Magnetic Field Camera

Milestone 3

Aditya Vema Reddy Kesari
Ashwajit Singh
NV Navaneeth Rajesh
Raunak Mukherjee
Suchet Gopal

Outline

1. Feedback from milestone 2
2. Progress updates
3. Proof of concept/ Block Diagrams
4. PCB
5. CAD Design and Power management
6. Critical Tasks and Key Risks
7. Future plans

01
Feedback

Feedback

Pictures of sensors — We added pictures of components to the presentation

Component choice — We explored the reviewer’s suggestions, particularly for the FPGA, and settled on the current choice

Clearer schematics — We created a hierarchy within the schematic and labelled nets for easier readability

02
Progress Updates

CAD design

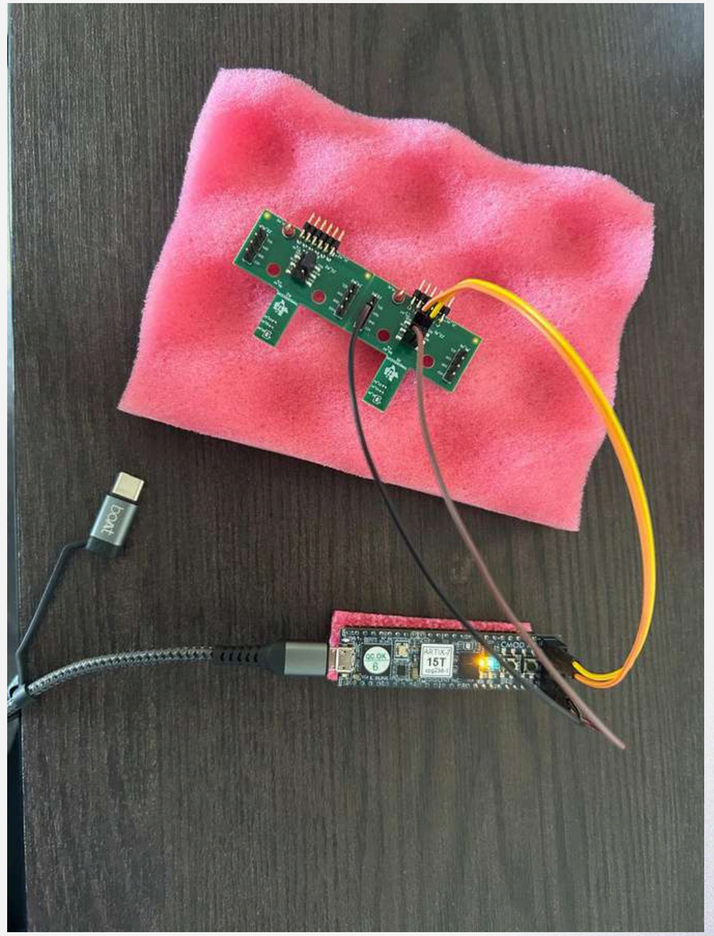
Progress :

- We have a complete CAD design, and will receive approval for 3D printing this week
- We had to make changes to our initial design ensure the weight of the motors could be supported by our 3D printed parts
- We still employ an r-theta-z system, but changed the positioning of the motors to ensure a more stable design

Breakout board

Progress :

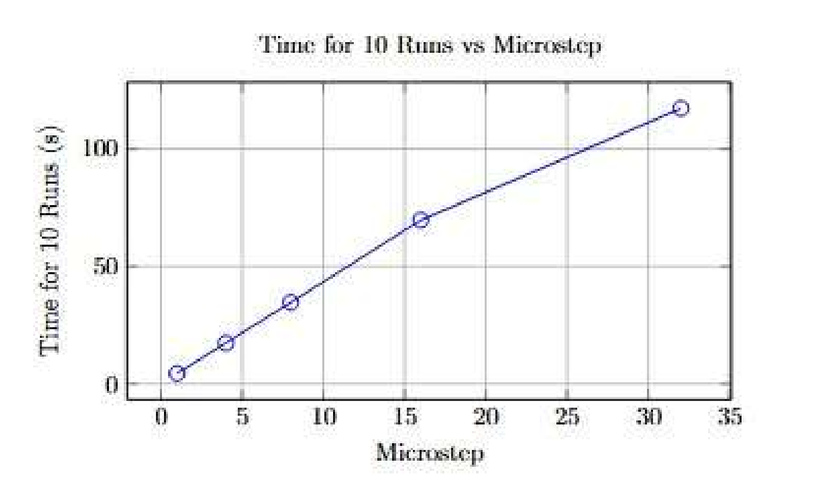
- Unfortunately we received the breakout board much later than expected (earlier this week) so we have not had time to debug our I2C interface from the FPGA to the sensor.
- This is not a very complicated task and should be done by next week.
- We will also have our sensor PCB back from assembly by next week to compare with the breakout board testing results.

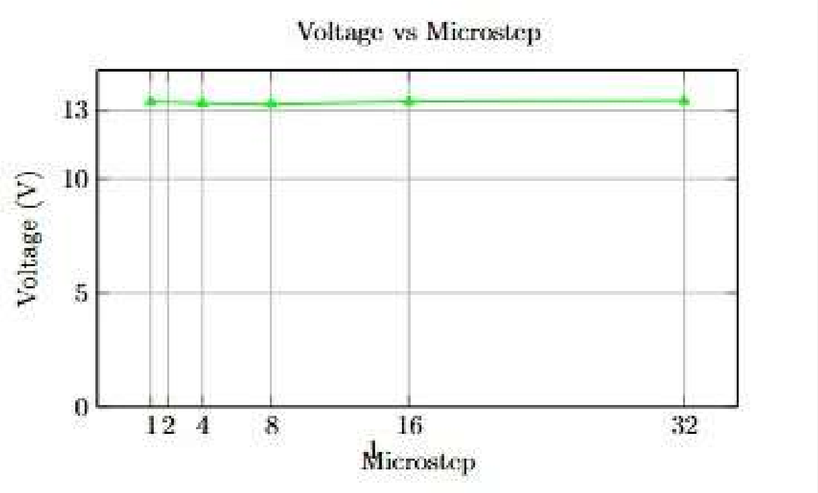


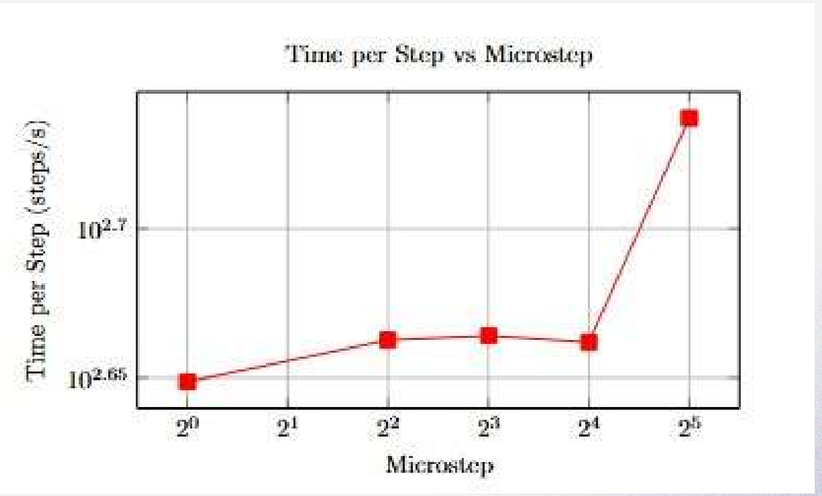
Rig movement (Motors)

Progress :

- Tested motor for multiple runs to check for cumulative error.
- Observed < 5 degree error thus concluding that the error is non cumulative.
- Measured time per step for each microstep.







GUI Design

- Implemented several techniques of visualisation available on matplotlib including 2D and 3D quiver plots, heat maps and contour plots.
- Finalised the design to be a 3D zoomable and rotatable heat map along with a 2D heat map
  of any plane specified by user
- Aim to implement a 2D heat map cum quiver plot of the plane parellel to the current orientation of the 3D heat map as default
- Also developed a web page instead of a window to improve speed of interface

Deviations from original plan

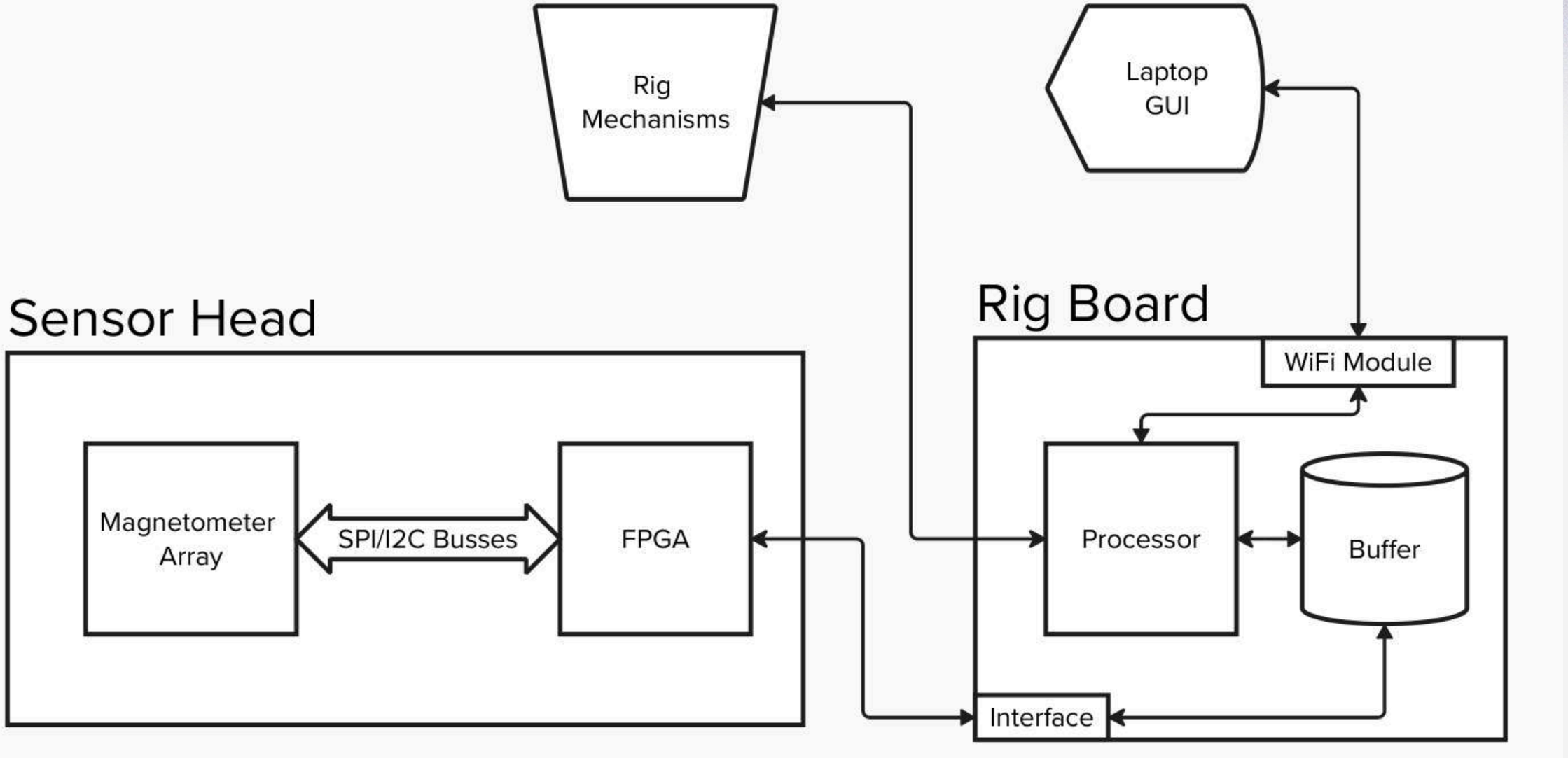
- We made changes to the initial CAD design after receiving and testing the motors, to ensure that it is more stable

- Obtaining steady SPI readout between the RPi and the CMOD A7 took longer than expected, as it was often very erratic

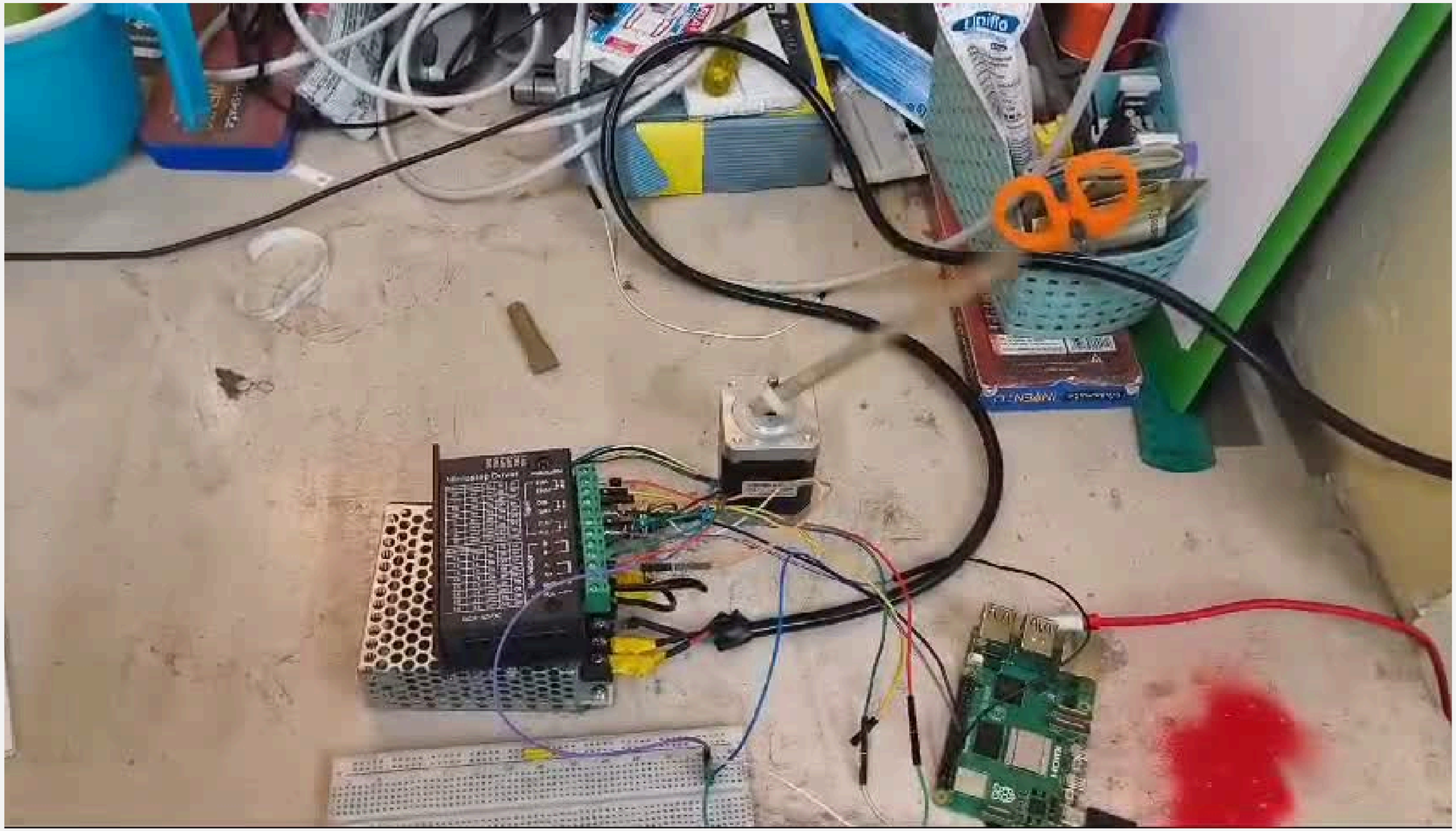
- Testing of TMAG3001 sensors delayed due to delayed arrival of the breakout board

03
Proof of concept/ Block Diagrams

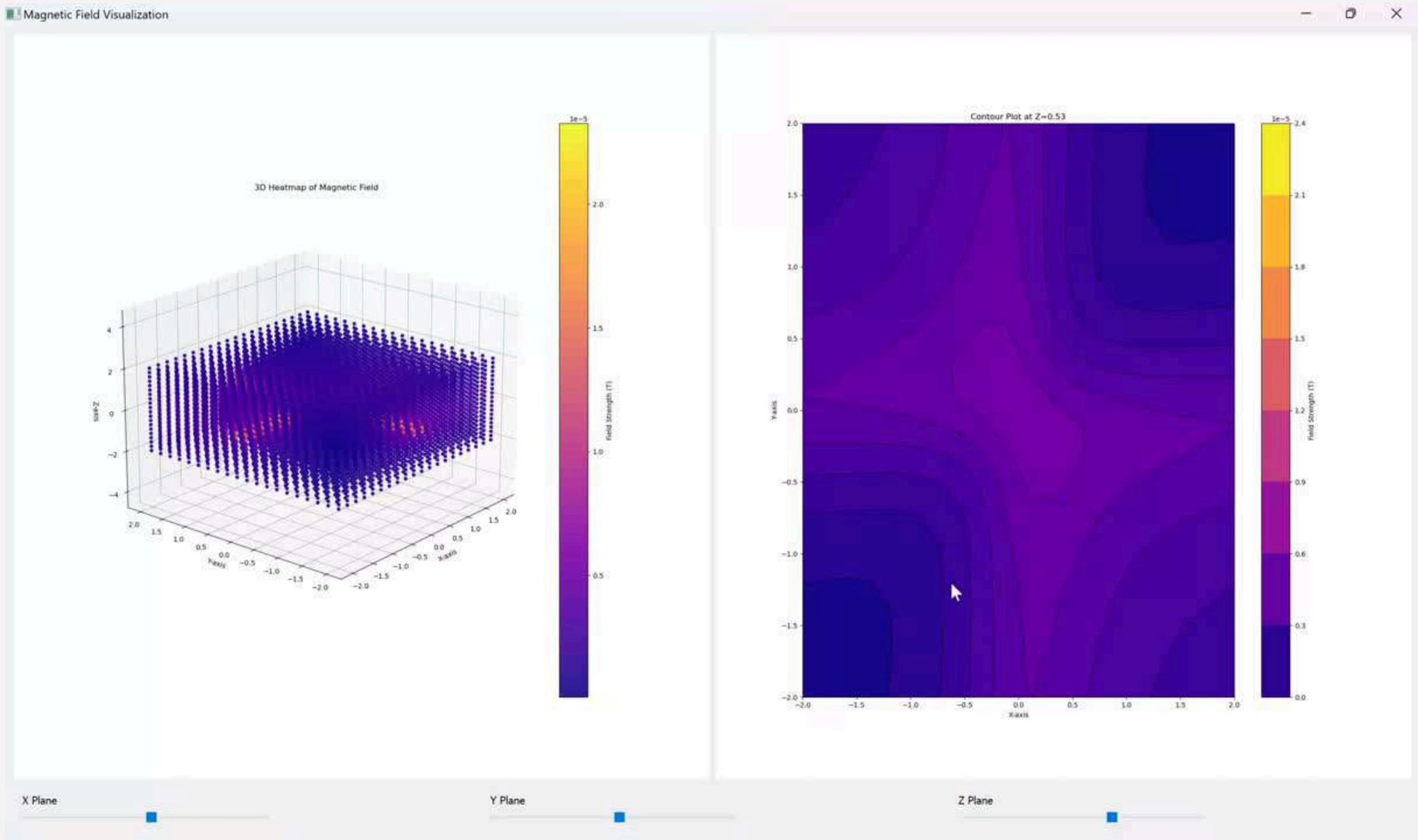
Overall block diagram



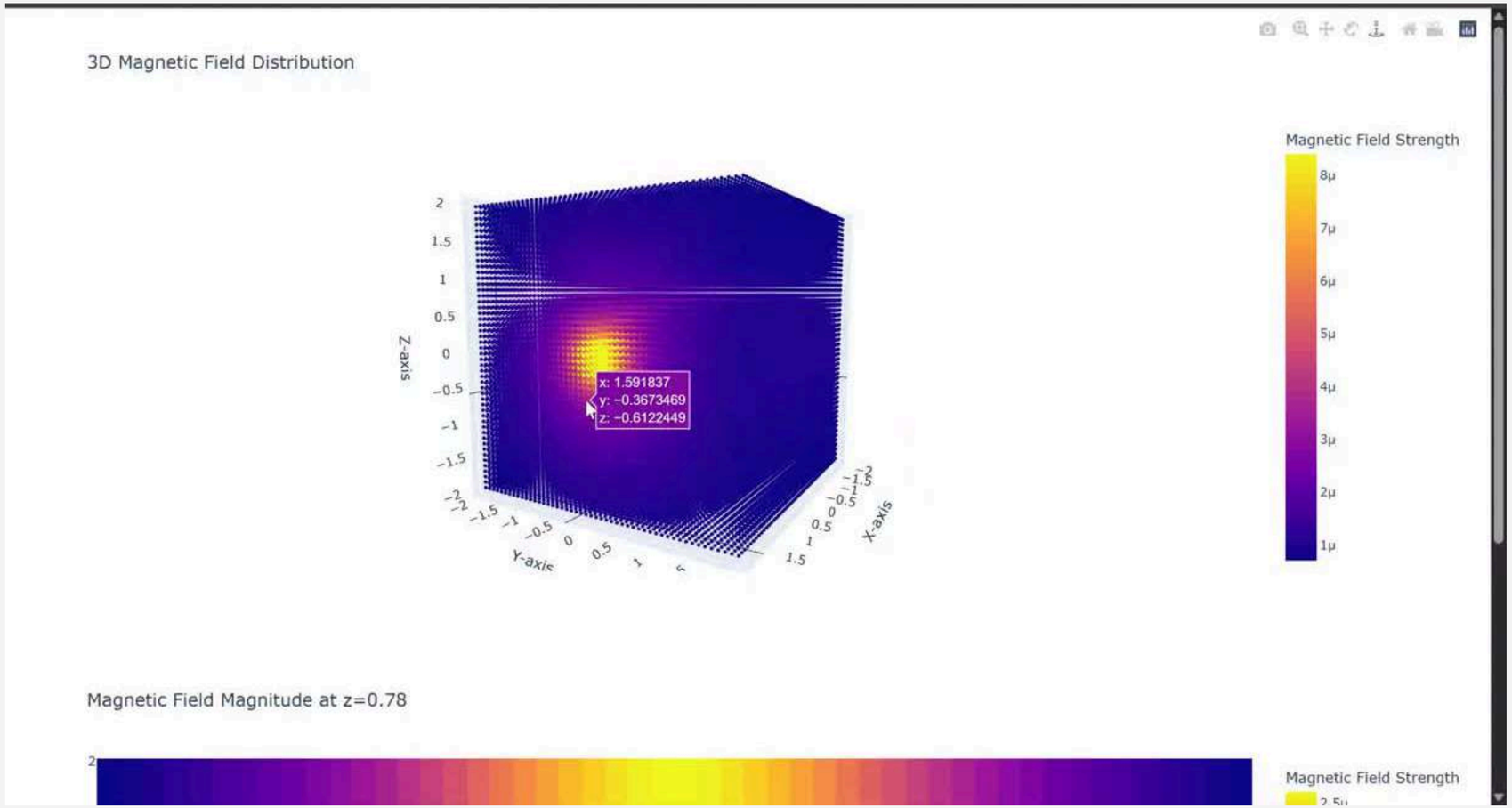
Working Demo Motor



Working Demo of GUI (Window)



Working Demo of GUI (Web)



04
PCB

Updates

- Three PCBs submitted for fabrication from PCB Power
  - TMAG3001 Sensor Array PCB
  - CMOD Board PCB
  - TMAG5170 Sensor Array PCB

- TMAG3001 and CMOD PCBs Gerber files approved by PCB power and fabricated, to be assembled next week.

- TMAG5170 Gerbers to be approved by PCB power.

- Stage 2 cleared and external vendor recommended for all PCBs

TMAG3001 PCB

TMAG3001 PCB

Audit-trail of PCB fabrication

Project title: P-05- MAGNETIC FIELD CAMERA

Group ID: MON-08

Student point of contact (name and roll no.): N.V.NAVANEETH RAJESH

Undertaking:

1. I certify that my team has thoroughly reviewed the correctness of schematic and layout of this PCB design.
2. We all understand that all PCBs that we need immediately have been submitted in this order.
3. We all understand that we will not be allowed to submit any more orders up to one week from "Approval for mask printing and start of PCB fabrication" (step 4 in table below)

Signature of all group members: Navaneeth Raunak K. Aditya Reddy

Justification for cases where all group member signatures are not present above:

Sr. no.	Task	Reviewed/executed by (name & sign)	Date & Time	Remarks
1	Layout footprint and floorplan review (WEL) Expected time to completion: 1 day	Aravindakshan/Rajesh/ Navneet/Aditya 19/05/2025		**Note:** Students should take photo of components placed on printout of board layout(s) on A4 sheet and show these during review in step 2 below
2	Layout approval and process selection (WEL) Expected time to completion: 1 day	Maheshwar/Ankur 19/05/2025		**Recommended PCB format:** A. Single-sided, no solder mask B. Double-sided, no solder mask C. With solder mask ✓ D. External vendor
3	CAM file review process compatibility (PCB Lab) Expected time to completion: 1 day	Vishnu/Shanmugam/ Sachin/Prayag		
4	Approval for mask printing and start of PCB fabrication (PCB Lab) Expected time to completion: 5 days	Yadnik		**Order ID:**
5	Receipt of PCB (from PCB lab)	**Received by:** Name: Roll no.:		

CMOD PCB

CMOD PCB

Audit-trail of PCB fabrication

Project title: P-05 - MAGNETIC FIELD CAMERA

Group ID: MON-08

Student point of contact (name and roll no.): N.V. NAVANEETH RAJESH 22B1215

Undertaking:

1. I certify that my team has thoroughly reviewed the correctness of schematic and layout of this PCB design.
2. We all understand that all PCBs that we need immediately have been submitted in this order.
3. We all understand that we will not be allowed to submit any more orders up to one week from "Approval for mask printing and start of PCB fabrication" (step 4 in table below)

Signature of all group members: Ramanuj Sumit Rawnak N. Aditya Reddy

Justification for cases where all group member signatures are not present above:

Sr. no.	Task	Reviewed/executed by (name & sign)	Date & Time	Remarks
1	Layout footprint and floorplan review (WEL) Expected time to completion: 1 day	Aravindakshan/Rajesh/ Navneet/Aditya		Note: Students should take photo of components placed on printout of board layout(s) on A4 sheet and show these during review in step 2 below
2	Layout approval and process selection (WEL) Expected time to completion: 1 day	Maheshwar/Ankur 29/03/2025		**Recommended PCB format:** A. Single-sided, no solder mask B. Double-sided, no solder mask C. With solder mask ✓ D. External vendor
3	CAM file review process compatibility (PCB Lab) Expected time to completion: 1 day	Vishnu/Shanmugam/ Sachin/Prayag		
4	Approval for mask printing and start of PCB fabrication (PCB Lab) Expected time to completion: 5 days	Yadnik		**Order ID:**
5	Receipt of PCB (from PCB lab)	**Received by:** Name: Roll no.:		

TMAG5170 PCB

TMAG5170 PCB

Audit-trail of PCB fabrication

Project title: P-05- MAGNETIC FIELD CAMERA

Group ID: MON-08

Student point of contact (name and roll no.): N.V. NAVANEETH RAJESH 22B1215

Undertaking:

1. I certify that my team has thoroughly reviewed the correctness of schematic and layout of this PCB design.
2. We all understand that all PCBs that we need immediately have been submitted in this order.
3. We all understand that we will not be allowed to submit any more orders up to one week from "Approval for mask printing and start of PCB fabrication" (step 4 in table below)

Signature of all group members: Navaneeth Rajesh Ramnek K. Aditya Reddy

Justification for cases where all group member signatures are not present above:

Sr. no.	Task	Reviewed/executed by (name & sign)	Date & Time	Remarks
1	Layout footprint and floorplan review (WEL) Expected time to completion: 1 day	Aravindakshan/Rajesh/ Navneet/Aditya 19/03/2025		Note: Students should take photo of components placed on printout of board layout(s) on A4 sheet and show these during review in step 2 below
2	Layout approval and process selection (WEL) Expected time to completion: 1 day	Maheshwar/Ankur 17/03/2025		**Recommended PCB format:** A. Single-sided, no solder mask B. Double-sided, no solder mask C. With solder mask D. External vendor
3	CAM file review process compatibility (PCB Lab) Expected time to completion: 1 day	Vishnu/Shanmugam/ Sachin/Prayag NA		
4	Approval for mask printing and start of PCB fabrication (PCB Lab) Expected time to completion: 5 days	Yadnik NA		**Order ID:**
5	Receipt of PCB (from PCB lab)	**Received by:** Name: Roll no.:		

Key Learnings

- The TMAG3001 magnetometer is very small (0.8×1.3 mm), which led to a few issues and learnings.
- Necessitated using similarly sized resistors and capacitors to minimize space usage. (0402 footprint)
- Due to the small size, it cannot be hand-soldered.
- The usual 0.2 mm track width is insufficient for connections due to the close pin spacing on the IC, requiring a smaller track width and clearance. (0.18 mm chosen)

Key Learnings

- A 2x2 layout for the PCBs was initially chosen but later we realized that the routing became complicated and switched to a linear design.
- The design evolved from a 2-layer to a 4-layer PCB to manage signals more efficiently and allow for closer spacing between the sensors.
- The SDA and SCL lines were kept in the same plane to minimize capacitance issues.
- Two power sources were used to ensure that the current limit of the ribbon cable (0.5 A) was not exceeded.

Choices made and Justification

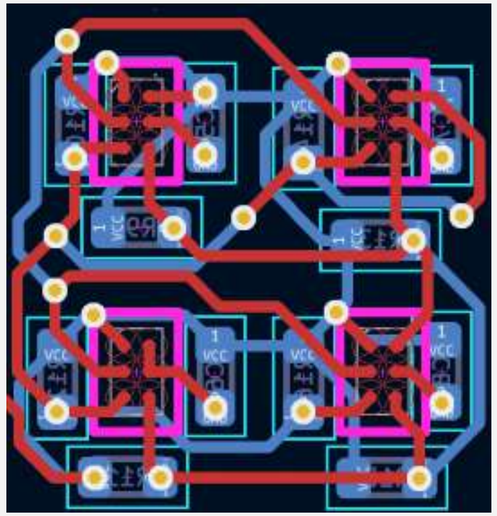
- The TMAG3001 was preferred due to its smaller size, allowing for higher resolution and I2C interfacing, which is more suitable for the array design.

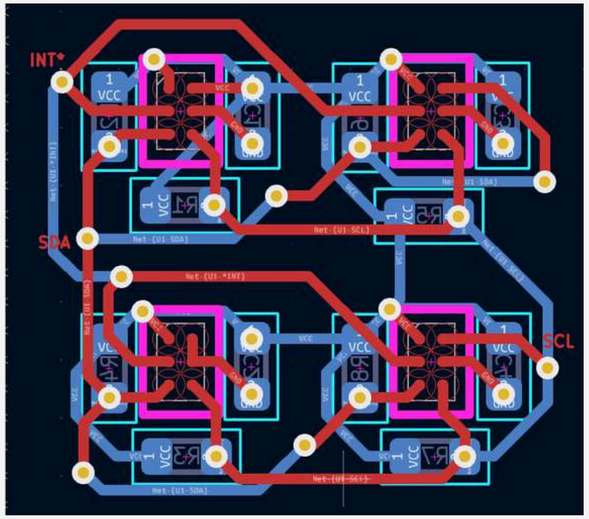
- Adopting a 4-layer design helped in managing the complex routing required, reducing clutter and improving signal integrity.

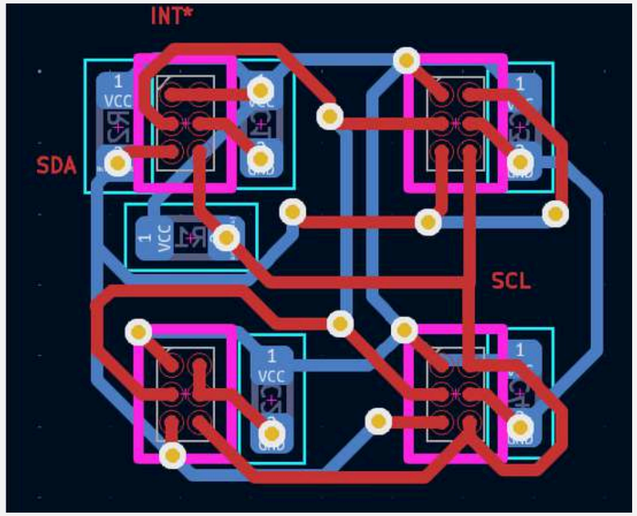
- TMAG5170 was chosen as our backup sensor as it uses SPI in which we have more experience. It is larger in size, allowing robust fabrication.

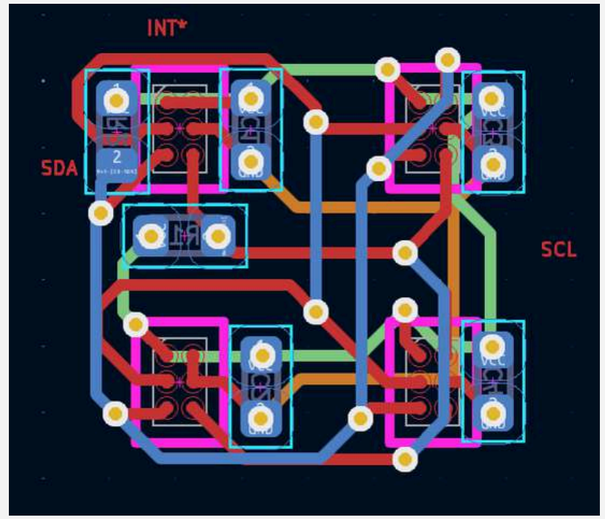
- We chose an external vendor as the IITB lab constraints were too large for either of our sensor PCBs.

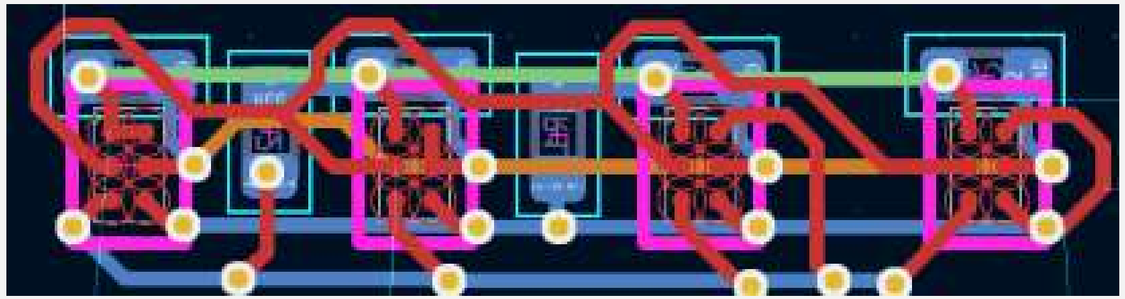
Design Evolution

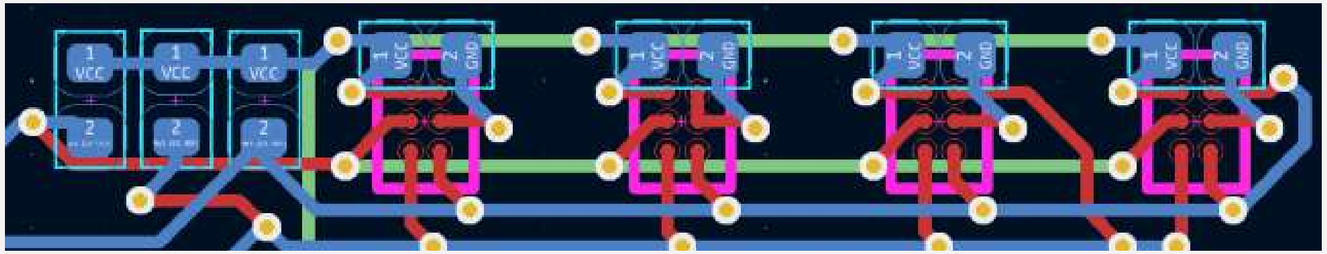




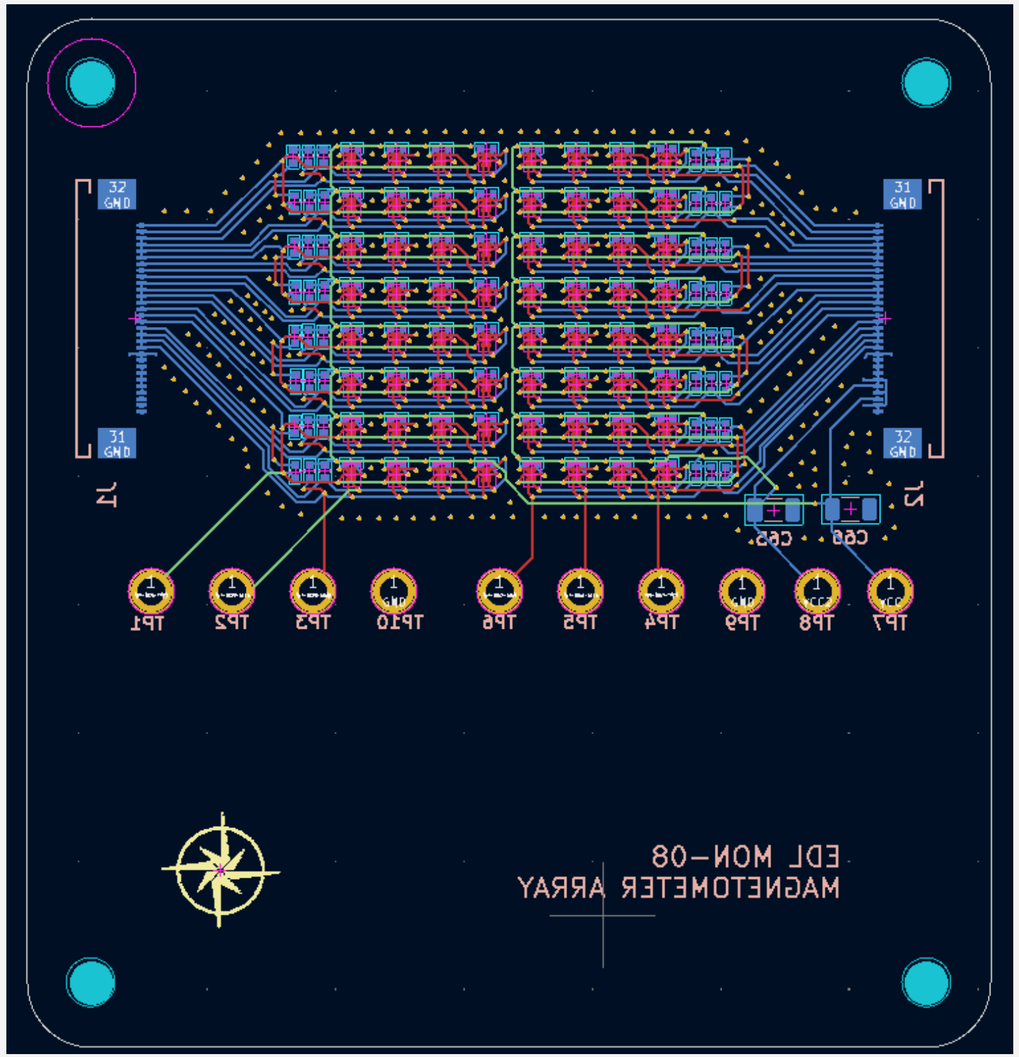




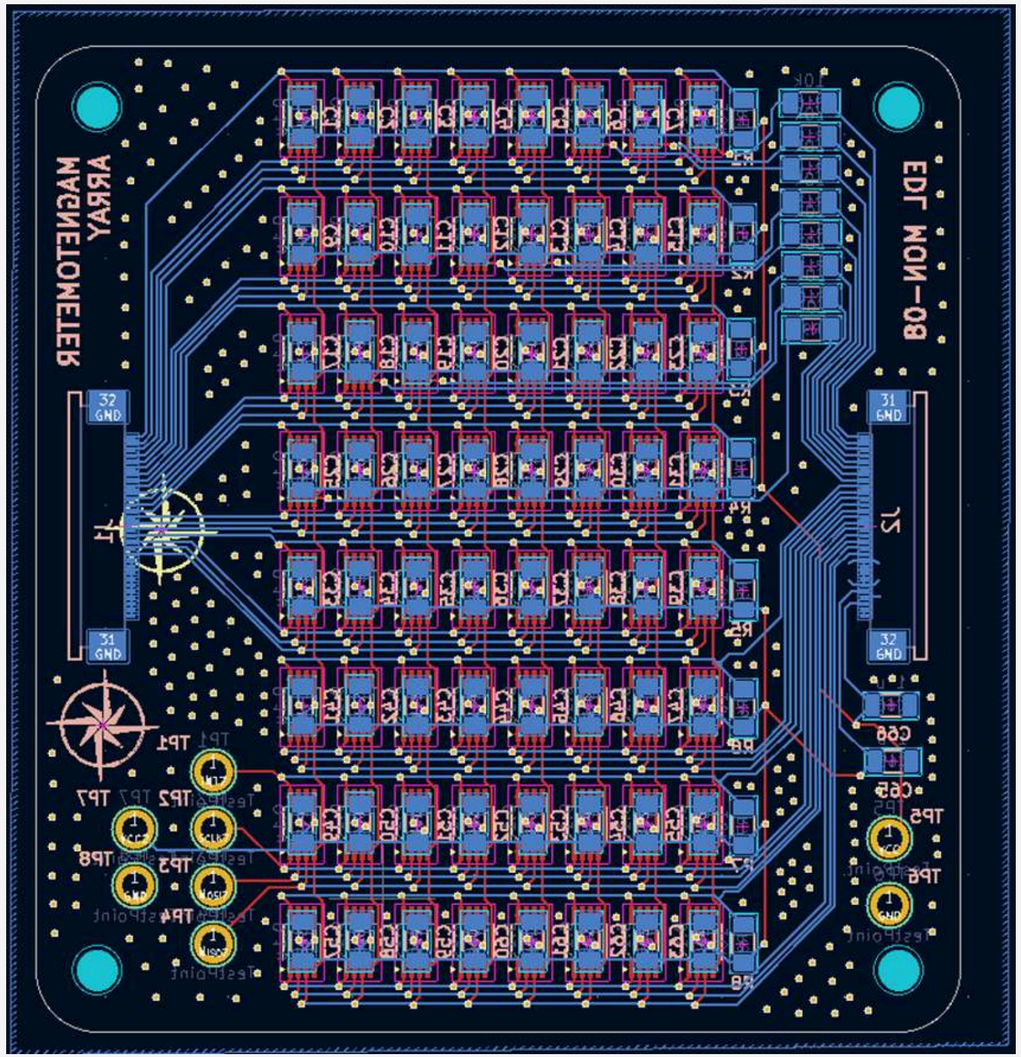




Final PCBs

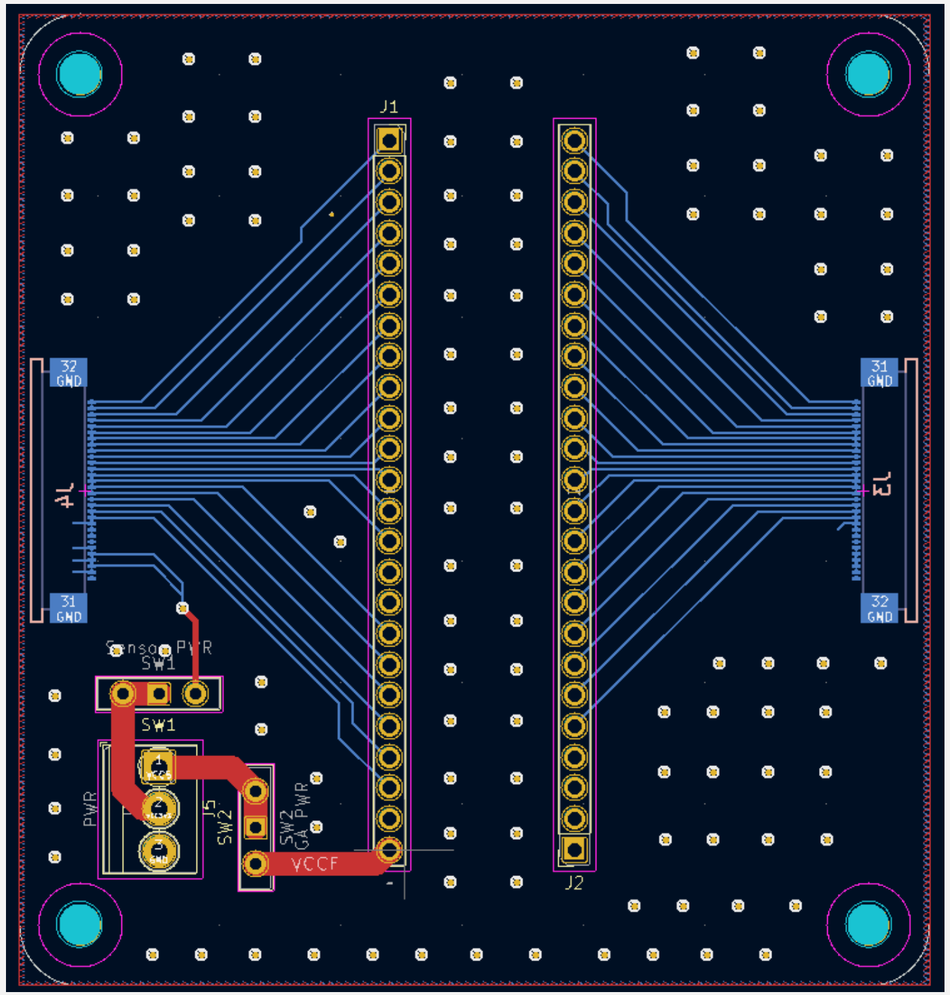


TMAG3001 PCB



TMAG5170 PCB

Final PCBs



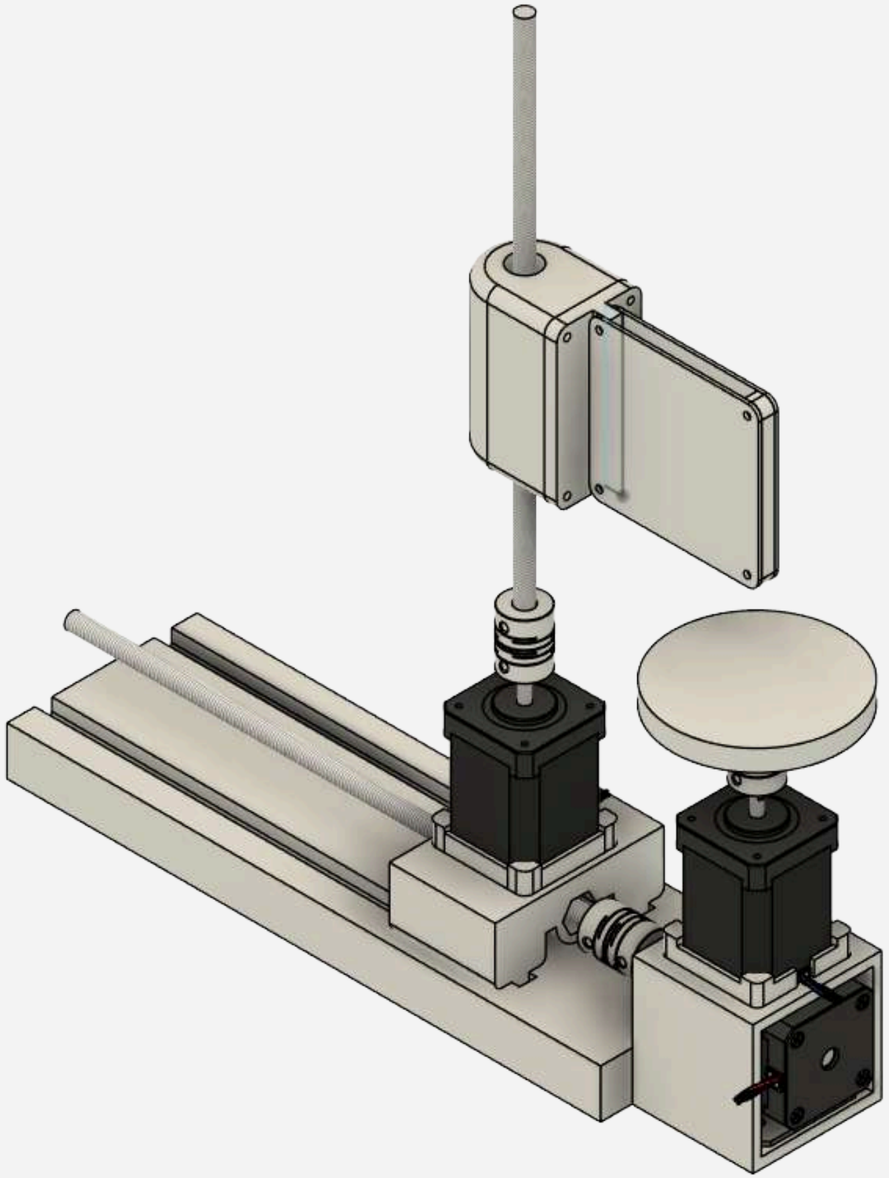
CMOD PCB

05
CAD Design and Power management

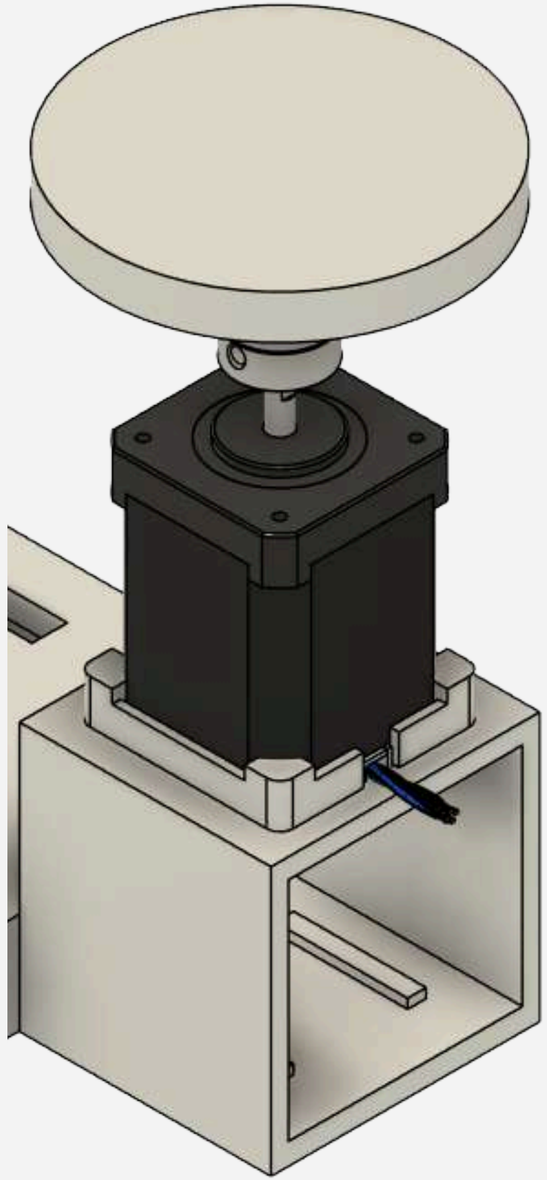
CAD Design

- Our CAD design got delayed beyond the last milestone since we received the motors late
- On testing the motors, we found that while they could support a high torque, supporting their weight on a moving platform could be difficult
- Our new design has all motors' weight supported by the base
- The design to enable movement of the arms in the r and z direction uses a threaded rod instead of rubber belts (like in traditional 3D printers) to ensure higher accuracy, since ensuring appropriate slack on a prototype is difficult
- We were unable to get it reviewed this week, but will do so in the lab on Monday

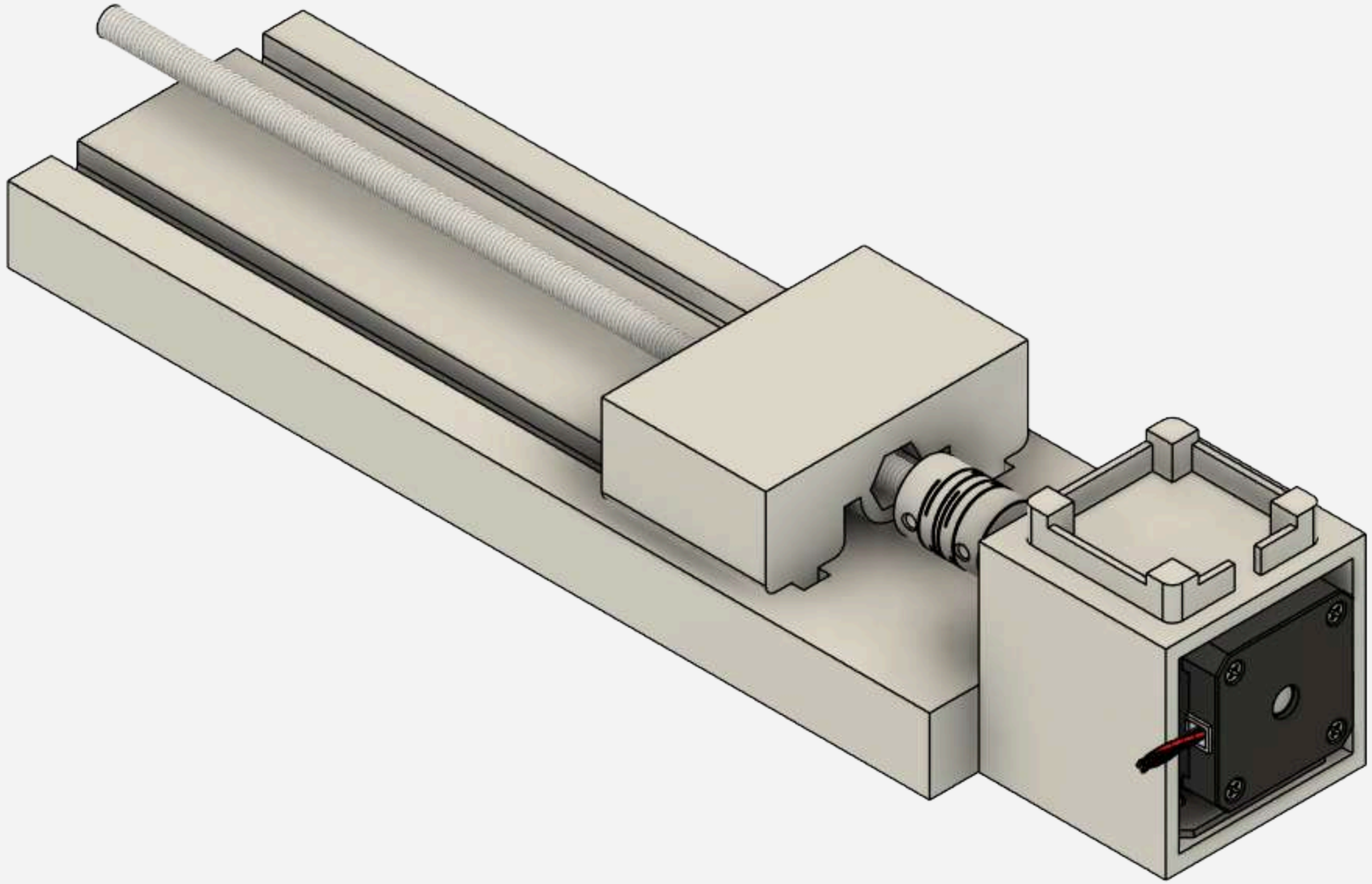
Overall Design



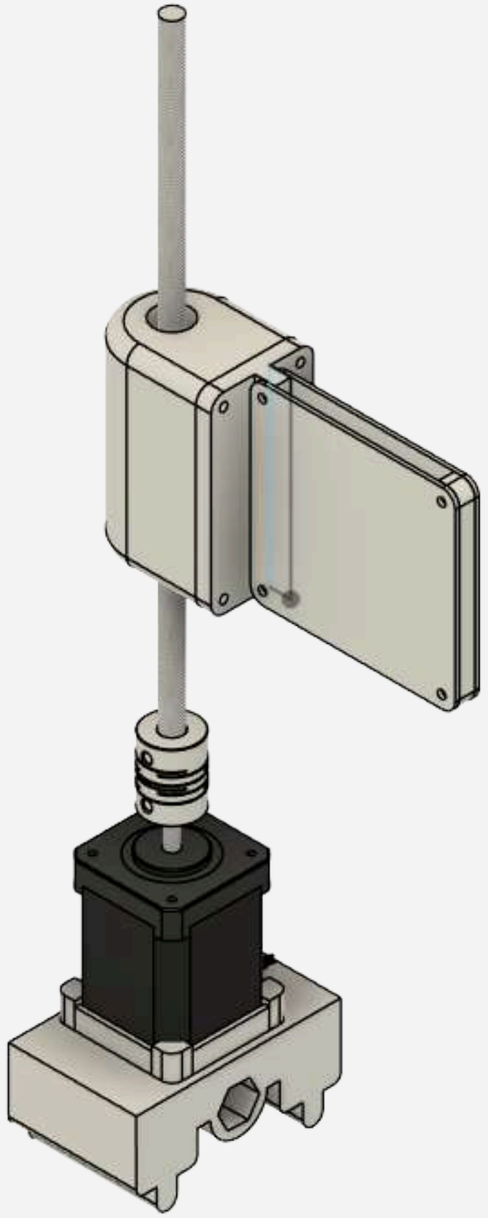
Theta movement



R movement

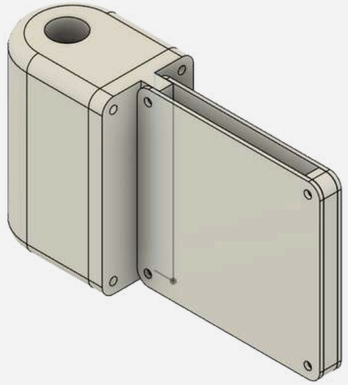


Z movement

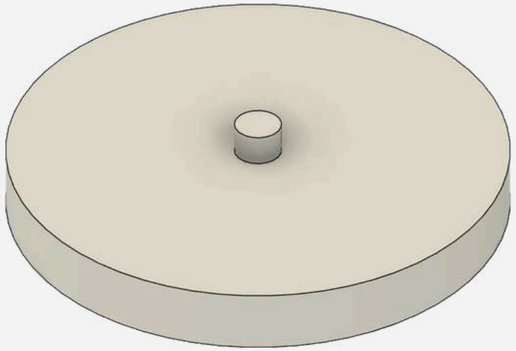


Components to be 3D printed

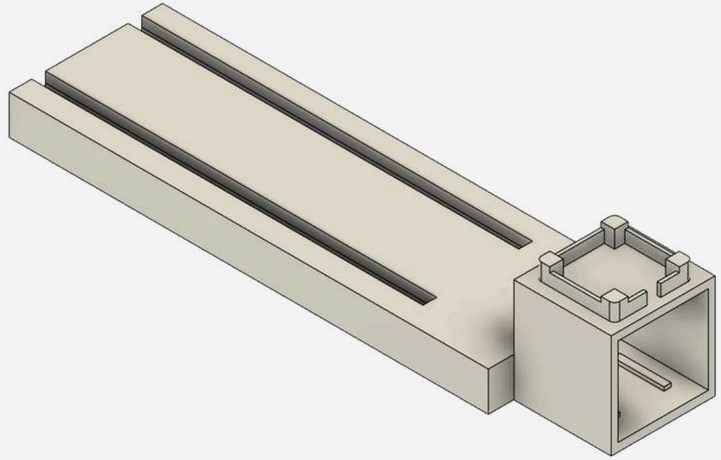
Sensor Head



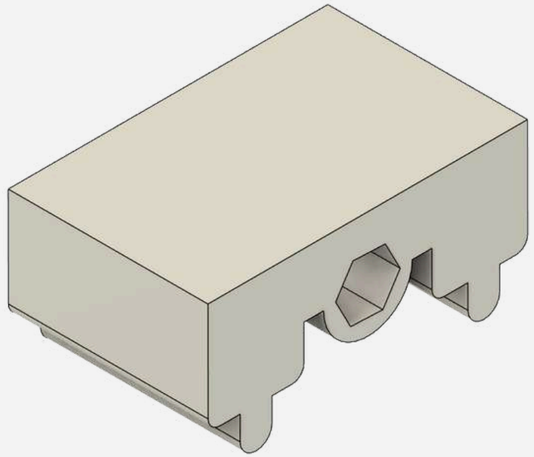
Platform



Base



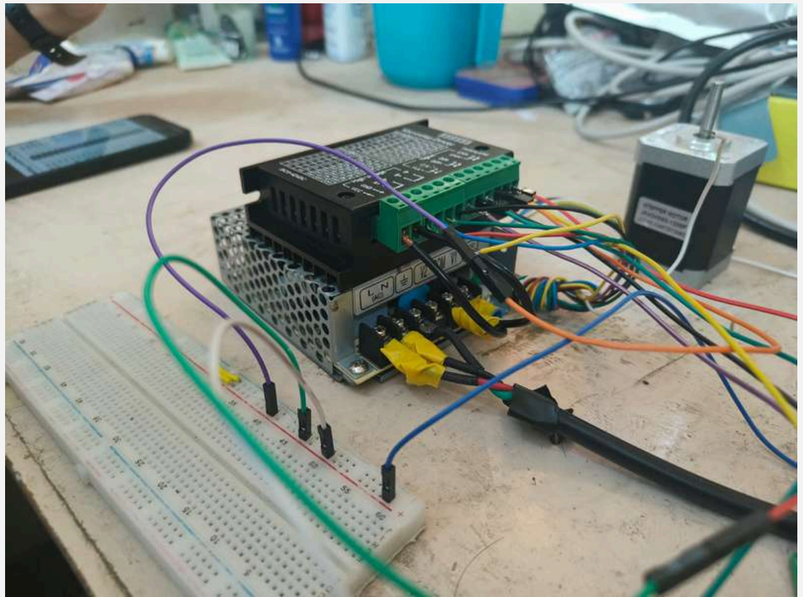
Track



Motor adapter

Power Management

- We have a single 12 V SMPS that we use to provide power to the microcontroller, the FPGA, the sensors and the motor drivers
- These components require 12V, 5V and 3.3V supplies which we provide using buck converters
- While there are individual buck converters that can provide both 5V and 3.3V, they did not meet our current requirement for the sensor, so we use two separate converters



06
Critical Taks and Key Risks

Critical Tasks

- Testing the breakout board with the I2C master
- 3D printing the required components, and testing the motor movement with the entire assembled rig to ensure we have millimetre accuracy
- Testing 64 sensor readout to the CMOD FPGA on receiving the PCB
- Testing the power management system with the whole system to ensure it meets current requirements

Key risks

- The passive heat dissipation of the motors is insufficient
  - **Solution**: We have found heat sinks, fans and an adapter to connect them to the motor if required. The CAD is designed such that it can easily be adapted to this if required
- The sensor PCB does not function as expected
  - **Solution**: We have a 64 sensor array with 16 buses, so one bus being affected still ensures we can obtain a map of the field. If the PCB as a whole does not work, we have a second PCB with a different sensor that we can use instead
- The rig movement is not accurate enough
  - **Solution**: We have motor drivers that allow for microsteps for higher resolution. We also have proximity sensors that can be used to calibrate position before a run

07
Future Plans

Work Distribution until Final Demo

- **Aditya:** Fine-tuning GUI and adding ability to control the system from the laptop
- **Raunak:** Completing integration of microcontroller code to allow control of all components simultaneously
- **Ashwajit:** Printing and assembling the rig and testing accuracy of movement, and making changes for heat management
- **Suchet:** Further breakout board testing, FPGA code for 64 sensors and readout to RPi
- **Navaneeth**: Solder work for PCBs, Testing readout of 64 sensor PCB, Interfacing sensor data with GUI.

Plan for Final Demo

- Once the power supply is turned on, all the components are powered, but they are controlled from the laptop GUI via the microcontroller
- We first send a command to do a sweep across the volume to note the background magnetic field
- We then place a motor or a permanent magnet on the platform to find the magnetic field generated by it
- This is sent back to the laptop via WiFi, and shown in the GUI as in a previous slide
- A stretch goal is to be able to show periodic magnetic fields like from AC current, which the sensors have the capacity to do