2008-2009

Biorobotics and Locomotion Lab

Final Report

Emily McAdams
TAM 491
ECE 2010
egm23@cornell.edu
919-619-7817
111 N. Haven Drive
Chapel Hill, NC 27516

Part I - 2008-2009 Overview

As a first-year member of the Ranger electronics team, my contributions to the lab throughout the year can be divided into three distinct projects. Upon joining the lab, the electronics team was in the midst of a complete restructuring of the electronics and software for Ranger. During the fall semester, the team was working on circuit schematic design and layout in an effort to have the boards for the new electronics completed and ordered by the end of the semester. My first project was the design of the RC module, which was one of several that would be used to create the different boards. While working on the schematic and layout for the RC module, I also became involved in building PC boards. By the end of the fall semester, the majority of the layouts for the new PC boards had been finished and the boards and parts were ordered. Throughout the spring semester, the majority of my time in the lab was spent building PC boards as necessary. In addition to building boards when needed, I also undertook the project of testing the new Bluetooth module for Ranger. On this project, I worked with Jehhal Liu and we spent time testing various features of the modules and testing communications with the module. During the 2008-2009 academic year, my work was not focused on a single project but I undertook several different projects contributing to the new electronics system for Ranger.

Part II - Projects

a. RC Module Schematic and Layout

After spending my first week in the lab learning the Eagle software, which is the software used in the lab for circuit schematic and layout, I began working on the RC Module. The RC module was one of the seven modules designed for the new "modular" electronics system. The function of the module was to provide a means of controlling three RC motors as well as measure parameters such as voltage and current in order to determine the power being used by the motor. This module would be connected with the A/D module and a MCU module to

complete the RC sensor board. This particular module went through many design changes throughout the course of the semester and eventually the decision was made that the RC sensor board would not be included in the first electronics system, but may be implemented in future versions. Although the module is not currently a part of Ranger's electronics, it is important to examine the problems we encountered and that remain so that they may be fixed for a future version of the electronics.

The first major issue that arose during the design of the RC module was a problem with the ground line. All of the modules in the electronics system were connected to the same ground line, but modules such as the Motor Control module and the RC module received power from the battery, while other modules on the same board were to received power from the CAN-BUS. This causes a problem because it could result in a discrepancy in the ground voltages on the different modules, affecting the measurements and potentially damaging chips. Our options for solving this problem included using opto-isolators between the two modules on a board or using sensors on the board to monitor the ground voltage. In the case of my board, we chose a different, more advanced current and voltage sensor to monitor the signals.

Another problem we encountered we discussed concerning the RC module was the problem of a voltage surge, as well as the ground voltage differential, from the RC motor that may damage the microcontroller on the board. The use of an opto-isolator between the RC module and MCU module was discussed as a possible remedy to this problem. I then spent time looking for an opto-isolator on Digikey that would meet our requirements. After choosing an opto-isolator to use on the board, I then had to make a library part, symbol to insert on both the schematic and layout representing the opto-isolator, which involved lots of time studying the dimensions and pin placement on the part and carefully designing the library entry.

As the fall semester was coming to an end, we were still working on resolving the various issues with the RC module. We were also trying to decide whether or not to continue trying to

solve the problems or move forward with the other electronics since it was not an essential module and address the RC module during future iterations of the new electronics system. One factor in our discussion was concern about the operation of our power measurement ICs, the MAX4210/4211. These ICs can only tolerate a maximum voltage of 28 volts and our goal for the operating voltage of Ranger is 24 volts. We were concerned about a possible voltage surge ruining our ICs and thus no longer allowing us to measure power consumption. Another consideration was the A/D inputs that would be required by the RC I/O and whether or not it was better to reserve these inputs for more important functions since the RC control was not necessary to proper functioning of the robot's electronics. Finally, we were still grappling with issues regarding ground voltage and isolation, and we had not yet come up with a viable solution. All of these factors along with the other problems already discussed, led to the decision put the RC module on hold and concentrate on completing the other modules and boards.

Although the module I worked on throughout the semester is not currently being used by the electronics system, the RC module will most likely become part of the next generation of electronics. In order to complete the module, it is important to understand and analyze the roadblocks and continue to consider possible solutions. One of the goals of the Ranger electronics project was to reduce power consumption and one key function of the RC module is to monitor the power consumption of the RC motors, so when this module is eventually incorporated into the electronics it will serve a useful purpose and the progress on the module should not be discarded.

b. Building Printed Circuit (PC) Boards

In order to reduce space on Ranger required by the electronics, the electronics team made an effort to design the PC boards to be as small as possible. In order to accomplish this goal, the majority of the parts used in the design are surface mount components and parts are to be placed

on both sides of the board. In order to attach surface mount components to the board, a specialized soldering technique referred to as surface mount soldering must be utilized. Surface mount soldering involves using a substance called solder paste, which is small beads of solder suspended in flux. To solder, small beads of the paste are squeezed onto the pads on the board and then the parts are placed on the solder paste. After the board is populated with parts, the board is then heated in a toaster oven until the solder reflows, which means the solder has formed an electrical connection between the component and the pad. Building an entire board can be time-consuming and tedious, but the only other option is to have the boards sent to an outside company to have them built.

During the fall semester, the CAN-USB board was finished and needed to be built. I had some experience with surface mount soldering and was interested in gaining more, so I volunteered to build the board. Even with Jason's assistance, building the CAN-USB board took us seven hours. Jason spent additional time in the subsequent days attaching the connectors and testing the board. Whether or not to build all of the boards ourselves was a frequent topic of discussion for the electronics team at the end of the fall semester. The best quote we received was $500/board + more for additional copies. For cost considerations, Jason chose to only have the most complicated board built by an outside company. This decision left us responsible for building the remaining boards. By the start of the spring semester, the majority of circuit designs were finished and the printed PCBs were beginning to arrive. Also at the start of the semester, I volunteered for the responsibility of building the boards as needed.

As we became more and more experienced at building boards, Jason developed an idea to streamline the process, save time, and make it easier for just one person to build the board. The company we had build the board asked us to prepare the parts for the board by printing labels for each part, adhering the labels to small anti-static bags, and placing the appropriate number of parts with some extra in the bags. We decided to use this technique in our own board-building

process. One advantage to using this method was that we were able to ensure that we had all of the necessary parts before we began building the board. The other main advantage was that while building the board, the parts are much easier to find by the builder and a second person is no longer needed to help find the parts. This step requires a few hours the day before building the board, but makes the actual board building much more efficient.

Another step in the board building process that Jason and I worked on standardizing was the reflow process. Instead of an expensive reflow oven, in our lab we use a standard toaster oven with a thermometer inserted through the top. Carefully monitoring and controlling the temperature reached by the board during the reflow process is very important. The ICs on the board are very sensitive to heat and it is important that they are not overheated by the oven, but it is also important that the board gets hot enough for all of the solder paste to properly reflow. We started standardizing our process by first looking at the data sheets for the solder in order to find the reflow temperature of the solder and the manufacturer's recommendations concerning optimal reflow process. We carefully tested our new process, recording reflow temperatures and the time the board spends in certain temperature ranges. The table in Appendix B compares the manufacturer's recommendations for solder reflow temperature and actual temperatures of our oven as well as reflow temperature and cooling time.

During the semester, I got the opportunity to share my acquired surface mount soldering skills with another student in the lab. I taught this student basic techniques and assisted him in building the board that he had designed. While trying to explain surface mount soldering to this student, I realized that it would be extremely useful if our lab had a thorough guide describing how to surface mount solder. While helping the student I took photos of the process and made careful notes in order to document the process later. The resulting documentation is included in this report in Appendix A. My intention is that a student attempting to learn how to surface mount solder has a guide complete with photos to guide him/her step-by-step through the

process.

I spent many of my hours in the lab during the spring semester building PC boards. In order to perform the "brain transplant" on Ranger and install the new electronics, it was necessary that we had boards that were complete. In that respect, the many hours I spent building boards contributed greatly to the progress of installing the new electronics into Ranger. Also, as a result of my work on this project throughout the semester I got the opportunity to share my skills with another student in the lab and I was able to create documentation for future lab students to refer to when building PC boards.

c. Bluetooth Testing

The third project I worked on during the year was testing the new Bluetooth module for Ranger. The new module was chosen with the intention that it would be ten times faster than the previous module. Myself and Jehhal Liu undertook the project of testing the new module and determining whether or not it was a real improvement over the previous version. Ranger uses Bluetooth capabilities to transmit data from the microcontrollers on the robot to a computer where the data can be further analyzed. The module we had been using was not able to transmit information as fast as we desired, so the new module was chosen with the intention of being able to transmit more data faster.

Our assignment was to test the new module to determine if indeed it worked better than the previous Bluetooth module. In order to begin testing, we first had to wire up a module for testing. I spent a day converting a USB cable into a power source and building a serial cable to connect the module to the computer. After the standalone module was appropriately wired, the next step was to use the computer's bluetooth to establish a connection to the module. Once we completed this task, we then tested the range of the Bluetooth by seeing how far a connection could be maintained between a computer and the Bluetooth. One problem that arose from this

test was that after the connection was broken, the computer could not reconnect to the Bluetooth. After establishing connection and testing range, we moved on to the task of testing the throughput and transmission rate of the module. Appendix C contains the specifications of the module according to the manufacturer.

In order to test these features, we knew we had to write a script of some kind that could connect to the module and transmit and receive data. After some exploration, we settled on using a small Java program that could connect to the module and also send a data. One advantage of using Java is its flexibility because it was possible for us to send either small or large amounts of data to the module. I was also inclined to choose Java because out of all programming language it is the one I am most familiar with and I decided that it would be the easiest way for us to quickly generate a program to test the module. One testing goal we had yet to accomplish by the end of the semester was transmitting large amounts of data and seeing how the module handled that data. It would not take too long to do this because Java can read a large file and send the contents to the module and we could observe how the module handled such a large amount of data.

Jehhal also came up with another idea for testing the Bluetooth using Matlab. Matlab has a toolbox that is capable of handling Bluetooth connections and it is another viable option for testing data transmission to the Bluetooth module. This is another potential direction for future testing concerning the Bluetooth module. As part of our testing we also built another standalone module for testing so that we did not have to rely on the internal Bluetooth in a computer. By having two standalone Bluetooth modules we were able to better simulate the Bluetooth connections that we will have with Ranger. Although our testing was not finished before the end of the semester, there are clear future threads for more research into the properties of the new Bluetooth. By establishing connections and sending and receiving some data we were able to establish that the Bluetooth may be used in the new electronics for Ranger.

Part III - Conclusion

This past fall was my first semester working in the Biorobotics and Locomotion lab. After a quick introduction, I was quickly immersed in the first of my three projects that I would undertake throughout the course of the academic year. My first semester was mostly spent working on the design for the RC module, one of the seven modules that would be combined to form the boards for the Ranger's new electronics system. I gained experience working with the Eagle software used for circuit schematic and layout and I also was a part of the design process identifying problems and working on possible solutions. Although the RC module is not a part of the current iteration of electronics for Ranger, it will be implemented in future version. Another project I spent considerable time on this semester was building PC boards. Throughout the spring semester, much of my time was spent building the new boards before the "brain transplant" could begin. I also was able to share my skills with another student and create a step-by-step guide to surface mount soldering for the lab. Finally during the spring semester I got involved with the testing the features of the new Bluetooth for Ranger. Overall this academic year, I gained valuable experience and built useful skills while contributing to the completion and installation of the new electronics system for Ranger.

Part IV – Acknowledgements

Thanks to Jason Cortell who was a great help and source of information throughout the year. He helped me learn the Eagle software and was also a big contributor to the building of the PC boards and the development of various surface mount soldering techniques. Also, thanks to Jehhal Liu who I collaborated with on the Bluetooth testing.

Part V - Appendices
Appendix A: Surface Mount Soldering Guide:

Biorobotics and Locomotion Lab:
Guide to Surface Mount Soldering
Compiled by Emily McAdams, 2008-2009

Step 1 - Preparation:

1 day before building the board:

1. Print Labels: The labels contain the part number, description of the part, and quantity needed for the particular board. Make sure to check the labels against the master part list to verify the information. Stick the labels onto small anti-static bags.

2. Find Parts: Fill each labeled bag with the appropriate number of parts, as indicated by the label. This step ensures that all the necessary parts for building the board are in the lab. If all the parts cannot be found, make a note and order the parts before the board-building begins. Passive components such as resistors and capacitors can be found in the black binder. Please be sure to note on the inventory list how many parts you remove. IC's and other parts can be found in a large blue storage drawer.

Before beginning to build:

1. Print board layout: Make sure you have copies of both the front and the back of the board. These documents are the map to matching parts with pads during board building.

2. Gather necessary supplies (Note: all of these supplies can be found in the drawer marked "solder" in the cabinet under the bench on the left hand side):
i. 1 Tube of Solder Paste
ii. 1 Tube of Flux, with the black cap
iii. Tweezers: Need at least one pair, the best are the ones with the curved tip
iv. Microscope: Some find it helpful to use when applying the solder paste and parts. Be sure to use the lab bench equipped with the microscope.
v. Board Stands: Small metal stands to prop the board up. These make it possible to solder on both sides.

3. Prepare the board: Choose which side to apply parts to first. The side with the fewest number of heavy parts and ICs is the best to start with. Apply the flux to the board and attach the stands.

 Warning: Before handling any parts, be sure to put on the electrostatic wrist band!

Step 2 - Apply Solder Paste:

1. Practice: Use a scrap sheet of paper and practice using the solder paste syringe to put down small beads of paste. Also, try practicing under the microscope to get a feel for the size of the bead you will need.

2. Applying the paste: Using the microscope, apply small beads of paste to the pads on the PC board. Try to apply from the top to the bottom choosing either to work left to right or right to left. This helps to avoid smudging of the paste. Try to keep the tip of the syringe clean to avoid smudging or applying too much paste!

3. Paste for IC pads: For areas where ICs will be placed, the pads are very close together. In these cases it is sufficient to put a line of paste along the pads hoping that the surface tension will help separate the solder.
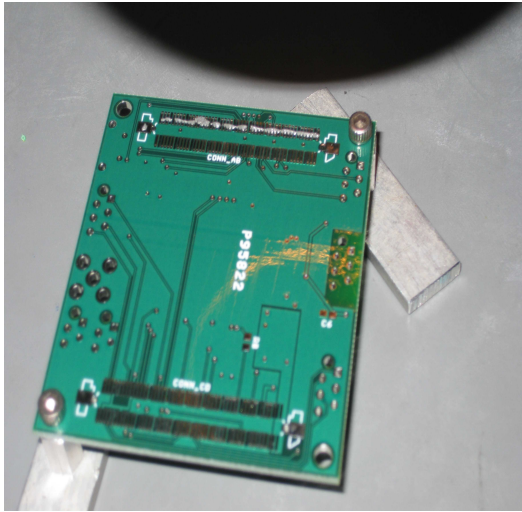
Picture 1: Board with solder paste applied
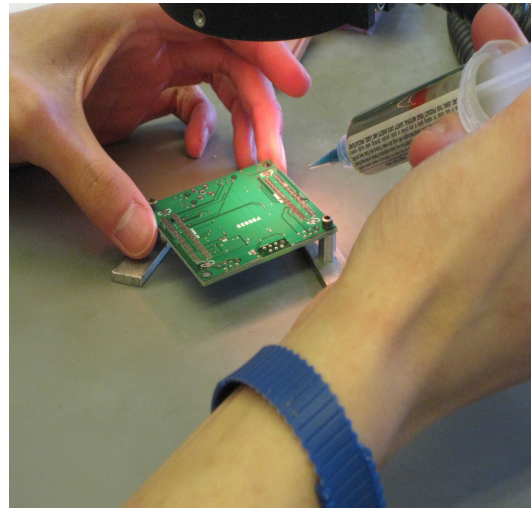
Picture 2: Applying solder paste to the board

⚠️ Warning: Solder paste contains lead. Make sure to wash hands well after use.

Step 3 - Populate the Board:

1. Getting Started: Once all of the pads on the board have solder paste, it is time to begin putting the parts down onto the board. As with applying the solder paste, it is recommended that you work from top to bottom and from one side to the other. Not all of the parts for the board will be surface mount components, during this step, only surface mount components will be placed on the board, the other parts will be added later.

2. Putting Parts on the Board: Consult the layout for the part number and match it with the labeled
bag. Remove the part and with the tweezers and place carefully on the pads. Pay attention to the direction in which the parts should be placed. For most resistors and capacitors the

orientation does not matter, but for parts such as LEDs, diodes, small ICs, etc. there should be some marking indicating the orientation of the part.

3. Putting larger ICs on the board: For some of the larger ICs, such as the microprocessor, the tweezers may not be large enough to get a good grip on the part. In these cases, it is usually the best to manually place the part onto the board, making sure to line up the pins carefully with all the pads.

Step 4 - Reflow:

1. Supplies: Once all the surface mount parts have been placed on the side of the board, it is time to "bake" the board.
i. Toaster Oven with thermometer through the top
ii. Stopwatch
iii. Test bead of solder on the board
iv. Flashlight

2. Baking the board:
   i. Take the tray out of the toaster and place the board on the tray, do not put it in the oven yet
   ii. Turn on the oven, use the bake setting, and set the temperature to 450 degrees
   iii. Monitor the temperature carefully, when the temperature inside the oven reaches 300 degrees, CAREFULLY place the tray with the board into the oven.
   iv. Start the timer, and using the flashlight monitor the test bead of solder. The solder will first melt and then it will "reflow" when it turns silver and draws back up into a bead. Check to see if solder on the other parts of the board has also reflowed.
   v. Once the solder has reflowed, continue to let the oven heat for 60 more seconds. After 60 seconds, turn off the oven, but leave the door closed for another 30 seconds.
   vi. Open the door and fan off the oven to bring the temperature down to 300 degrees. Once the temperature hits 300 degrees, it is okay to let the board cool down gradually.
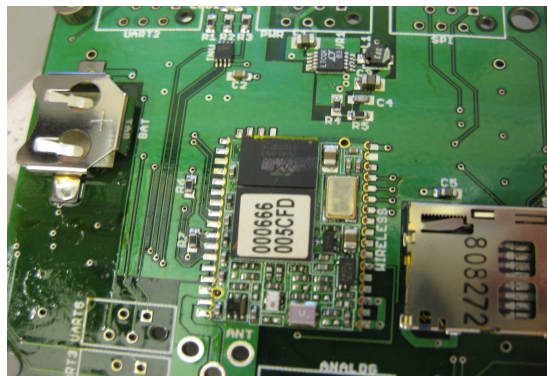

Picture 3: PC board after reflow

Warning: Be careful when removing the board from the toaster oven, the board and the PC stands may still be hot

Step 5 - Fix Solder Bridges:

1. Identify solder bridges: Solder bridges are areas on the board, usually along the pins of ICs where the solder has connected two pins. They must be removed in order for the board to function properly. Examine the board under the microscope carefully in order to find all of the bridges on the board.

2. Supplies: These supplies can be found in the same drawer as the solder paste
   i. Soldering Iron
   ii. Solder wick
   iii. Copper wick

3. Fix the bridges: Wet the sponge, heat the soldering iron to 750 degrees, and clean the tip of the iron. Place a small bead of solder wick on the tip of the iron and place the copper wick near the solder bridge. Heat the solder until it melts and the solder will adhere to the copper wick. Some solder bridges are so small, it may be necessary to apply more solder to the area in order to remove all of it.

4. Other issues: Solder bridges are not the only errors that can result from the reflow process. One example is not enough solder. If there is not enough solder, than a connection is not being made and it is necessary to add more solder to ensure the pin is connected to the board. Also, sometimes parts may shift during the reflow process. In these cases, it is necessary to remove existing solder and re-solder the part to the board.
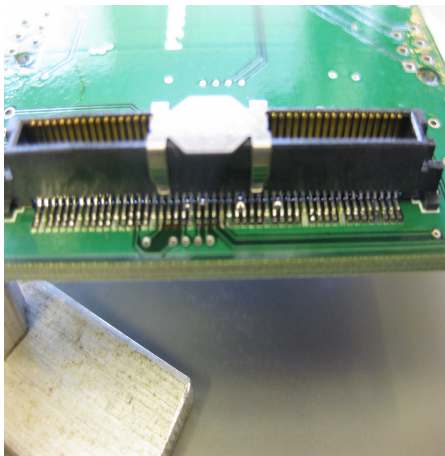
Picture 4: Example of resulting solder bridges
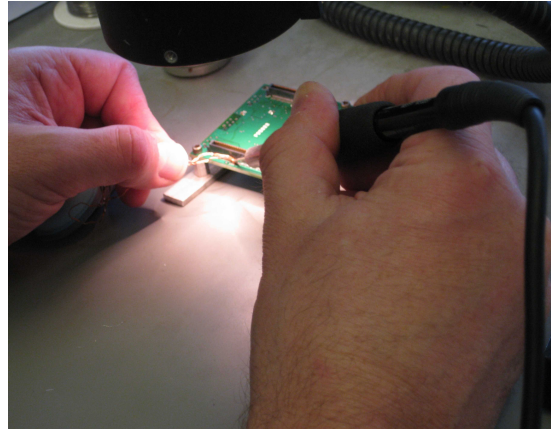
Picture 5: Repairing the solder bridges with the soldering iron

Warning: The copper wick heats up quickly, so hold by the wick container instead of the wick. Also, be aware of the location of the soldering iron in order to not burn yourself.

Step 6 – Finishing the Board:

1. Other side of the board: Repeat steps 2-5 for the other side of the board

2. Remaining parts: After both sides of the boards have all of the surface mount parts soldered on, it is time to solder on the remaining parts. These parts are usually connectors, jacks, etc. These parts should be soldered on using the soldering iron and the solder wick.

3. Testing: After all parts have been applied to the board including the connectors, the board can be connected to a power supply and if the board turns on, then at least minimally it is confirmed that the board is working.

Appendix B: Observed Solder Reflow Temperature Profiles

Time	Solder Data Sheet Recommendations**	Reflow Trial 1	Reflow Trial 2
0:00:00	300 degrees F	300 degrees F	300 degrees F
0:01:30	350 degrees F	330 degrees F	370 degrees F
0:03:00	450 degrees F	*Solder reflowed at 400 degrees F* 410 degrees F	*Solder reflowed at 400 degrees F* 400 degrees F
0:04:00	No Data	420 degrees F *Oven turned off*	430 degrees F *Oven turned off*
0:04:30	No Data	410 degrees F*	370 degrees F*
0:05:00	No Data	350 degrees F*	300 degrees F*
0:06:00	No Data	250 degrees F*	250 degrees F*

These numbers indicate the time required for the oven to cool down to a safe temperature for the board. It is important to minimize exposure of the board to temperatures above 300 degrees F, so we tried different methods of bringing down the temperature faster such as opening the door and fanning off the oven.

Recommendations from the data sheet for the Kester R276 solder paste

Appendix C: Roving Networks RN-24 Bluetooth Module Specifications

Features of the RN-24 Bluetooth Module as Specified by Manufacturer Roving Networks:

Operating Frequency: 2402 to 2480 MHz
Data transfers > 230kbps.
330' (100m) range, 12dBm output transmitter, -80dBm receiver sensitivity
Error correction for guaranteed delivery
UART or wireless configuration
Auto-connection mode