

gPod Accessible Blood Glucose Meter

Week 8
March 20-24
Matthew Bularzik

Work Completed

This week I mainly worked on the PCB board development and circuit layout modifications. I spent a good portion of the week working on the PCB board layout with deciding where the board will go in the hand held box and where the many holes should go in the board. The first version of the PCB board was meant to be mounted on the bottom of the hand held case. The problem with this layout was I believe the postholes and PCB screw holes were slightly off. This design also did not take into consideration the glucose sensor or the placement of the SP03 speech module.

For the second design the board placement was moved to the top of the board with most of the components facing into the box. The size of the board was increased from 90x100mm to 91x163mm which was the maximum size the hand held case allows. This now means that there will have to have four postholes through the PCB board and four PCB mounting screws. This size will allow for most of the circuit to be located in the lower portion of the board and the glucose sensor located at the very top. This will also allow for the LCD screen to be mounted on the PCB board since it requires four screws for mounting instead of the two the case allows. The LCD screen and glucose circuit will be mounted on the bottom of the PCB board so that they will be facing the top of the case. This will allow for the screen to line up with the screen slot and the glucose sensor to be the orientation. The circuit board had to be updated a third time due to the through holes for the posts could not be made large enough. Figure 2, below shows the latest PCB board layout with the corners cut out for the postholes.

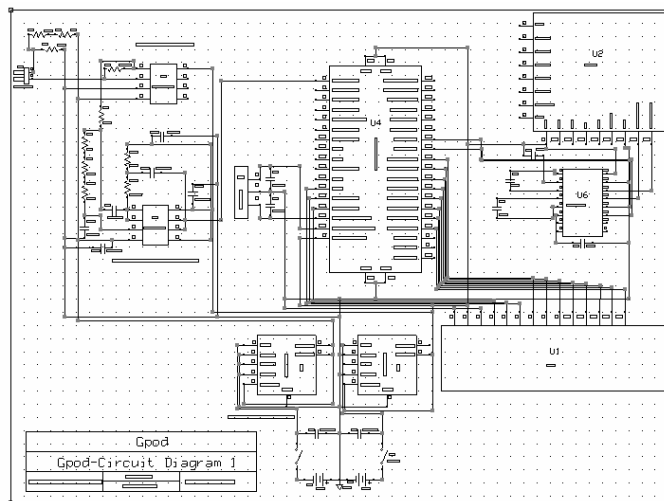


Figure 1: Circuit Schematic

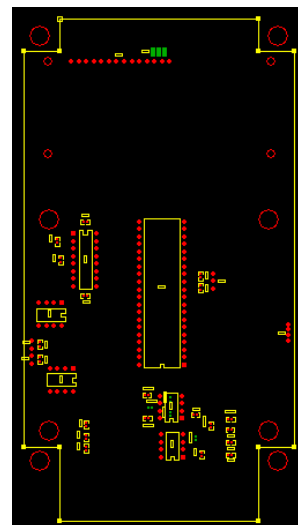


Figure 2: PCB Diagram

I updated the circuit schematic for the changes that have been made to the circuit as can be seen by, Figure 1 above. A switch was added to the circuit that had the capability to turn on and off both the +/- 5V batteries. A double pole single throw, DPST, switch will allow for this to happen.

To create the +/- 5V power sources from +/- 9V batteries voltage regulators will be needed. I researched voltage regulators for this process and found a few possibilities. The uA7805 and uA7905 were the first voltage regulators to be looked at since a circuit for dual sources was included in the applications. This circuit though was overkill and included both diodes and capacitors. While looking for a simpler circuit the MAX663 and MAX664 voltage regulators were found and they require no extra components. Free samples were ordered from Maxim for the two Max chips.

The case and its accessories were ordered from Okw. This included a hand held case, a case for the vial scanner, PCB mount screw, PCB mounting brackets, 9V battery leads, and a plastic window.

Future Work

I will continue to work on the PCB board layout. With Dave finishing the integration of the circuits early this week the circuit should then be finalized. This will allow me to make a final revision of the circuit schematic and then start drawing the traces on the PCB board. I will also place the final parts order list.

Project Review

This past week has been fairly productive. The glucose circuit is just finishing up and the accuracy is now being worked on. The LCD is fully integrated though we are hoping to get our new screen in soon to see about using 2 lines. The SP03 speech module still has a few bugs but should be done soon. The project should be fully integrated by the end of the week and ready for the next step. Total costs to date are \$985.67

Hours Worked

Hours worked on the project: 30 Hours