Senior Design — SDMAY24-14

Periodic Report 2-02

i281/e Hardware Implementation

Reporting Period: January 28th through February 9th

Project Personnel

Dr. Alexander Stoytchev (Advisor/Client)	Daryl Damman
Logan Lee	Grant Nordling
Braxton Rokos	Gavin Tersteeg

Progress Summary

This period's focus was the layout for the PCB implementation. Before we could start properly creating PCBs, we needed to know how the CPU was going to be built. We spent most of the first week brainstorming different methods until we landed on a design that would both fulfill our requirements and our client's expectations. The next week, we focused more on designing and laying out our design. We started the initial PCB layout for many of the modules. We sent in a MUX to our PCB manufacturer to test out their process and ensure we want to move forward with them. Some of our accomplishments for this period were designing and implementing the write-back and video card modules. Additionally, we started adding the breadboards permanently to our wooden board which will be used for final testing in the upcoming period.

Decisions Made

1. Spent much time working on the overall PCB layout so that we can work on the individual PCBs boards.
    a. Started with the initial plan of making 6 PCB islands and using connectors in between each section.
    b. Changed to a better modular and movable design with each component of the design getting a module and all attached to a back PCB with connections between each module.

Past Week Accomplishments

Individual

- Daryl Damman:
    - Updated multiplexer and register file schematics for current implementation.
    - Fix website UI bug.
    - Create DXF file for ABE waterjet for breadboard mounting boards.
    - Alongside Gavin, standardize footprints and components for PCB implementation.
    - Order a sample MUX PCB (set of 5).
    - Perform parts inventory and labelling.
    - Update branding for i281e and related components.
    - Wire the second half of the 7-segment displays on the video card.
- Logan Lee:
    - Created schematics for the writeback module and implemented the writeback module in its breadboard version.
    - Started work on learning PCB design on KiCad.
    - Updated schematics to match the breadboard prototype: Program Counter, Writeback module.

    - Started layout of Program Counter PCB.
- Grant Nordling:
    - Updated the schematic for the control table.
    - Created financial report in both Word and Excel forms, logging total financial spending throughout our work.
    - Created the initial PCB design for the control table.
- Braxton Rokos
    - Developed video card schematic.
    - Built the video card on the breadboards.
    - Updated the ALU schematic for PCB use.
    - Started the layout for the PCB of the ALU.
- Gavin Tersteeg:
    - Finished DMEM RAM and decoding circuitry.
    - Created schematic for PCB front panel.
    - Continued work on DOS/281.
    - Wired up and tested the current configuration of the processor.
    - Started work on hooking up UART to DMEM module.

Team

1. Started PCB brainstorming layouts.
2. Started layouts of individual PCB modules
3. Assembled the Breadboard protype for testing on a backboard.

Individual contributions

Name	Individual Contributions *(Short List)*	Weekly Hours	Cumulative Hours (Starting from Biweekly 2)
Daryl Damman	Complete PCB for MUX, Register file PCB layout, and waterjet prep	45	45
Logan Lee	Breadboard Write back module Program Counter PCB Layout	20	20
Grant Nordling	control table PCB layout created total financial report	Week 1: 2 Week 2: 5	7
Braxton Rokos	Breadboard Video Card Module, PCB ALU Module	27	27
Gavin Tersteeg	Work on DMEM, Physial Assemby, Various Software Projects, UART Integration	30	30

It shall be noted that the hours above are estimated within reason. Appropriate hour tracking shall be performed following this period. The first hour spent during our twice-weekly meetings is not included in this report's estimate.

- Team Meeting Hours: 6 hours

Upcoming Actions

Individual

- Daryl Damman
    - Finish Register File PCB
- Logan Lee:
    - Finish Program Counter PCB
    - Start the combine Data Memory Schematic and PCB
    - Update design iterations with new versions of the modules.
- Grant Nordling:
    - Finish Control Table PCB layout.
    - Begin documentation on remaining final deliverables.
- Braxton Rokos:
    - Finish ALU PCB design.
    - write documentation based on changes and decisions made.
- Gavin Tersteeg
    - Testing and debugging features of the breadboard implementation.
    - Finalize components of DOS281

Team

1. Testing the breadboard protype with everything assembled.

Plans for Next Reporting Period

- Create a method to log hours going forward (i281e Minutes).
- Create an estimate for the total cost of all PCBs with connectors and chips.
- Order all PCBs.
- Order all needed parts for two processors.

Meeting Notes

Jan 29

-Showcased PCB layout to Alex. 7-segment display is suggested to be on the DMEM module instead of on the interface panel.

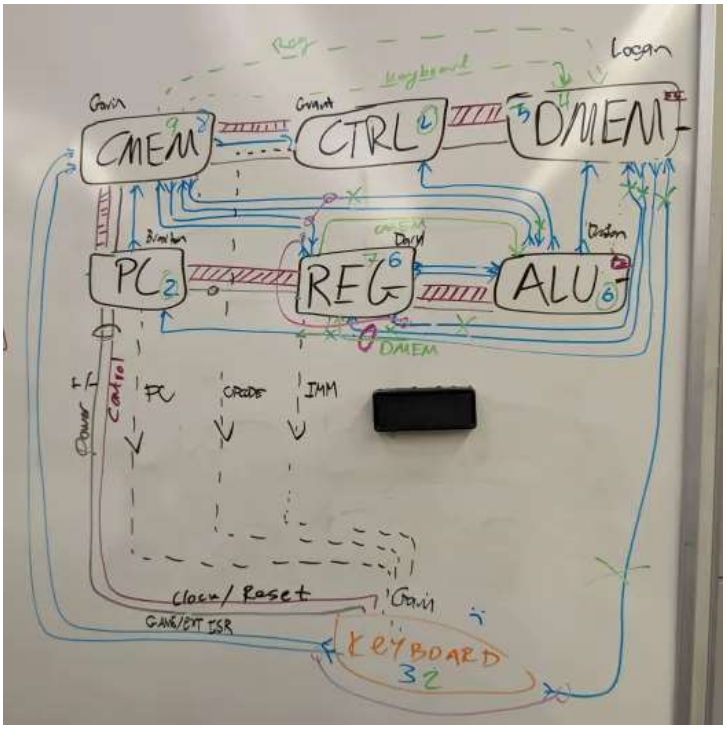
-Streamline control and Power throughout all PCBs. Power Supply is included in IP/SWB. Bus for data will still be scattered

-Current layout may need to be adjusted for readability purposes. Rectangle is subjective.

-CMEM will contain “write-back” module and DMEM will contain boot device/memory

-Standardizer connection for all connections. Flags are located on the ALU and need to travel to the control table. Special Cable? Same as power cable?

-Many concerns with how the final wiring will look presentable. With merged multiplexers, the silkscreen can illustrate where the MUX sits. Still operating on 8-bit connections using 16-bit ribbon cable.



-This version would require 16 wires plus a lot of daisy chained cables.

-To counter the issues of connecting wires incorrectly, we could have different colors for input and ouput on-board connectors

-Other Option: Have trust in the user and have the TA check over the boards prior to powering up the board. Use the silkscreen for labeling

Feb 2

-Redesigned the board once more to reduce the number of wires needed.

-This model uses a back motherboard to route all the signals

-There are breakout cable connectors for each signal at the bottom

-Each module hooks into the motherboard via Zif connectors.

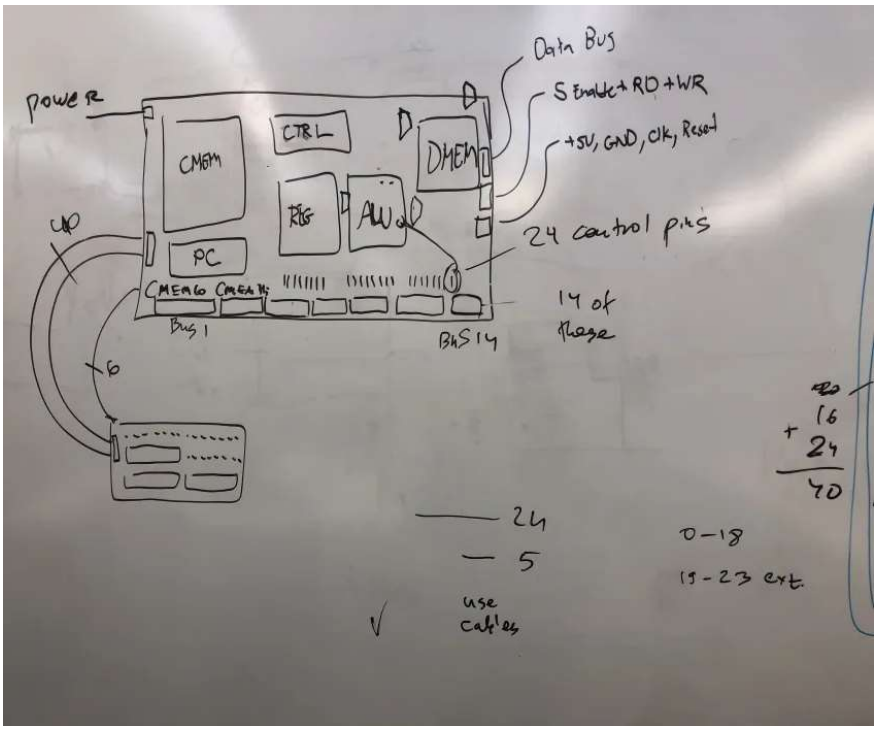
-Modules can be removed and added with ease

-This model would allow students to hook up their designs to the board

-This version unmerges the MUXs from modules

-The maximum PCB dimensions that are manufacturable are 400mm x 500mm

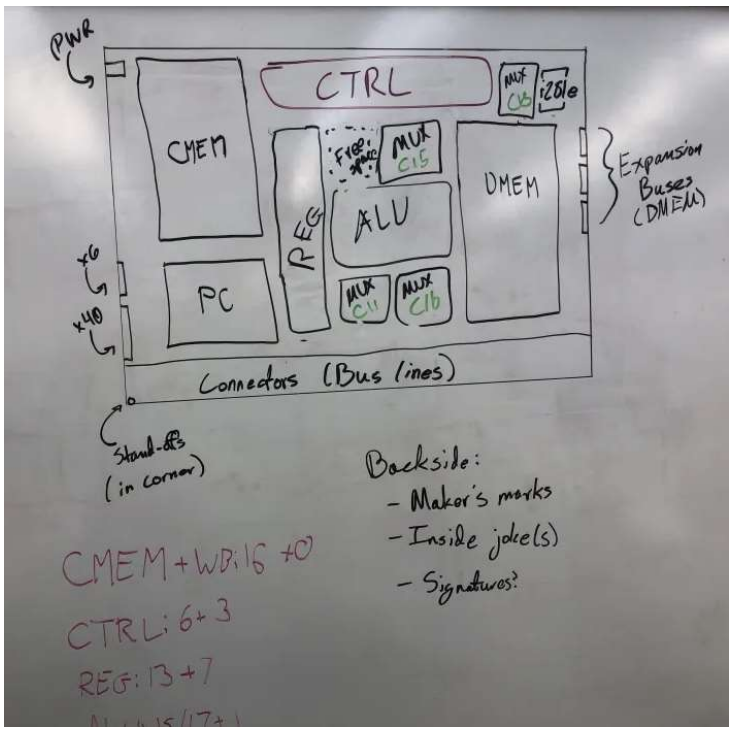
-Need to create a quote for the total cost of a PCB with the Chips and connectors factored in



Feb 5

-This meeting we decided the general layout for the PCBs.

-This is the layout that the final product will follow and reference



Feb 9

-Workshopped KiCad layout

-Updated various schematics and layouts

-Set up breadboard version of project on a wooden board for final testing

-Wrote weekly report