

**ECE Design Challenge**

Section 2: PCB Design

Mitchell Stride

Mohammed Birama

Colin Hunt

Sakif Mohammed

PROTOTYPING

BLOCK DIAGRAM Fr. DRAW.IO and My Notes

For testing and proof of concept the team made a small prototype on a perfboard with components on hand and a few ordered pieces. It also ensured that our code would function properly on the new board.

Pics of the prototype that we made

The power supply was assembled as a daughter board for noise isolation and for safety so the mains voltage is separated from the rest of the controller. The operator can freely touch the motherboard without fear of shorting the main live wire.

A breakout mechanical relay board was chosen to test because the team had one and it could simulate the SSR. It was controlled via a simple switch that simulates the beagle bone I/O.

The rest of the board was assembled the same as the PCB would be made aside from the different component packeges.

BOARD DESIGN

Design Justification

Please see attached schematic and board.

* **Power supply:** If the board was fitted in an enclosure an IEC connector can be used to bring a solid connection of mains voltage to the board. Then internally it would be connected via screw terminals or a molex connector for vibration reasons and so the board would still be removable.

Due to time constraints it was not feasible to order components and design a SMPS for the board so we choose to design a linear power supply. Given more time a SMPS could be created to make the board even more compact and efficient but the team decided it was best to go with a tried, proven design for this critical function

* **ON-OFF CTRL:** We created this circuit with the idea to control the entire PCB from the pod’s BeagleBone Black I/O. So 1x2 screw terminal brings in one GPIO and GND from the board to energize a small DC SSR. When the load circuit is closed this connects the 9V rail to the rest of the circuit.
* **Microcontroller:** Several options were compared for our main microcontroller...

|  |  |  |
| --- | --- | --- |
| MC | Pros | Cons |
| ATTiny Series | Most Compact | Limited I/O |
| ATMEGA32u4 | HID, USB2 | No DIP package for prototyping |
| ATMEGA328P | Scalable, DIP or TQFP32, | Slightly larger package |

Given that this PCB was a one off board the cost difference was negligible so that eliminated that factor from consideration. This design could have been done via analog means or purely hardware so the implementation of a MC was overkill. So we assumed the MC was implemented for alternate function/scalability. This led us to select the ATMEGA328P for its plentiful I/O, and SPI, I2C, and UART buses. This extra N/C pins can be used with an application like a sensor communicating via I2C for motor control.

* **Motor Controller Circuit:** The team decided there was no point in using the common H-Bridge setup with 4 MOSFETs and 2 BJTs for the MCC because this board is not for mass production. Another viable option was the simple single NPN BJT or MOSFET with a flyback diode. But the group ultimately chose the superior option of the L293NE IC for its proven reliability and scalable support for a second dc motor.
  + This gives our circuit the ability to attach any small DC motor from ranges 3-12VDC. The last step would be to determine the teeth ratio for attaching a pinion for a rack or gear.