Kelley Welsh
James Finley

# Project report – Tri Board PCB V2

## Introduction

The Trio board that is currently being used for the Elcano project is in need of an update. It has some erroneous measurements on a few of the headers and a number of changes have been made in the design of how the board was to be used. For example, the sonar board connection was switched from two RJ45 connectors to a single connector, and the Arduino Nanos were to be swapped out with Micros instead.

## Changes

- The sonar connection to the Trio board was reduced down to one RJ-45
- Differential line amplifier was added to sonar board so a complimentary one was placed on the Trio board
- DAC was placed onto Trio board for possible future use
- UM7 connector was added for direct connection of the INU
- Arduino Micros will be used in place of the Nanos

## Project

Our project involved working with the stripped down version of original Trio PCB design and a partially completed schematic of the new board. Using an Excel spreadsheet, provided by Dr. Folsom, we began making the required connections between pins on the schematic. We also went through each of the required connections and made sure that everything seemed correct. When we found a potential problem with the wiring we contacted Dr. Folsom to ensure that the design was correct. Some of the connections in the Excel sheet were no longer relevant or required us to find an available pin that would fit its needs.

The new Trio board required multiple new components to be added to the schematic and board. The first was the UM7 INU that had a different connection than the UM6 and required us to create our own eagle component of the device. We then had to add a DAC to allow for future upgrades and use of actual analog signals from the Arduino and a differential line driver to incorporate the changes made to the sonar connection.

Once the schematic was completed and all the connections were double checked, we then continued our project by positioning the components on the board where the needed to be placed and so everything fit correctly, this including fixing the spacing for the GPS shield to attach to the Trio board as intended. In order to get the shield spaced properly we had to overlap 4 pins of

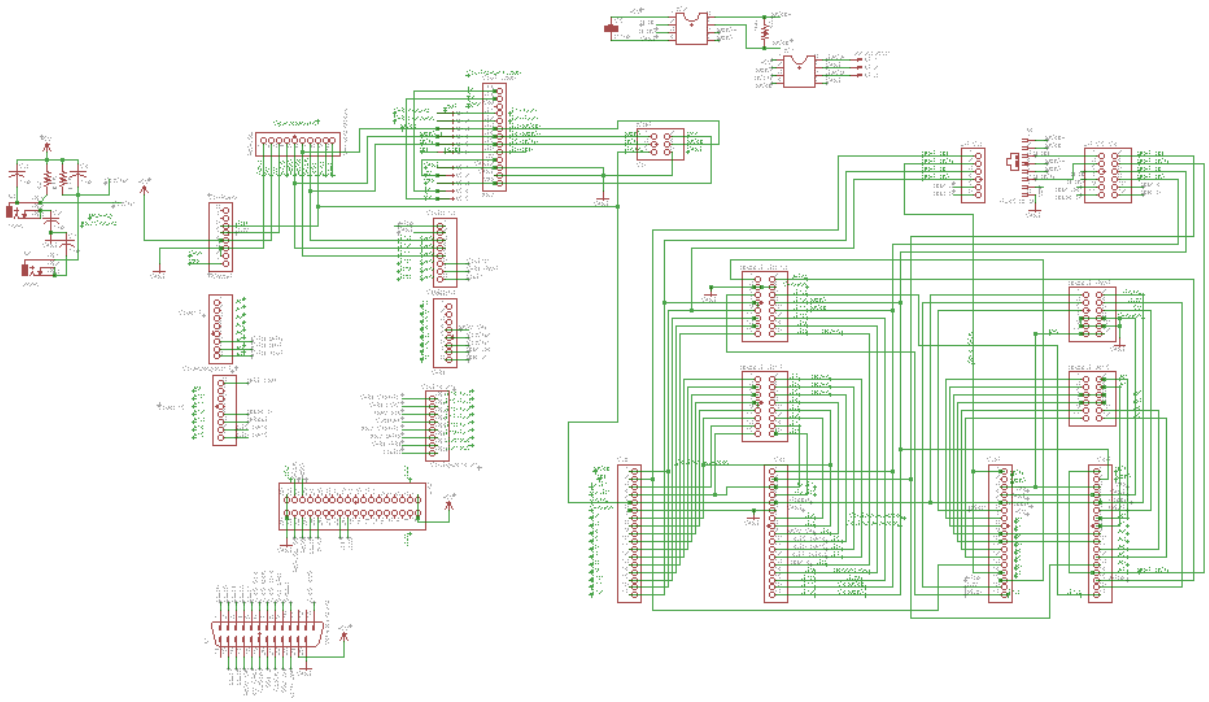
the Mega's bottom digital pin connector, those pins are not currently needed. After all of the components were correctly placed and connected, we then used auto-route to lay the actual PCB traces on the board. With auto-trace all of the trace sizes are small, but that should not be an issue since the Arduino's are not capable of sourcing or sinking enough current to require large traces. Once auto-routing had completed, we then double checked all of the connections to ensure that there was no mistakes and that all the connections were still correct.

## Summary

After spending a lot of time reviewing the schematic and spreadsheet to figure out how each component will be connected and then creating the needed eagle components, we were finally able to lay out the PCB. After completing the layout, we used auto-route to create our traces and reviewed the completed board for any errors.

Attached are the eagle board and schematic files for our final design.

**Trio V2 Schematic**



## Trio V2 Board Layout