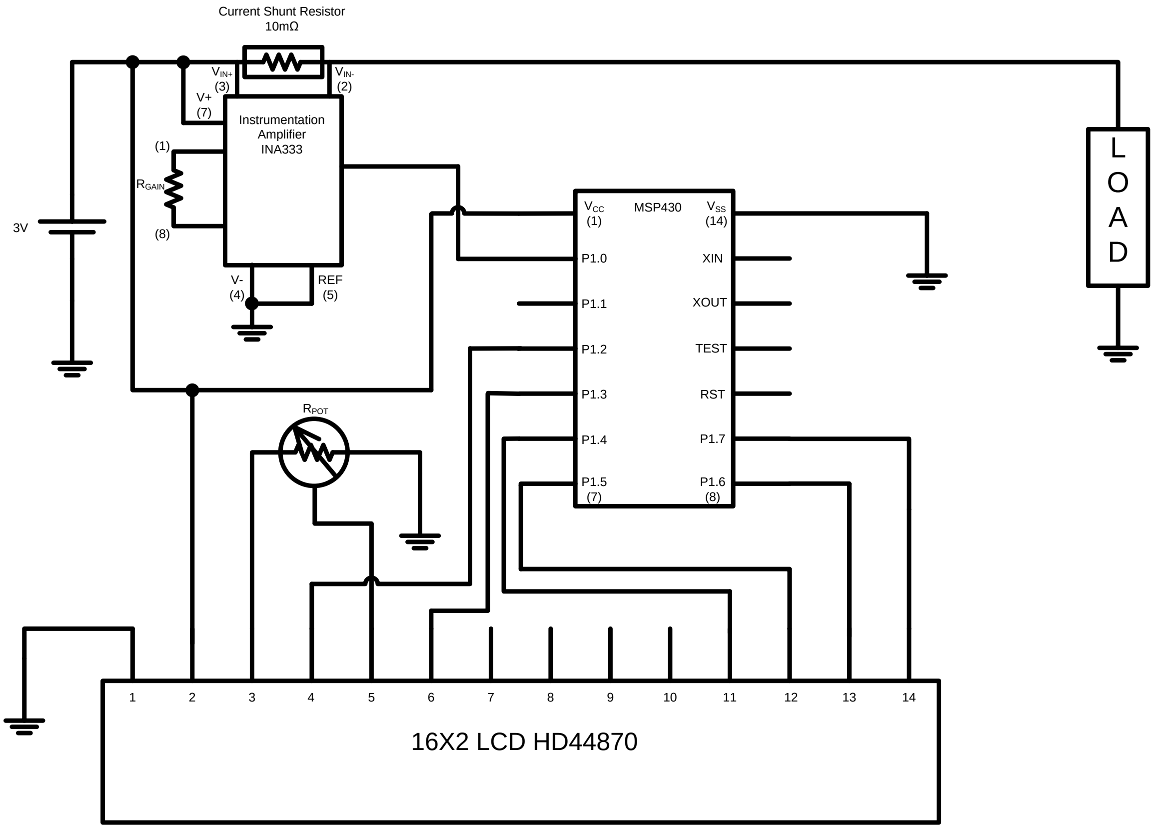


Figure 19: Application Two Schematic

From the dimensions of the parts used, the final board size may be about 1.016 cm by 1.778 cm (0.4 in by 0.7 in). This small board size is accomplished when using the TSSOP package for the MSP430, which is about 5 mm by 6.5mm. Although it does come in an even smaller (4mm by 4mm) package, Design Team 5 does not have access to the tools required to build a circuit using packages smaller than the TSSOP. The mentioned sizes are excluding the external LCD, which is several times larger than the rest of the circuit.

3.2.4 Alternative Design Suggestion

One important component missing from the proposed circuit is a capacitor to reduce voltage ripples, which should be added between power and ground pins on the MSP430. Other than that hardware change, most of the alternative designs deal with using a different microcontroller or changing the software. From the testing results, discussed later, this circuit appears to have the capability to resolve down to 1 mA, sufficient for this project, but not nearly as good as the INA333 and SDADC should be able to offer. One method of improving the resolution may be to oversample the analog signal, or use the 16-bit SDADC twice, with an overlapping byte, to get 24-bits of resolution. This method may reduce the number of samples per second, but there should still be more than enough to provide accurate power consumption measurements. Another alternative design suggestion is to optimize the software to reduce power consumption of the current measuring circuit. The current code does not take advantage of the many power-saving features of the MSP430, such as turning off the CPU when it is not being used, or reducing the operating voltage.

3.2.5 Application Two Design Challenges

One problem during the design process was creating the code to interface with the LCD using the HD44870 protocol without interfering with the SDADC operation. The two pieces of code also had to communicate with each other, with required careful bit manipulation to convert the SDADC results into a form that was human-readable on the output of the LCD. Since none of the Design Team 5 members were familiar with the MSP430, several hours were spent learning the basics of how the microcontroller operates, and how to program the chip.

Besides the challenge of producing code that behaves as intended, there was also the design challenge of migrating from the MSP430 target board with USB debugger to a PCB that operates on a 3 V battery source. This challenge remains unsolved at the time of writing this report. When connected to the target board, the MSP430 reacts as expected to external stimulation, but does not appear to work when soldered onto the PCB. The MSP430 was programmed with the Release code instead of Debug, and still works when de-soldered from the PCB and placed back in the target board, which suggests that the microcontroller was not damaged during the soldering process. Design Team 5 is currently inspecting the components that are on the target board that are required for proper operation of the MSP430, and attempting to modify the PCB so that it will operate as well.

3.3 PCB Design & Fabrication

Due to the low resistance requirements for connecting a current shunt monitor to a current shunt resistor, as well as the small package sizes require in this project, the traditional method of prototyping with a proto-board is not useful. Due to the nature of this project, Design Team 5 required several rapid prototypes, which would have easily pushed the team over-budget if professional fabrication services were employed. Michigan State University allows student access to an in-house PCB milling service; however, this service did not have the required resolution to deal with the small package sizes such as the TSSOP. Therefore, Design Team 5 turned to chemically etching PCBs, which allows for low-cost, high-resolution PCB fabrication. To reduce development costs, Design Team 5 used EAGLE, a PCB design software offered as freeware to educational users, by CadSoft. The intent was to develop the prototype boards using this cheap chemical etching process, and produce a final PCB using a professional fabrication service, such as Sunstone Circuits, which offered a quote of $ 121.00 to create one application. However, due to time constraints this professional fabrication was not used, since the boards would not have arrived before Design Day.

3.3.1 PCB Design

The design of the PCBs was done in EAGLE, a commonly used piece of software in industry. There were several design issues with generating PCB layouts, such as ensuring a proper connection between the current shunt monitor and current shunt resistor existed, and routing traces without the use of a second signal layer. Although the chemical etching process does allow for two-layer signal routing, it greatly increases the complexity of the hardware, thus increasing the changes that the hardware may fail. For this reason, Design Team 5 designed the PCBs to use a single layer and manually routed the traces between components placed on the PCB. This process required a significant investment of time, as a library of component dimensions and connections had to be created specifically for this project. In the interest of brevity, the details of how the PCBs were generated are omitted from this report, please refer to Application Note: PCB Design with EAGLE, available on Design Team 5's website under the Documents tab.

3.3.2 PCB Fabrication

Once the PCBs were designed in EAGLE, the layout was printed as an inverted (color) image onto a transparency slide. This transparency acts as the "mask," used to transfer the layout design to the substrate, as shown in Figure 20.

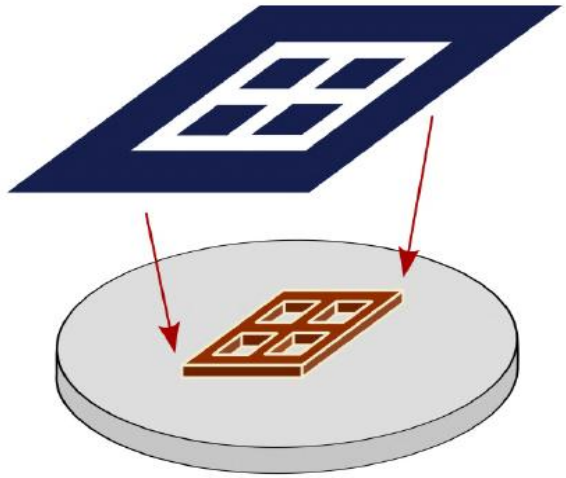


Figure 20: Mask to Substrate Transfer

Once the mask is successfully created, it is placed between a treated substrate and an ultraviolet light source. The ultraviolet light will pass through the clear sections on the mask and leave an imprint on the treated substrate surface; these clear sections correspond to the final copper traces. The substrate is then etched in a solution of sodium persulfate ($Na_2S_2O_8$) which removes any copper that was not exposed to the ultraviolet light. The etching solution and substrate after expose are shown in Figure 21.

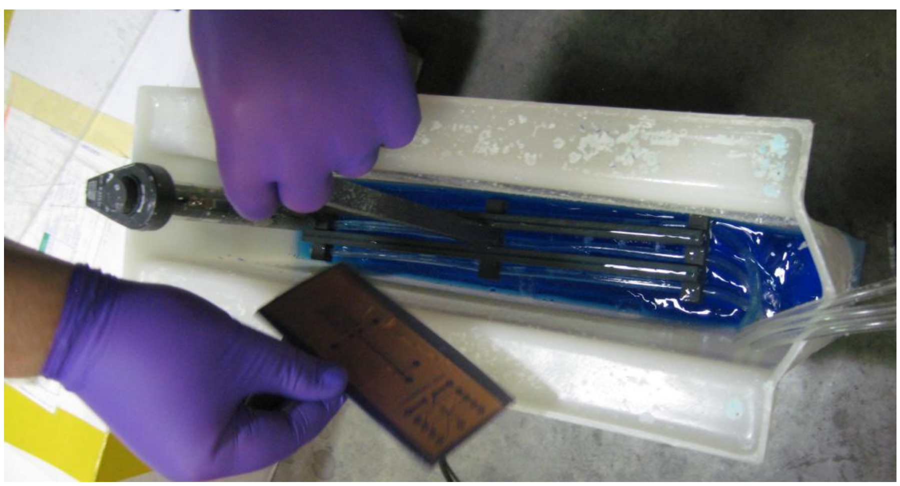


Figure 21: Submerging the Substrate in Etching Solution

Once the excess copper is removed, the board is washed in acetone to remove the treated layer, and is ready for polishing or plating for a high-quality board. For a more detailed description of this process, please refer to A Process of Chemically Etching Circuit Boards, another document available on Design Team 5's website. The PCB fabrication using this method requires only a few house to perform, is low cost, and can achieve a resolution of approximately 150 µm. An image of a chemically etched PCB is shown in Figure 22.

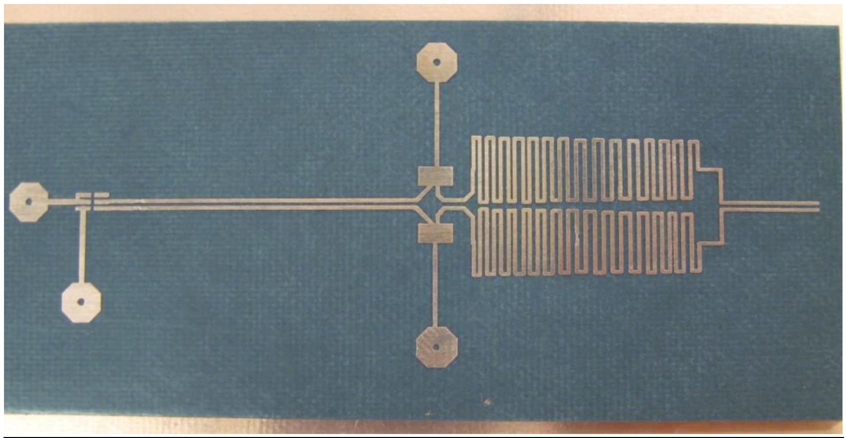


Figure 22: Result of PCB Fabrication

Chapter 4: Results & Proof of Functional Design

4.1 Application One Results

4.1.1 Final Design with Only Hardware

As a complete system, application one operated as expected and shutoff power when current being drawn to the load was 1 A. The following oscilloscope screenshot shows that when the current shunt monitor outputs 2.5 V, the output at 1 A, the switch is disabled and power being supplied to the load is deactivated.

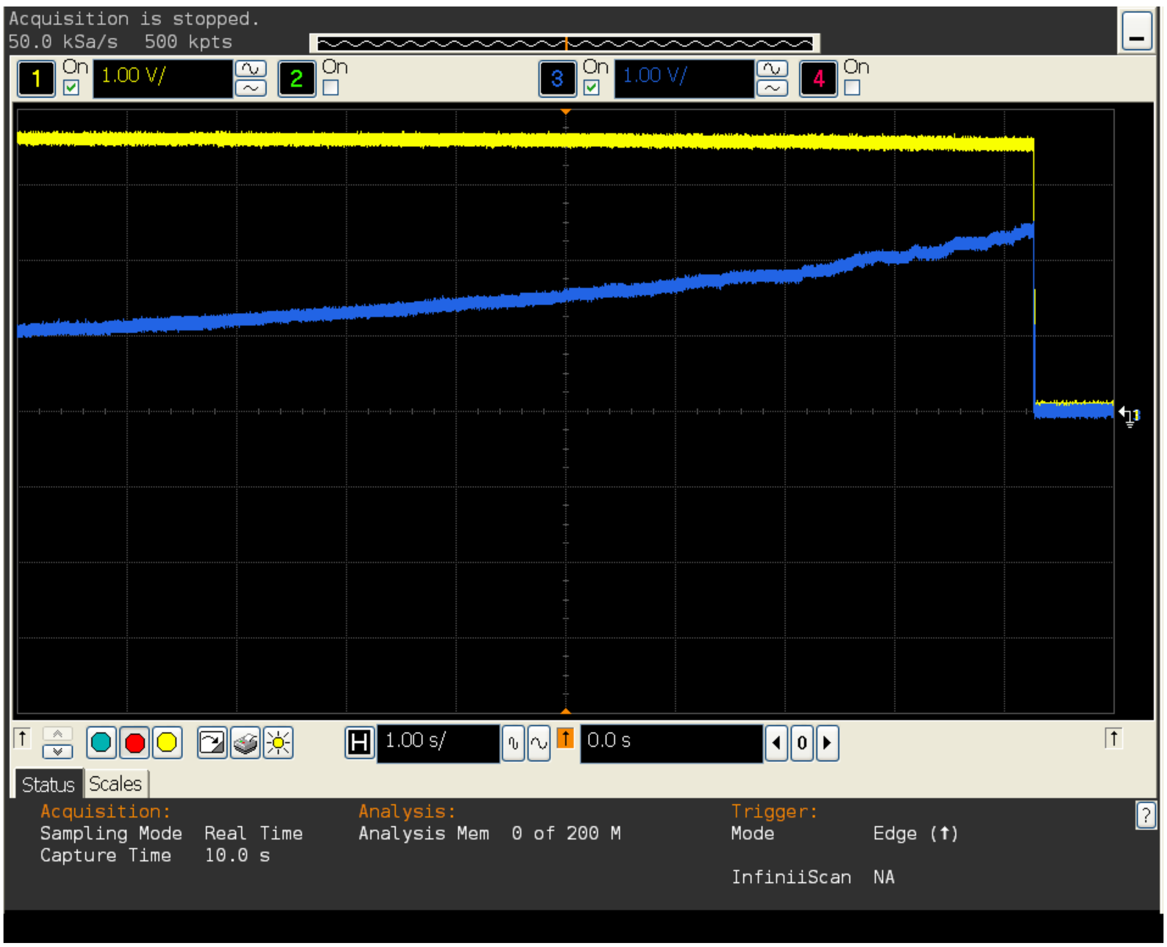


Figure 23: Load Voltage (Yellow) v. Current Shunt Monitor Output (Blue)

To create this result, a potentiometer simulated the load and was slowly decreased until it reached the 1 A threshold. The voltage output to the load is 3.6 V, as expected, and dropped to 0V when power was cut. This result was taken using an SN74LS74 DIP, as it was the only working circuit at the time. The following is the oscilloscope screenshot focused on the switching speed of the circuit, again output voltage vs. the current shunt monitor.

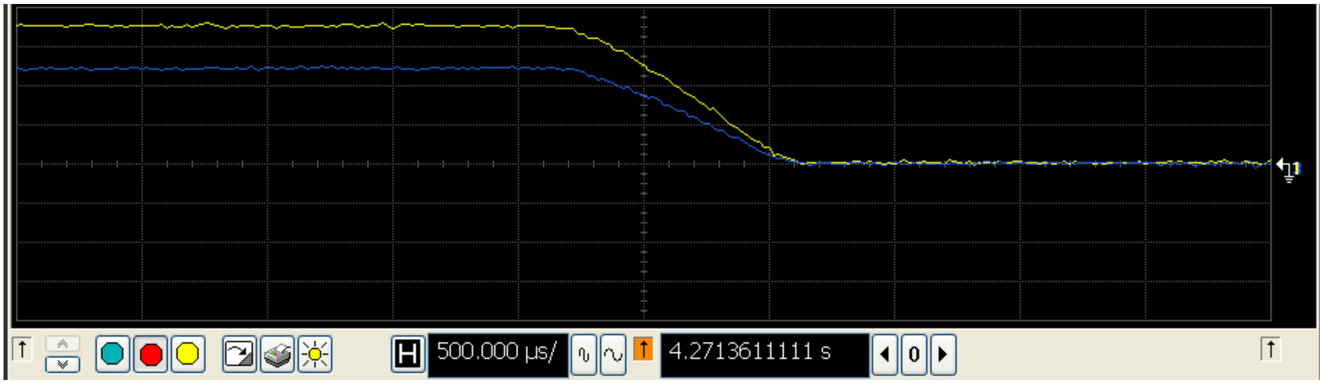


Figure 24: Fall Time of the Switching System

This shows that the fall time with the DIP is about 750 µs, slower than expected, but still much faster than the circuit using the TPS2033, which will be shown later in this chapter. A switching speed test using a surface mount D-flip flop was not completed, since the initial state never allowed the switch to close and supply power to the load. Some other specifications were able to be tested, such as size and power consumption. The following table compares application one results using a SN74LS74 DIP package flip flop vs. a SN74AUP1G80 SC-70 package flip flop.

Table 5: Size and Power Consumption Results of Hardware Final Design

Type	Size	No-Load Current Draw	Power Consumption
SN74LS74	0.97 in X 0.75 in	7.58 mA	27.288 mW
SN74AUP1G80	1.01 in X .529 in	3.83 mA	13.788 mW

The circuit using a DIP consumes significantly more power than the circuit using a small surface-mount part. However, they are both drawing more power than desired. This is due to the comparator, which has a very high current draw to be active. To optimize this design, a different comparator should be considered or possibly use the MSP430 microcontroller design.

4.1.2 Application One Results with MSP430

Design team 5 also tested application one using a MSP430 instead of the comparator and flip flop to process the signal output by the current shunt monitor and trigger the switch. The following shows the switching speed of the only the MSP430.

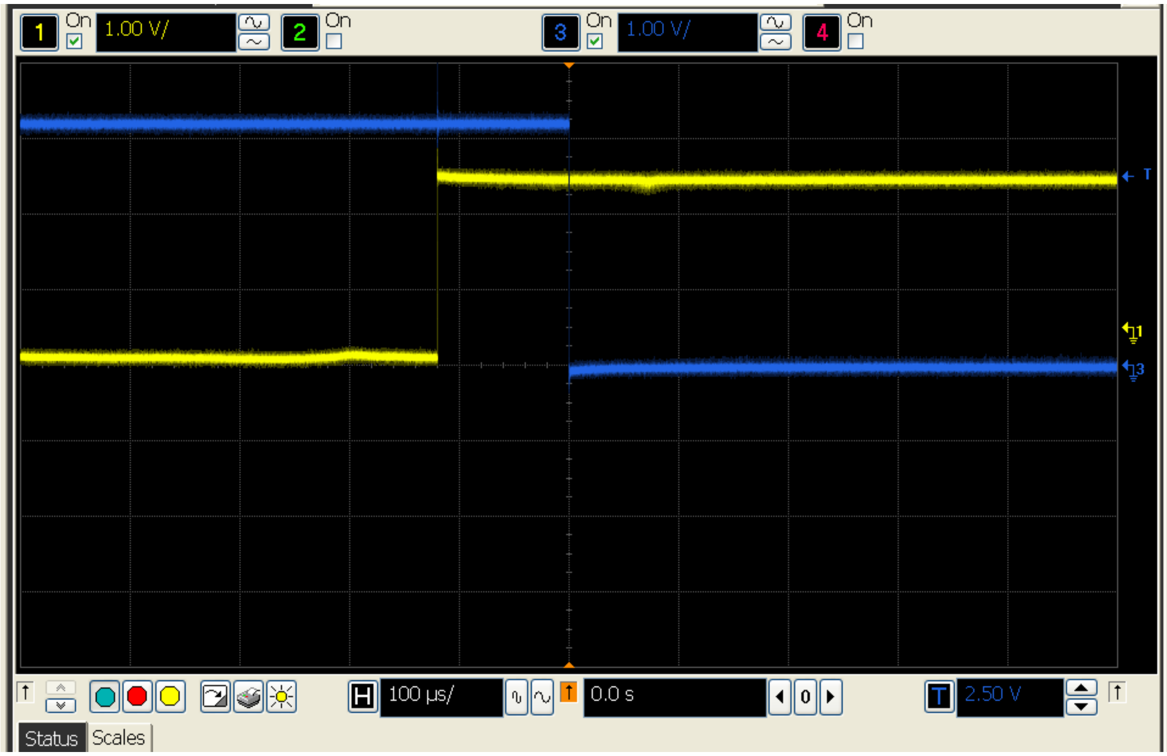


Figure 25: Switching Delay Between Input and Output of MSP430

This shows that the MSP430 switches in 100 µs. The only other speed factor in the system is the FDS8858 MOSFET load switch, which, as shown in chapter 3, switches in under 10 µs. The system using the microcontroller is expected to be seven times faster than the circuit using a comparator and flip flop. The MSP430 also consumed less power than the other circuit.

Table 6: Size and Power Consumption Results of Microcontroller Final Design

MSP430 Mode	Current Draw	Power Consumption
Sleep	4.8 µA	17.28 µW
Active	80.2 µA	288.72 µW

4.1.2 Application One Results with Early Design

The first result tested showed that the basic functional idea worked. However, this is where the team determined an additional component would be required to latch the signal. The following is the results and the schematic of the first prototype.

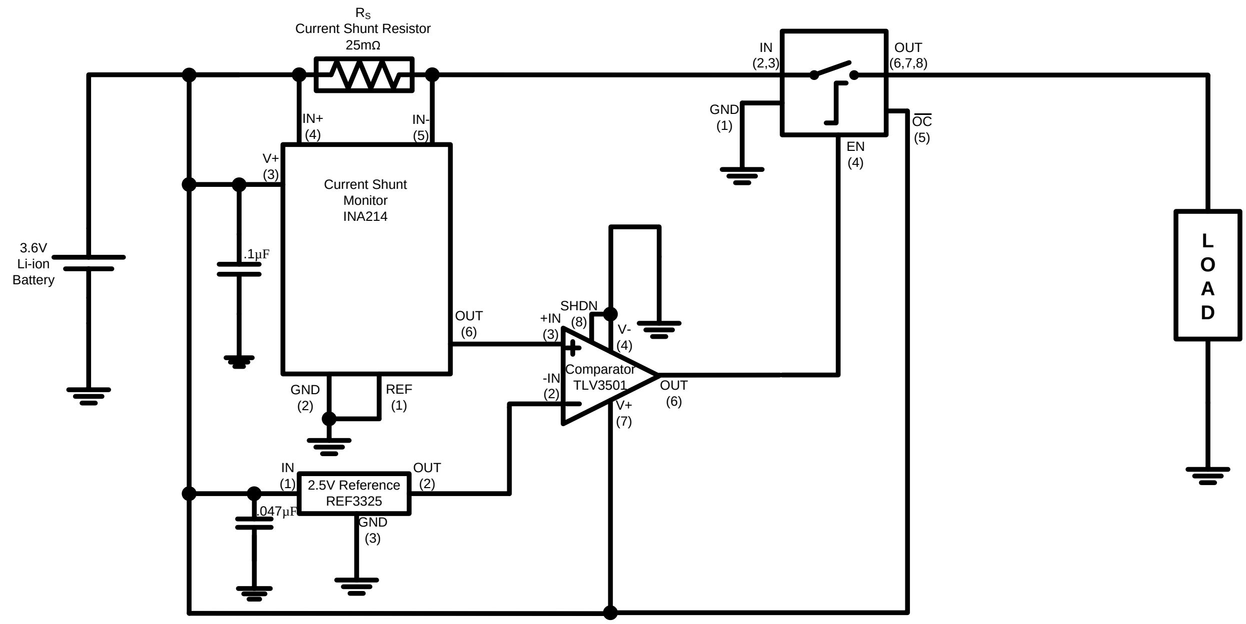


Figure 26: Initial Prototype Schematic

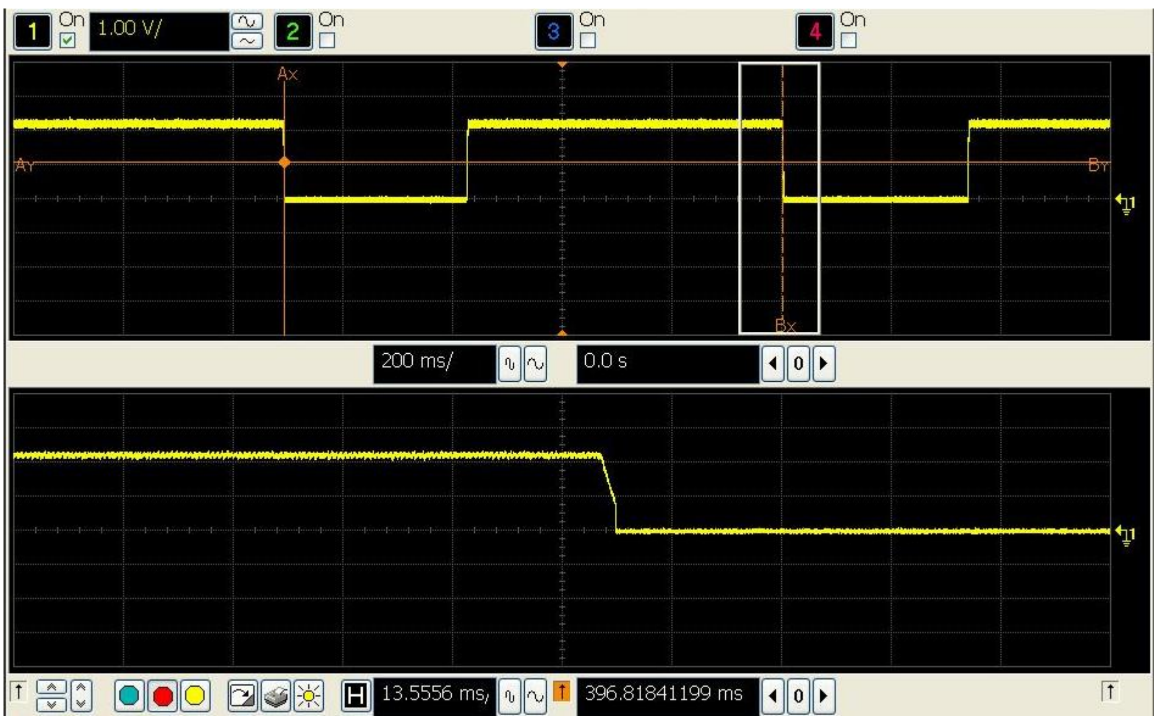


Figure 27: Results from Initial Prototype

The yellow trace on the oscilloscope screenshot is the voltage supplied to the load. The switch is opening when 1 A of current is drawn, however once it is opened, the current drops and allows the switch to turn back on, and it continuously oscillates. The above screenshot also shows that the TPS2033 is very slow, switching with a fall time of about 3 ms.

4.2 Application Two Results

When the MSP430 is connected to the target board, the circuit performs as expected, and meets the project specifications. The target board with the output displayed on the LCD, and measured using the Hewlett Packard 34401A is shown in Figure 28. The LCD is displaying two measurements, the previous result of the SDADC is shown on top, and the current measurement is shown on the bottom line. This allows a user to monitor how the current levels have changed since their last measurement. The refresh rate for the measurements is set to around 200 ms, using a NOP delay loop, so that the measurement is displayed long enough for human observation.

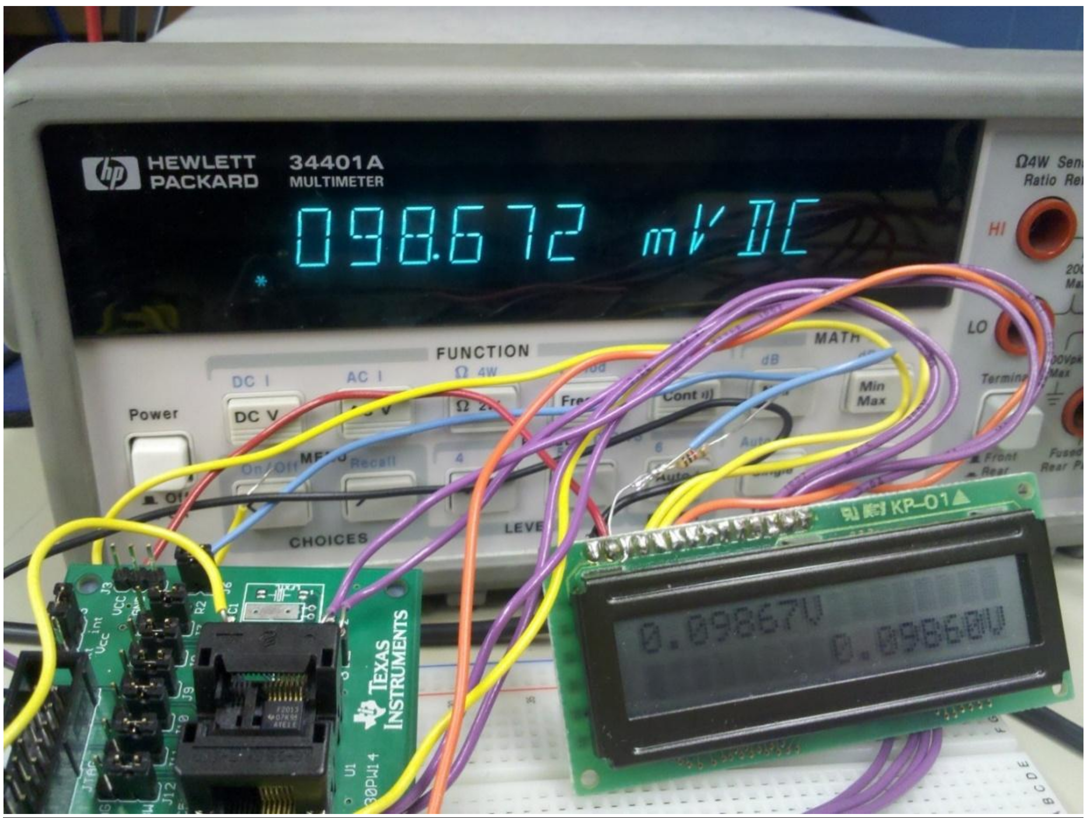


Figure 28: Application Two Using Target Board

The test setup in Figure 28 takes the output of the INA333, which is an amplification of the voltage drop across the current shunt and converts it to a digital signal using the MSP430. The signal is then successfully displayed on the LCD, as shown. One point that should be explained is that the LCD is currently showing units of volts. This is because the test was attempting to measure the voltage output of the INA333, to convert this to current is as simple as replacing the ‘V’ with an ‘A’. From this successful measurement, it is also simple to infer the power consumption of the circuit.

Application two works, as long as the MSP430 target board is used. This application is a "successful failure," since it does not work when the MSP430 microcontroller is removed from the target board and placed on a PCB. The reason for this failure is not completely understood by Design Team 5, but it is assumed that the target board contains components that allow the MSP430 to properly initialize, it may be as simple as a missing capacitor on the final PCB.

4.3 PCB Design Study Results

Current sensing using resistive shunt applications involve taking a very precise measurement since small signals are involved. Therefore, adding unnecessary disturbances in the circuit to this measurement can greatly affect the output of the shunt monitor and cause unexpected operation. TI has requested various study cases to display how common PCB layout errors in current sensing applications can skew the signal being output.

In order to do this, Design Team 5 designed four possible PCB layouts of an INA214 current shunt monitor with a 25 mΩ current shunt resistor. The load was varied to draw more or less current to see how error may be affected. The current being drawn to the load was measured using a Hewlett Packard 3401A ammeter. This reading was used to determine the expected output of the current shunt monitor using the following equation:

$$V_{OUT} = Gain\ x\ Current\ Shunt\ Resistance\ x\ Measured\ Current\ Draw$$

The gain of the INA214 is 100 V/V. The actual output of the INA214 was measured using the Hewlett Packard 34401A voltmeter. This measurement was then used to calculate the final percent error involved. Additional error may have been introduced due to the use of wires connected to a proto-board, solder joints, and inconsistencies due to using a different current shunt monitor for each case. The test board is shown in Figure 29. The following results will show that the use of different ICs causes negligible error since errors from the traces are so great. The other sources of error should be equal in each case, meaning most of the difference in error is caused by the traces. The study cases completed were the ideal recommended traces for current sensing applications, long distance traces, unsymmetrical traces, and traces not using Kelvin connections.

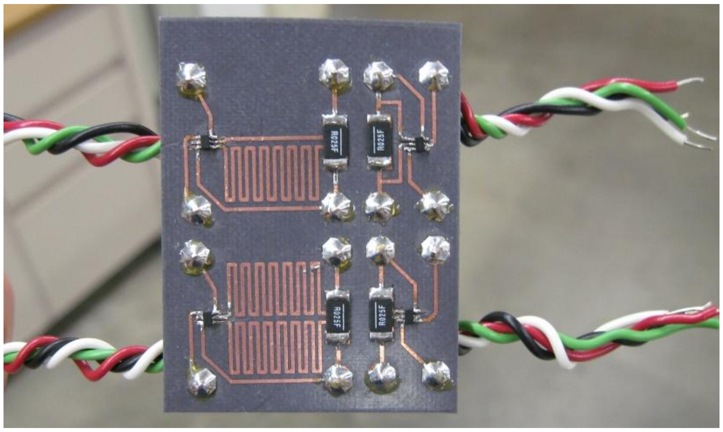


Figure 29: Study Test PCB

4.3.1 Ideal Connection

The first case tested in the recommended traces from the shunt resistor to the inputs of the shunt monitor. The recommended traces are short traces of the same length and width that are separated from high current carrying traces. This was completed in order to compare error from the ideal case to all of the other non-ideal cases.

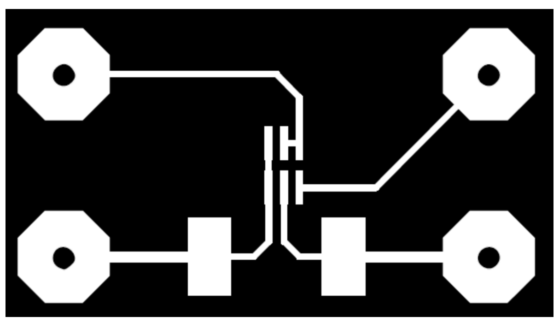


Figure 30: PCB Trace Used for Ideal Case

The results from testing the ideal case are shown below. As expected, it has the lowest error compared to the non-ideal conditions shown in the following sections.

Table 7: Ideal Traces Error Results

Load	Expected Current Draw	Current Draw	Expected Output	Output	% Error
300 Ω	10 mA	10.27 mA	25.675 mV	28.345 mV	10.4%
30 Ω	100 mA	118.92 mA	297.3 mV	323.5 mV	8.81%
10 Ω	300 mA	280.29 mA	700.7 mV	763.39 mV	8.95%
3.33 Ω	900 mA	872.4 mA	2.181 V	2.4894 V	14.14%

4.3.2 Long Distance Traces

A very common mistake in current sensing applications is placing the current shunt resistor far away from the current shunt monitor on the PCB. This adds unnecessary resistance in the input path and may greatly affect the final reading of the current shunt monitor. Similar parasitic resistance may be added when using vias to connect the shunt monitor to the sense resistor when using a board with multiple layers.

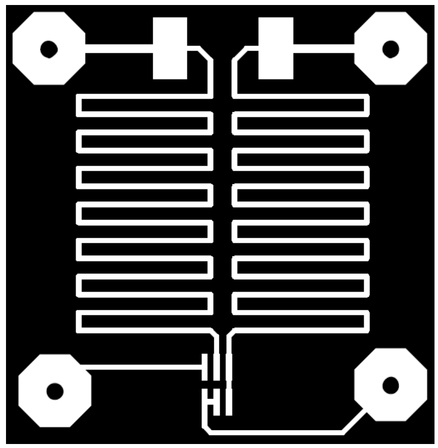


Figure 31: PCB Traces Used for Testing Long Distance Traces

The results shown below summarize how long traces have a very adverse effect on the output of the shunt monitor. Any precision required in the current measurements would be completely lost if extremely long traces with parasitic resistance are used in the PCB.

Table 8: Long Traces Error Results

Load	Expected Current Draw	Current Draw	Expected Output	Output	% Error
300 Ω	10 mA	10.21 mA	25.5 mV	45.27 mV	77.53%
30 Ω	100 mA	111.42 mA	278.55 mV	493.89 mV	77.31%
10 Ω	300 mA	254.3 mA	635.75 mV	960.5 mV	51.08%
3.33 Ω	900 mA	852.2 mA	2.1305 V	2.44 V	14.5%

4.3.3 Unsymmetrical Traces

Another common error is making the traces from the current shunt resistor to the current shunt monitor different lengths. This would cause different resistances going into each output and changing the results.

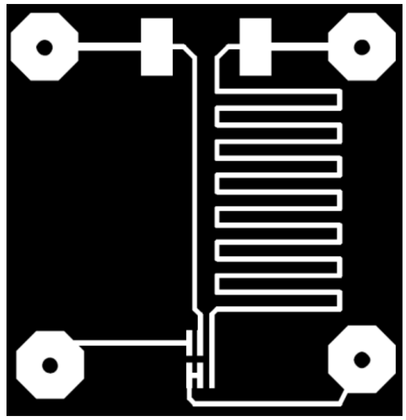


Figure 32: PCB Traces Used for Testing Unsymmetrical Traces

The following results are showing a much smaller error than expected. The specific traces used may somehow be cancelling additional error. Errors may have also been made during the testing procedure. However, according to the results below, unsymmetrical traces have very little effect on the output as the error is similar to the ideal traces.

Table 9: Unsymmetrical Traces Error Results

Load	Expected Current Draw	Current Draw	Expected Output	Output	% Error
300 Ω	10 mA	14.323 mA	35.8 mV	39.615 mV	10.65%
30 Ω	100 mA	100.224 mA	250.56 mV	273.77 mV	9.26%
10 Ω	300 mA	297.3 mA	743.25 mV	814.3 mV	9.56%
3.33 Ω	900 mA	895.12 mA	2.2378 V	2.509 V	12.11%

4.3.4 Non-Kelvin Connections

Kelvin connections separated high current PCB traces from low current traces. It places the measurement traces on the inside of the current sensing resistor pad, separated from the traces that are carrying current to the load on the opposite sides of the pad. Accuracy of the current sensing can be greatly affected if this is not done.

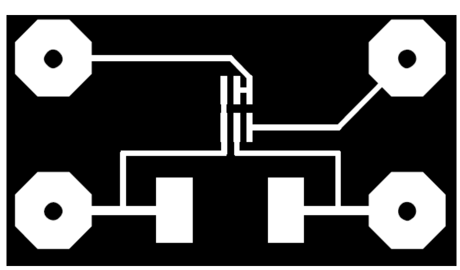


Figure 33: PCB Traces Using Non-Kelvin Connections

The below results show that not using kelvin connections can have a significant effect on the output of the current shunt monitor. It may add an additional 25% error to the output, which may cause unexpected operation throughout the rest of the system design.[3]

Load	Expected Current Draw	Current Draw	Expected Output	Output	% Error
300 Ω	10 mA	10.097 mA	25.242 mV	34.44 mV	36.4%
30 Ω	100 mA	99.513 mA	248.78 mV	325.34 mV	30.77%
10 Ω	300 mA	278.1 mA	695.25 mV	908.7 mV	30.7%
3.33 Ω	900 mA	883.7 mA	2.209 V	2.93 V	32.64%

Table 4: Non-Kelvin connections error results

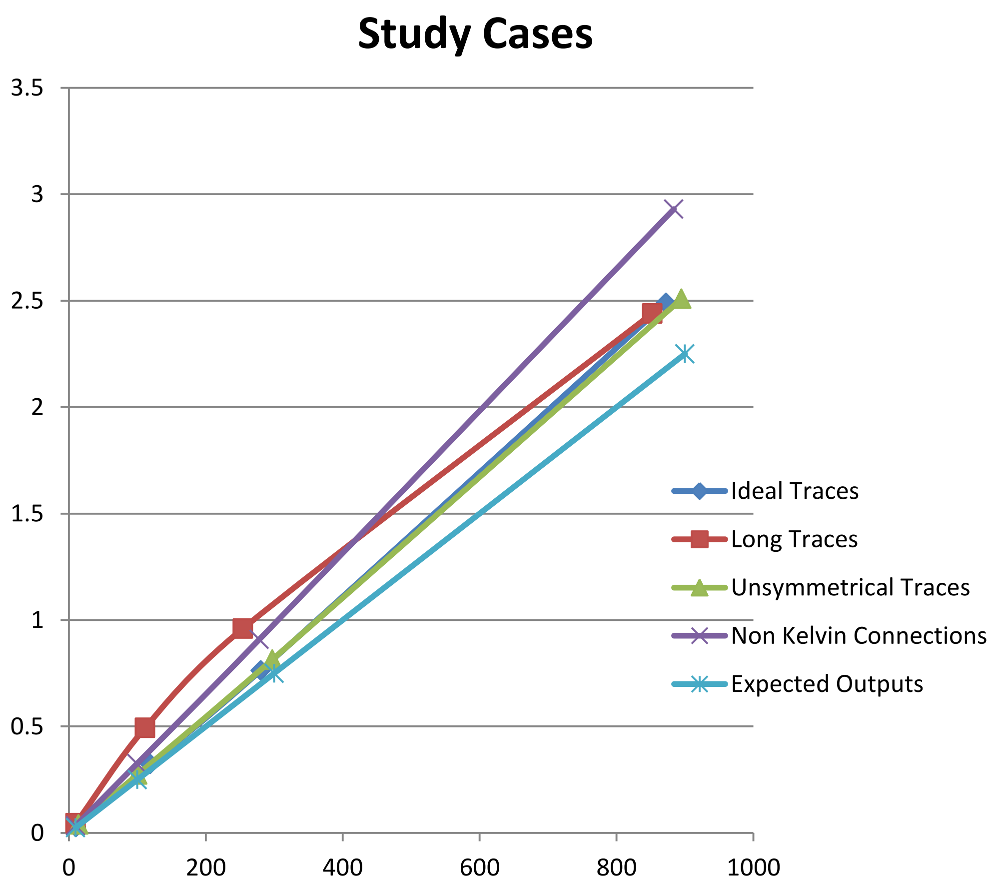


Figure 34: Graph of Study Cases v. Expected Output

4.3.5 Conclusion

From the study results, it is obvious that using the ideal case shown is necessary for successfully measuring current using resistive shunt current sensing. A graph is shown above, that displays all of the study cases against some calculated values, referred to as expected output in the key.

Chapter 5: Summary & Conclusions

5.1 Conclusion

Design Team 5 successfully designed two systems, one that simulated an emergency shutoff situation in a tablet PC and another that monitored current delivered to a cell phone with precision accuracy. Two designs were completed for the first application. The first did not utilize a microcontroller in the design. This worked as expected but did not produce the desired results, as it consumed high power and included too many parts. The circuit using a MSP430 microcontroller consumed much less power and was also faster than the other design. The biggest problem with the first application was fabricating PCBs and soldering the small parts to them. The design without the microprocessor used a flip-flop that never seemed to work as expected so a large DIP package flip flop was used, since it was the only one that seemed to work in the design. This could be greatly improved if other options for the switching were explored. Also, a different comparator could be used in the system to reduce power consumption. The design using a microcontroller never worked on a PCB. This problem could be explored by a future group that wishes to continue this project.

The second application also worked, with decent accuracy compared to a voltmeter in lab. Accurate current sensing using an Instrumentation Amplifier executed in this design. This application had many of the same issues as the first application. The PCB fabrication process made it difficult to test the design. Finally, the board with the MSP430 soldered on did not work, as in the first application.

Design Team 5 also successfully displayed how PCB traces in current sensing applications can have a significant negative affect on the output of the current sensing device. Four cases were tested: ideal PCB traces, long traces, unsymmetrical traces, and traces not using Kelvin connections. All of the non-ideal traces added significant error to the output of the current shunt monitor. This is very useful in the engineering industry since PCB errors in current sensing applications are very common.

5.1.1 Budget & Cost

Over the course of this project, the team was able to stay well under the $500 budget given by the college of engineering. By fabricating PCBs using chemical etching in the lab, the need to purchase professionally fabricated PCBs was eliminated. This saved time and budget. Also, since most of the components used were from Texas Instruments, could be easily sampled. Texas Instruments also provided any development tools necessary for completion of the project. The following tables show what part of the budget was used, what was sampled, and what the minimum cost would have been without the use of sampling.

Table 10: Budget Summary

Parts Purchased	Cost
10mΩ Resistor - WSL2512R0100FEA	$3.81
15mΩ Resistor - WSL2512R0150FEA	$2.54
25mΩ Resistor - WSL2512R0250FEA	$15.24
50mΩ Resistor - WSL2512R0500FEA	$3.81
Current Shunt Monitor - INA214	$11.25
Dual MOSFET - FDS8858CZ	$5.46
10Ω Potentiometer - 026TB32R100B1A1	$7.22
Part Total	$49.33
Shipping and Handling	$35.09
Total Budget Used	$84.42
Remaining Budget	$415.58

Table 11: Sample Summary with Cost of a Single Part Listed

Parts Sampled	Cost
Application One Sample Parts	
TLV3501	$3.36
MSP430F2013	$7.10
REF3325	$3.00
INA214	$4.50
SN74LS74	$0.67
390kΩ Resistor	$0.33
1µf Capacitor	$0.60
MSP430 Development Board and Debugger	$149.00
Total	$168.56
Application Two Sample Parts	
MSP430F2013 Microcontroller	$3.00
INA333 Instrumentation Amplifier	$5.25
Hitachi HD44780 LCD[4]	$12.00
MSP430 Development Board and Debugger	$149.00
Total	$169.25
PCB Design Study Sample Parts	
INA214	$9.00
Total	$9.00
Total Cost Saved by Sampling	$197.81*

* MSP430 Development Board and Debugger used in both projects so only one is needed also if only one of each part were ordered

Also, to fabricate a PCB professionally is $121 at Sunstone circuits with three day ground shipping. Design team 5 fabricated approximately 50 PCB for testing of the designs. Therefore, by fabricating PCBs using chemical etching, the team saved approximately $6,050.

5.2.2 Scheduling

Design Team 5 kept track of progress of the project on stayed on schedule using the Gantt chart. See chapter two which gives a detailed description of the Gantt chart for additional scheduling information.

Appendix 1: Individual Contributions

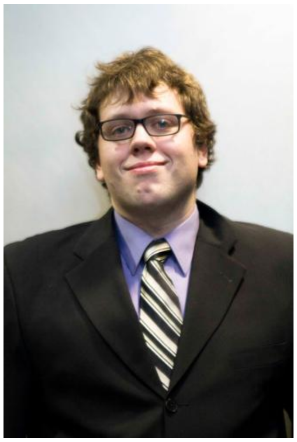
A1.1 Stephen England

Stephen England was the lead designer on application 1, primarily focusing on the simulation and design calculations. Stephen provided much of the research involved for current sensing applications, especially current sensing using current shunt monitors. He provided information on the differences between high side and low side, how specifications of the specific current shunt monitor affect the operation, and selecting a sense resistor based on the gain and restrictions of the current shunt monitor. Stephen used this information to select the current sensing method for the first application. Utilizing TINA-TI Spice, he accurately displayed how the current shunt monitor operates under different conditions, and showed how the gain of a current output IC can be controlled with a resistor while the voltage output have a set gain and the output is purely based on the output of the current shunt monitor. Stephen also aided in selecting all of the parts for the first application. This included finding a faster comparator than the original part in the proposal and a more accurate current shunt monitor. Also, when the original switch turned out to switch in milli-seconds, much slower than desired by the sponsor, Stephen further researched how to use a MOSFET transistor as a load switch. Upon finding an application note describing how to accomplish this, he selected a part with the two parts required included within the package. The part also had a very fast switching speed and a low on state resistance, required by the project specification. Also, he ran simulations to prove that each selection would work in the rest of the design. Finally, Stephen provided testing support and aided in some of the PCB designs and fabrications. Stephen provided a schematic early in the design process with a basic idea that was still seen in the final design.

A1.2 Joshua Myers

During this project, Joshua Myers was responsible for the overall design and testing of application one, as well as the fabrication methods of both applications. He began assisting in the design of application one early in the semester and determined that a process of creating our own low cost PCBs would be required. These PCBs were required to properly test and document the design of both applications due to the low resistance needed in monitoring current flow, as well as the accuracy required in the components which were selected. This was a very important aspect of the project primarily because professional PCBs take weeks to receive, and cost an upward of $100 each. Joshua decided on the primary method of chemical ethching which he performed numerous times throughout the semmester for every iteration of the design of each application. This chemical etching process was able to turn around OCBs in a matter of ours, at a very low cost to our design team. During the testing of each phase of the iteration, he was also responsible for the soldeirng, packing, and testing of each design. His testing led to the initial discovery of the need for a latching system in application one to stop continued osciliation in the switching circuitry. This lead to the design implementation of various flip-flops and logic to attempt to latch the output signal of application one. In the end, an inverting D flip-flop was used in the final design to latch the signal, allowing us to arrive on the final working design for application one. Joshua then continued with Kenji Aono to attempt to improve the speed and effeciany of application one by using a MSP430 micro controller to replace mcuh of the analog componenets. However, this improvement ended up becoming un-successful, primarilly due to difficulties in uploading the final program to the microcontroller. Joshua was also primarily in charge of designing the test fixtures and project display to be used at design day for both applications.

A1.3 Ryan Laderach



Ryan had the technical roles of being in charge of interfacing the MSP430 with an LCD, assist in the selection and ordering of the parts, and testing the designs and acquiring measurements for the parts. For application two, one of the requirements was to have the monitored current constantly displayed. For the role of interfacing the MSP430 with an LCD, an LCD was chosen that does not have a backlight to continue towards the requirement of low power consumption. He researched how the MSP430 can interface with an LCD in either 4-bit or 8-bit mode. In order to maintain the low cost priority of the application two, the decision was made to use a 2xx series MSP430 and 4 bit mode would be used. Ryan helped with testing the LCD when interfaced to the MSP430 to determine the limits of the system. The role of selecting and ordering of the parts was very crucial to reach the goal of having a working solution in the given time period. Ryan assisted with the selection by selecting the possible parts and determining if parts were feasible in reaching the final design of the project. With ordering parts Ryan saved money by timing the orders so that the shipping could be reduced to allow the parts to be received when PCB fabrication was taking place and optimize testing. For the last role of testing and acquiring measurements, Ryan assisted with overall testing and troubleshooting of designs. With the large amount of changes to the design made during the semester, every non-working design needed to have troubleshooting to quickly and effectively redesign to reach the final goal. He acquired measurements from designs in order to compare them to the sponsor's requirements and help in determining the parts that would need to be switched in order to allow the final goal to reach the customer's expectations.

A1.4 Kenji Aono

Kenji Aono had a technical role of lead software design & support, according to the final proposal. He satisfied this objective by creating the code for use with TI's MSP430 line of processors. Although the final code is less than 100 lines, and is relatively simple, it took several days of research to find the appropriate registers and values that required modification for proper operation. The final code produced allows for input of analog signals, which are converted a digital value using a Δ-∑ ADC, and displays that value on a LCD using the 4-bit HD44780 interface protocol. Kenji was also the lead designer for application two, and suggested the use of an instrumentation amplifier insead of a current shunt monitor to minimize the potential for errors introduced by input offset voltages to the integrated chip (IC). He also contributed to the design of application one, such as suggesting the use of a voltage reference IC to replace a simple voltage divider, which offers lower power consumption as well as higher accuracy in measurements. He also designed application one with a MSP430 to replace several of the smaller ICs such as the voltage reference, comparator, and D flip-flop. It is theorized that this method will allow for a smaller footprint, less power consumption, and lower cost of production. Kenji acted as the sole designer for the printed circuit board (PCB) layouts, using the EAGLE design software by CadSoft. The process of creating custom PCB layouts was nontrivial, and required the creation of a custom component library before the schematic was drawn in software. Kenji used the datasheets for all of the components, as well as measurements of the components to determine the required spacing of the pads for the surface mount to create the library. He also manually placed all of the components and routed traces between components to create PCB layout designs, which were transferred to a mask, for use in chemical etching of the PCBs. He also helped during the actual process of etching the boards, as well as testing of the boards once components had been soldered on.

Appendix 2: Literature & Website References

1 - Information found on Fairchild Semiconductor Application Note 1030

2 - INA333 Datasheet http://www.ti.com/lit/gpn/ina333

3 - Section 2.4 of http://focus.ti.com/lit/ml/slua366/slua366.pdf

4 - LCD Price from http://www.crestcomponents.com.au/lcd-displays-c-2.html?page=2&sort=3a

Links to Datasheets

SN74LS74 - http://focus.ti.com/lit/ds/symlink/sn74ls74a.pdf

INA 214 - http://focus.ti.com/lit/ds/symlink/ina214.pdf

TLV 3501 - http://focus.ti.com/lit/ds/symlink/ina214.pdf

TLV 3491 - http://focus.ti.com/lit/ds/symlink/tlv3491.pdf

FDS8858CZ - http://html.alldatasheet.com/html-pdf/197678/FAIRCHILD/FDS8858CZ/488/1/FDS8858CZ.html

REF 3325 - http://focus.ti.com/lit/ds/symlink/ref3325.pdf

MSP430x20 - http://focus.ti.com/lit/ds/symlink/msp430f2013.pdf

INA 333 - http://focus.ti.com/lit/ds/symlink/ina333.pdf

TPS 2033 - http://focus.ti.com/lit/ds/symlink/tps2033.pdf

TPS 1100 - http://focus.ti.com/lit/ds/symlink/tps2033.pdf

TPS 22907 - http://focus.ti.com/general/docs/lit/getliterature.tsp?genericPartNumber=tps22907&fileType=pdf

SN74AUP1G80 - http://focus.ti.com/general/docs/lit/getliterature.tsp?genericPartNumber=sn74aup1g80&fileType=pdf

Appendix 3: Technical Attachments

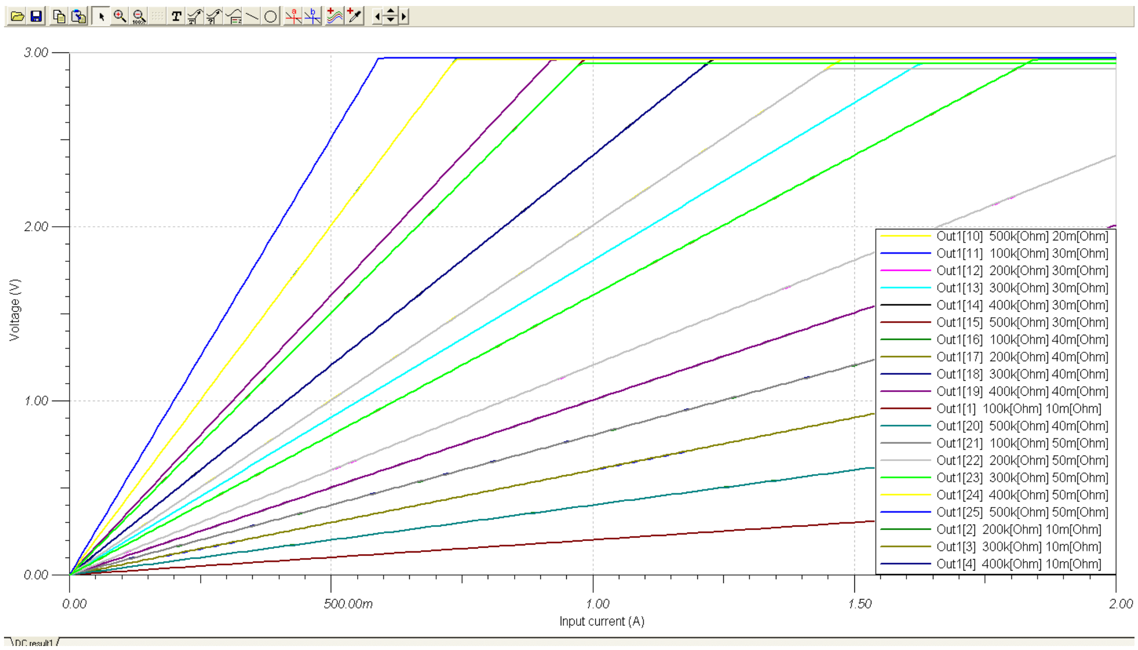


Figure 35: INA138 Simulations

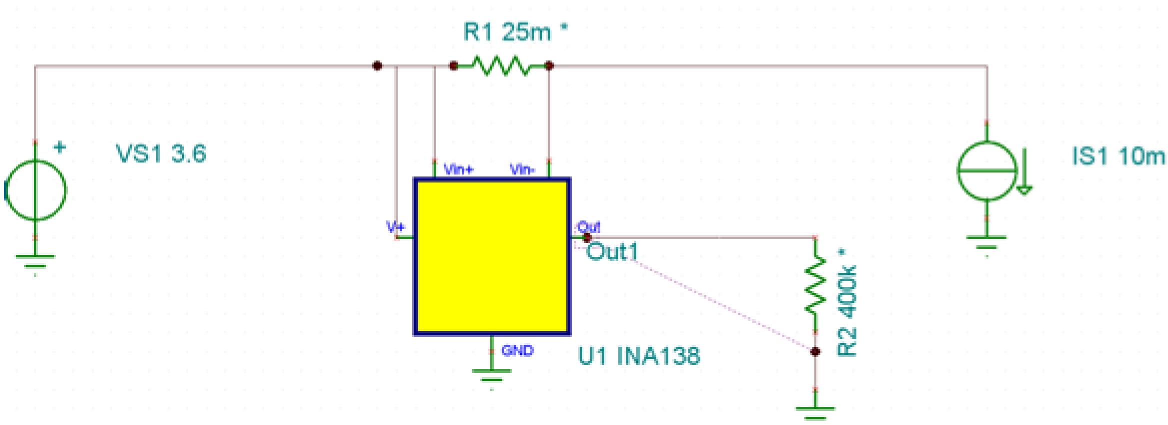


Figure 36: INA138 Schematic

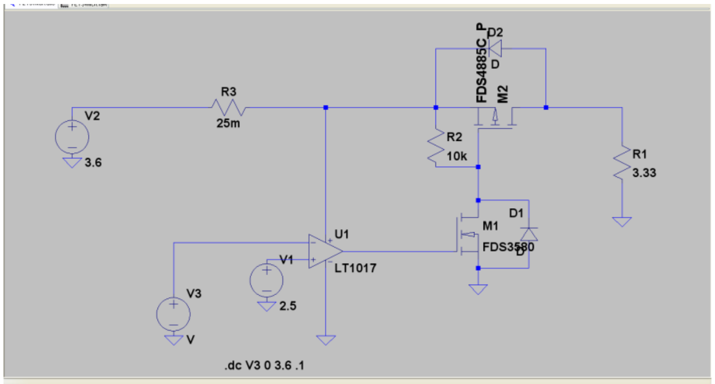


Figure 37: MOSFET Switch Schematic

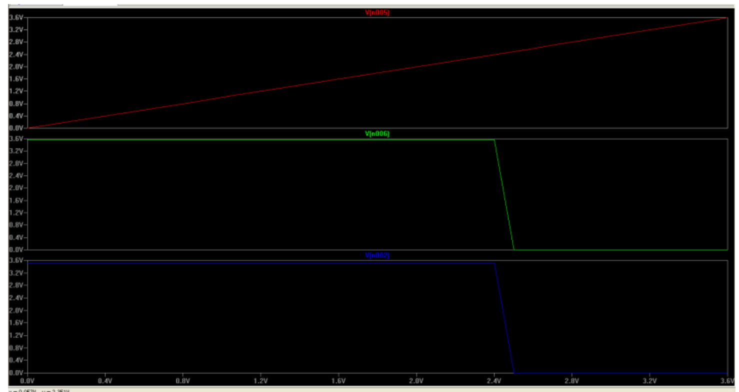


Figure 38: MOSFET Switch Simulation

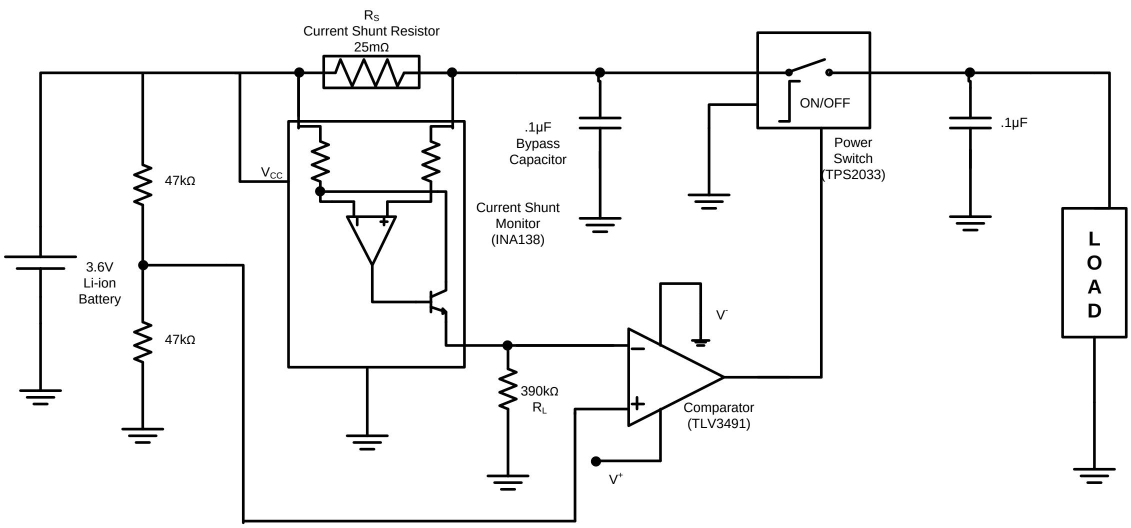


Figure 39: Proposed Application One Schematic

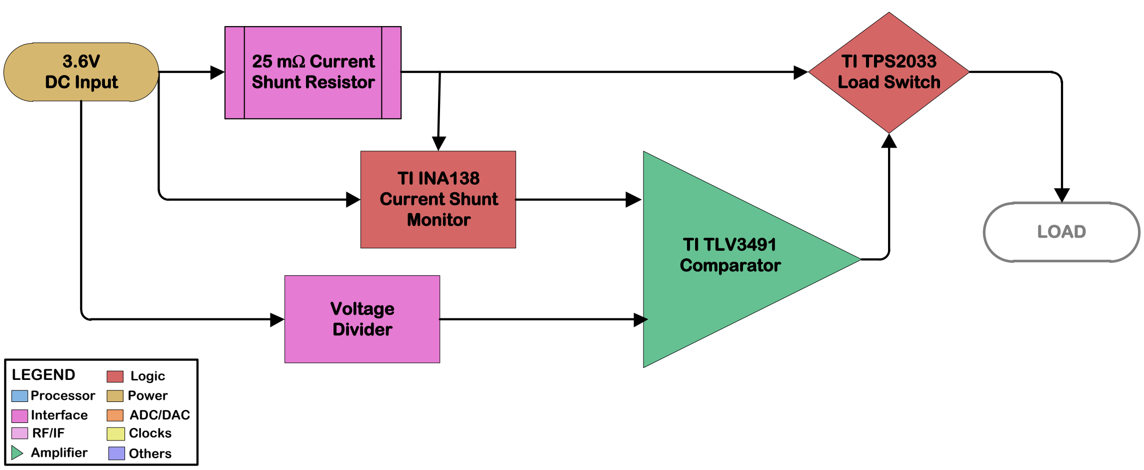


Figure 40: Proposed Application One Block Diagram

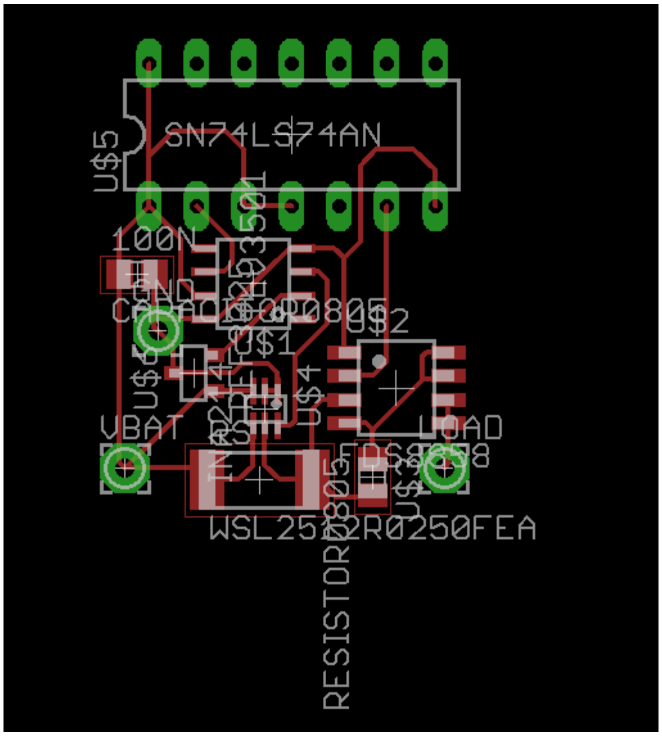


Figure 42: Application One Final

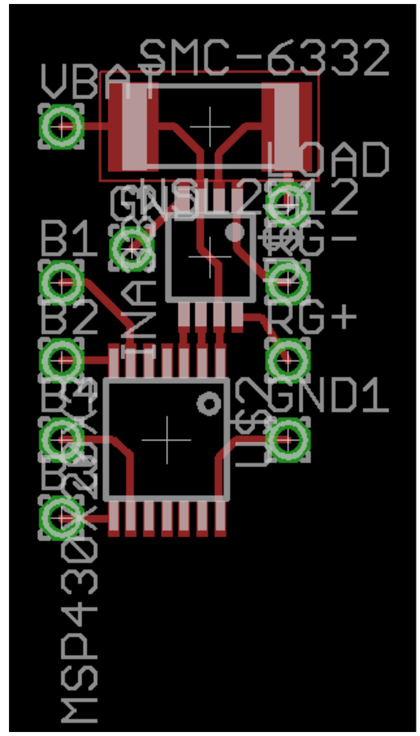


Figure 41: Application Two Final

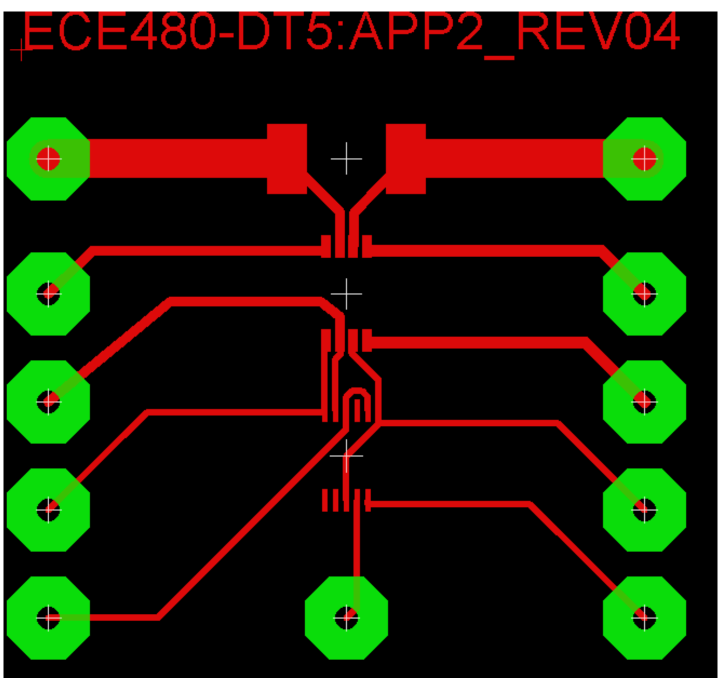


Figure 43: Application 2 Initial Testing

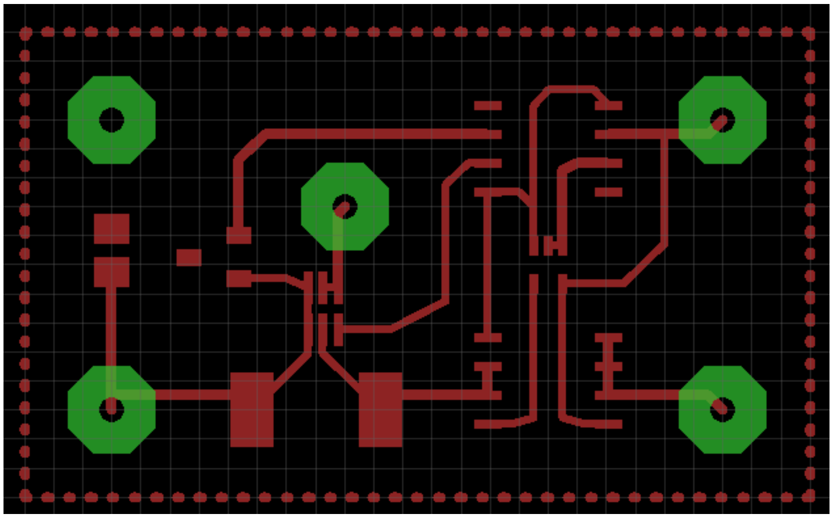


Figure 44: Application One Revision Five

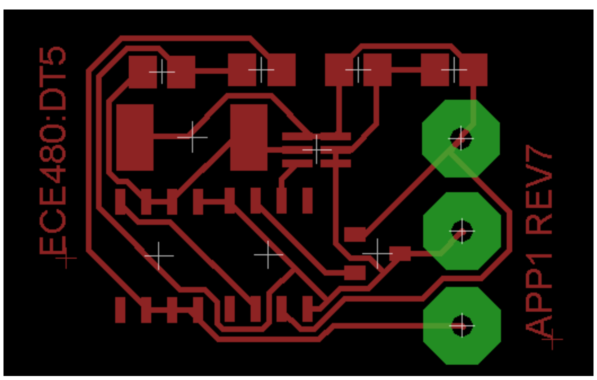


Figure 46: Application One Revision Seven

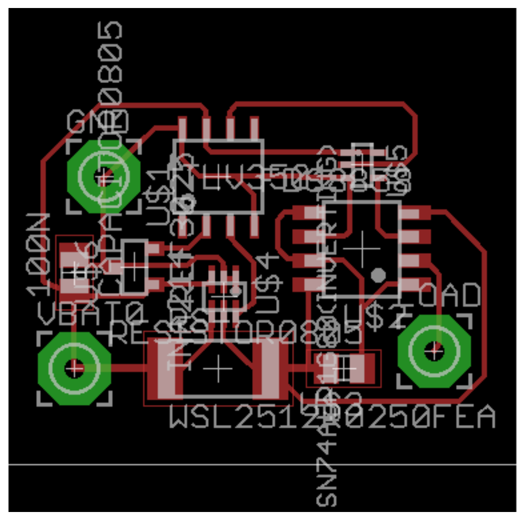


Figure 45: Application One Revision Ten

Additional PCB layout designs can be found on Design Team 5's website at:

http://www.egr.msu.edu/classes/ece480/capstone/spring11/group05/gallery.html