

Worcester Polytechnic Institute
Department of Mechanical and Materials
Engineering

C term March 5th, 2025

ME 5303 Applied Finite Element Methods
Final Project

**Stress, Strain, and Deformation Analysis
of the Chest of a Humanoid Robot**

Student

Jack Aylesworth
jmaylesworth@wpi.edu
Will Pagliarulo
wbpagliarulo@wpi.edu
David Strom
dstrom@wpi.edu

Course Professor

Prof. Nima Rahbar
nrahbar@wpi.edu

Teaching Assistant

Alireza Mahdavi Nejad
amahdavinejad@wpi.edu

Table Of Contents

List of Figures.....	3
List of Tables.....	4
Introduction.....	5
Goals.....	6
Methods.....	6
Results.....	7
Conclusions.....	13
References.....	13

List of Figures

Figure 1: Model of humanoid robot in isometric view (Left) and front view (Right)

Figure 2: Model of old robot chest chassis design in isometric view (Left) and side view (Right)

Figure 3: Model of new robot chest chassis design in isometric view (Left) and with Ansys generated mesh (Right)

Figure 4: Ansys stress simulated results of the old chassis

Figure 5: Zoomed in Ansys stress simulated results of the old chassis

Figure 6: Ansys strain simulated results of the old chassis

Figure 7: Zoomed in Ansys strain simulated results of the old chassis

Figure 8: Ansys deformation simulated results of the old chassis

Figure 9: Zoomed in Ansys deformation simulated results of the old chassis

Figure 10: Ansys stress simulated results of the new chassis

Figure 11: Zoomed in Ansys stress simulated results of the new chassis

Figure 12: Ansys strain simulated results of the new chassis

Figure 13: Zoomed in Ansys strain simulated results of the new chassis

Figure 14: Ansys deformation simulated results of the new chassis

Figure 15: Zoomed in Ansys deformation simulated results of the new chassis

List of Tables

Table 1: Max stress, strain, and deformation for the old and new chassis design

Introduction

The part that was analyzed for this project came from the ongoing humanoid robot MQP which the majority of this group has been working on since A-Term. The goal of the MQP is to improve the design of two robots, named Ava and Finley, with the intention to add to their functionality as at-home assistive care. In the past few months, many different parts have been redesigned and finalized as improved versions of the robot parts that came from past MQPs. One part in particular, what we call the “Chest/Back” is a very unique part of our project, as it’s the combination of the “Chest” and “Back” parts created in previous years. We chose to combine these two parts to eliminate unneeded degrees of freedom, add structural support, and reduce manufacturing time. This analysis will be useful in providing us with a visual of the structural pressure points of this improved design.

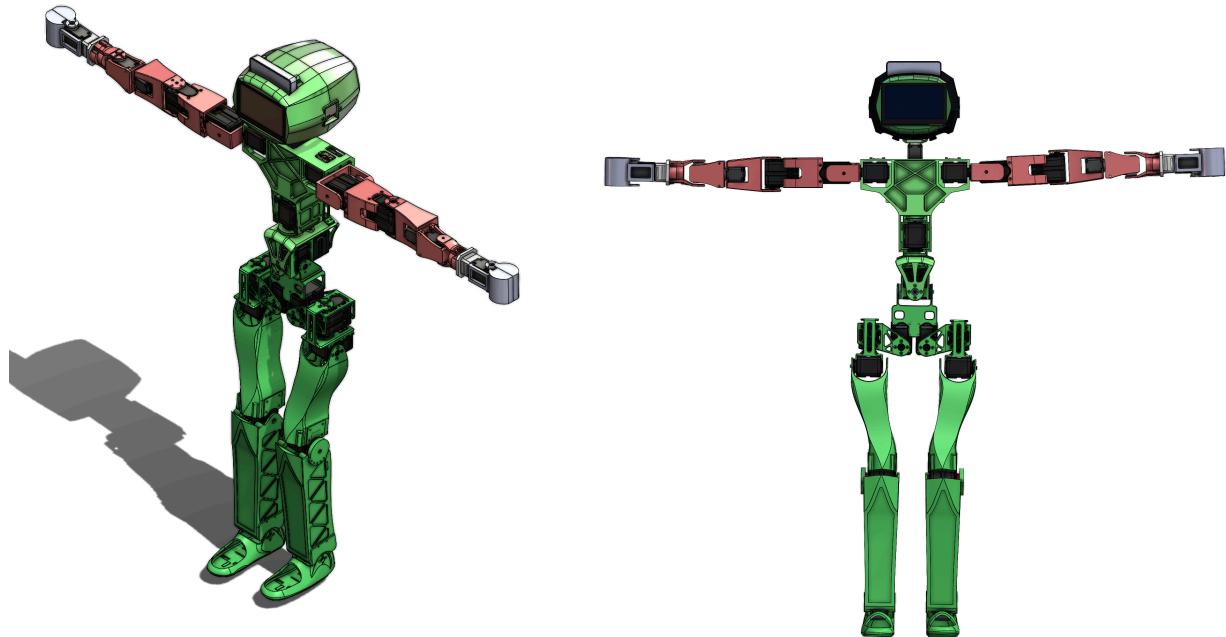


Figure 1: Model of humanoid robot in isometric view (Left) and front view (Right)

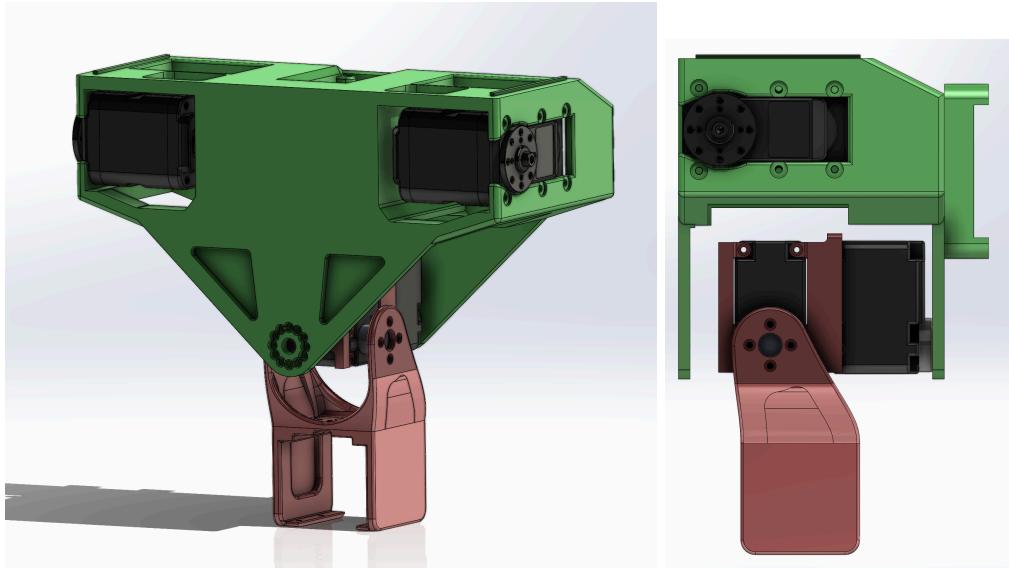


Figure 2: Model of old robot chest chassis design in isometric view (Left) and side view (Right)

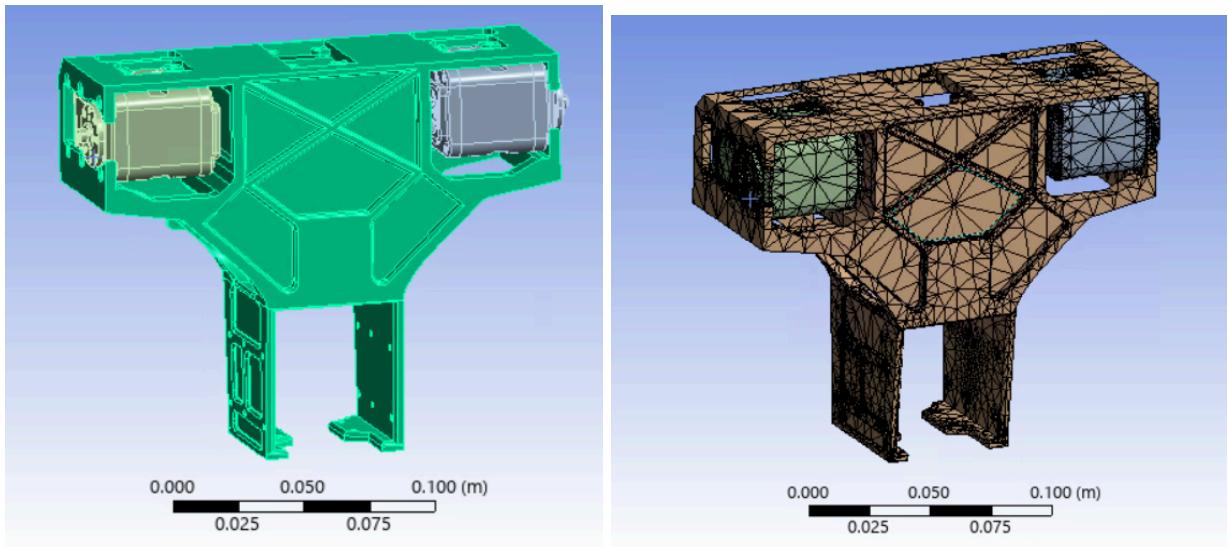


Figure 3: Model of new robot chest chassis design in isometric view (Left) and with Ansys generated mesh (Right)

Goals

Performed with Ansys Finite Element Analysis:

1. How does the old chest design's structural performance compare with the new design?
2. Is there a significant risk of new chassis deformation or failure under the applied loads?

Methods

Ansys is an incredible tool used for finite element analysis, we've used it plenty of times for stress analysis and structural data for our MQP in the past. Due to this past experience, we

were well prepared to dive into this process of analyzing the stress, strain, and deformation of our newly designed chest piece. With our goals in mind, we used the program for each design accordingly and got results that were honestly unsurprising.

Results

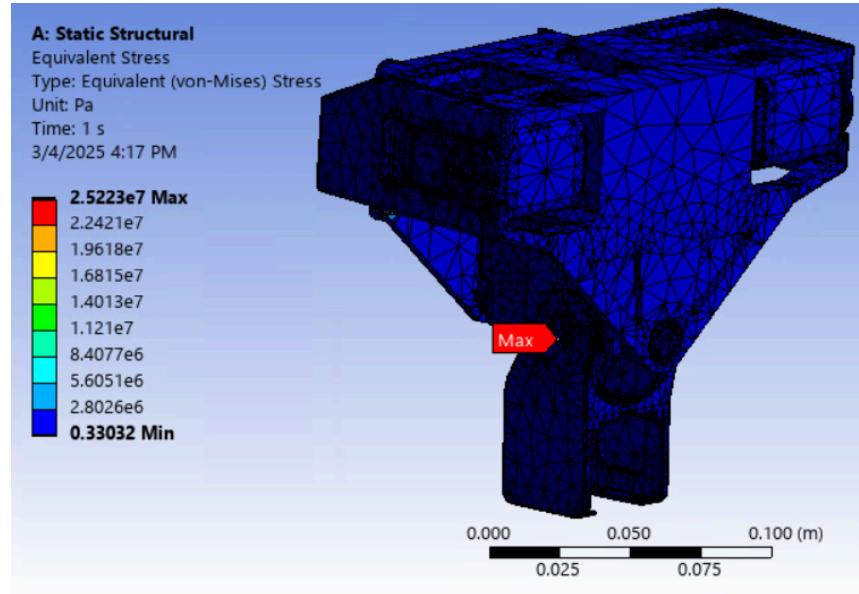


Figure 4: Ansys stress simulated results of the old chassis

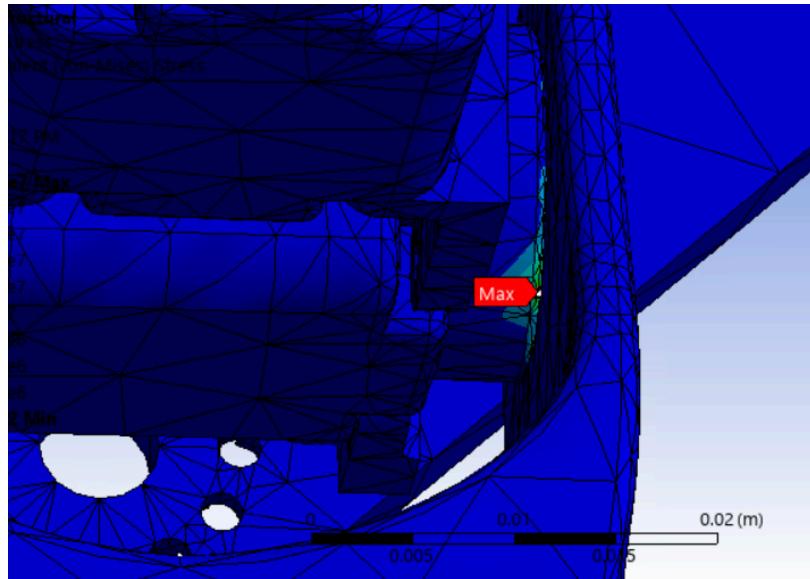


Figure 5: Zoomed in Ansys stress simulated results of the old chassis

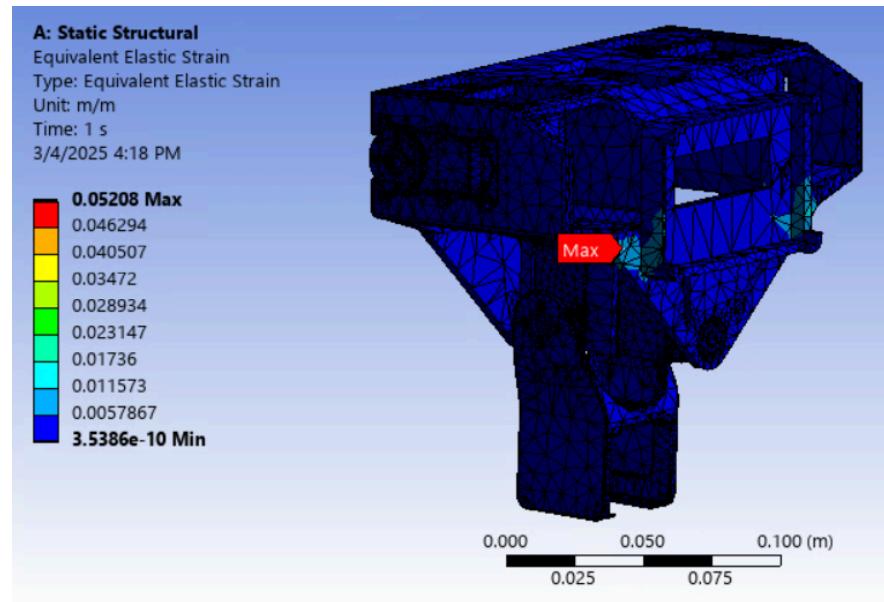


Figure 6: Ansys strain simulated results of the old chassis

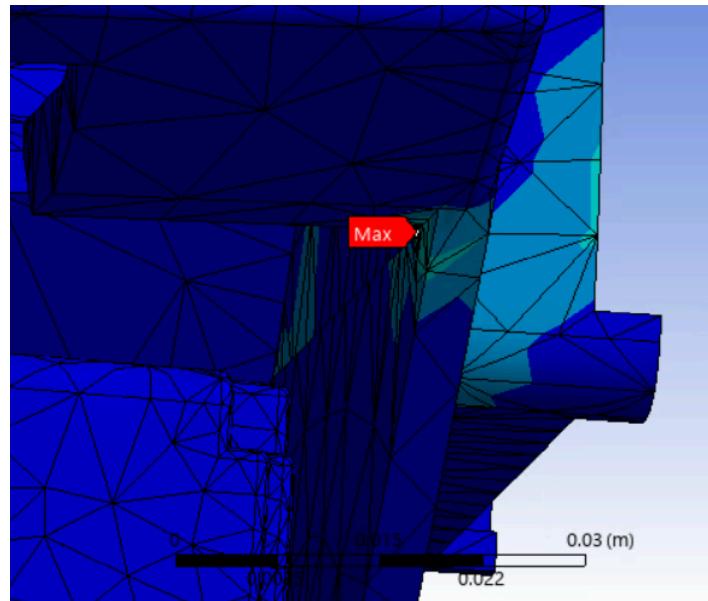


Figure 7: Zoomed in Ansys strain simulated results of the old chassis

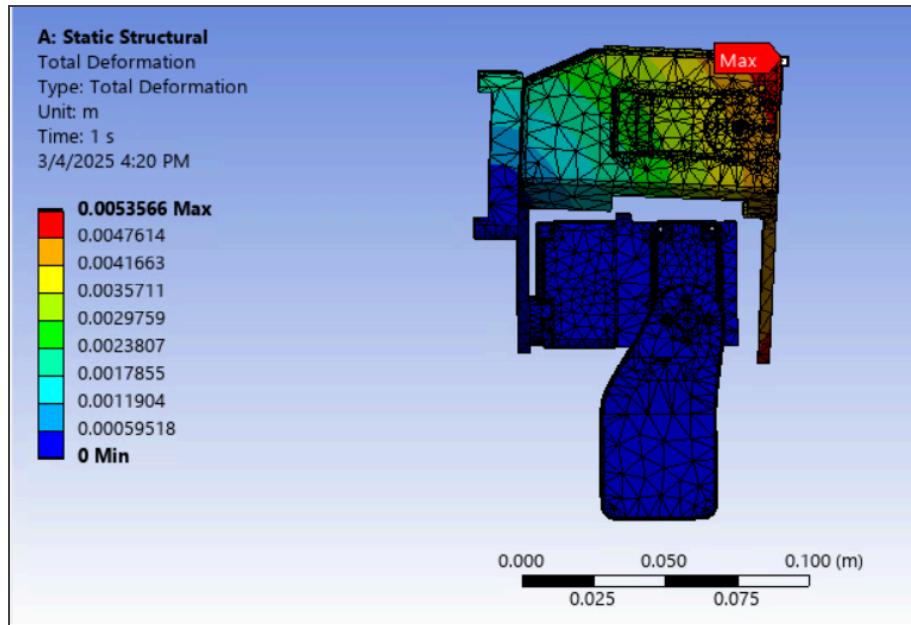


Figure 8: Ansys deformation simulated results of the old chassis

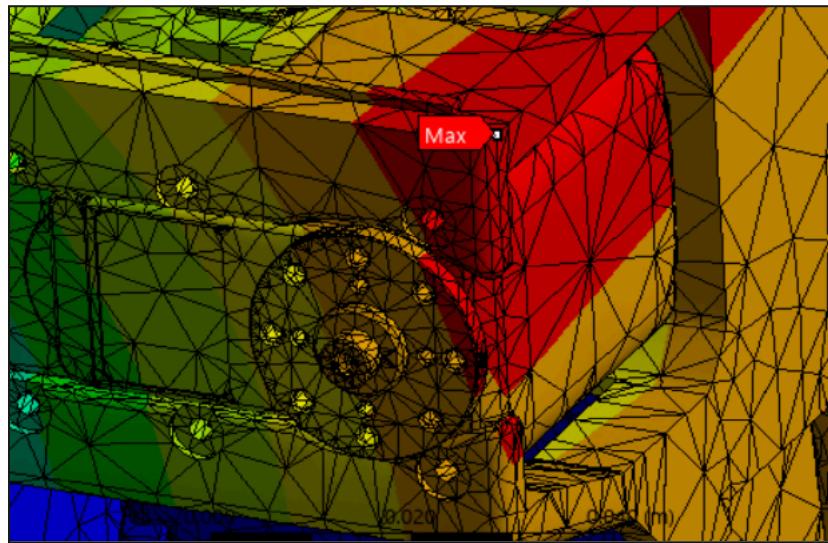


Figure 9: Zoomed in Ansys deformation simulated results of the old chassis

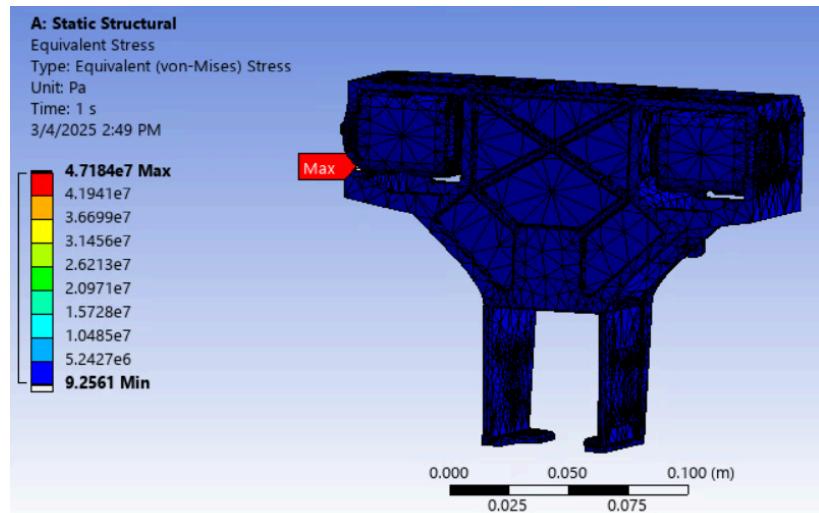


Figure 10: Ansys stress simulated results of the new chassis

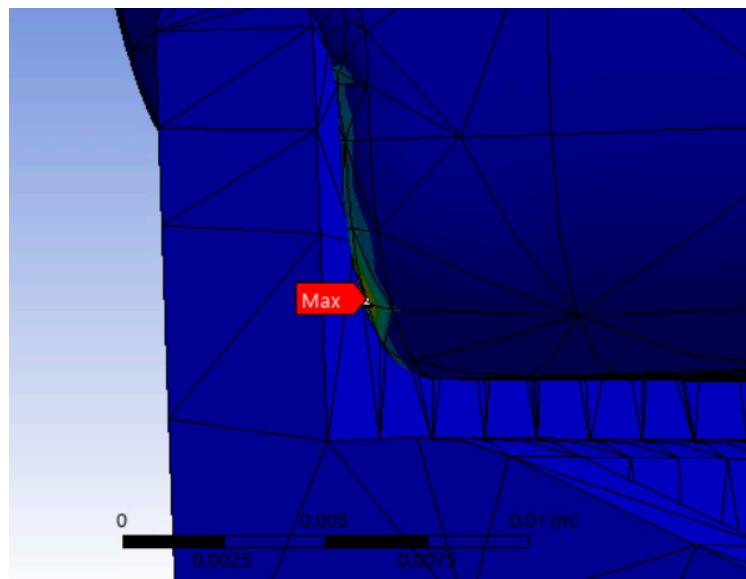


Figure 11: Zoomed in Ansys stress simulated results of the new chassis

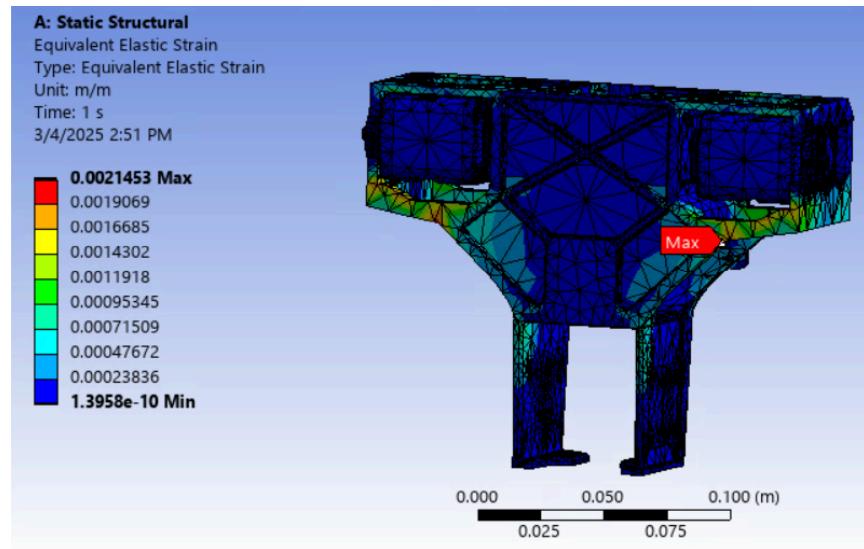


Figure 12: Ansys strain simulated results of the new chassis

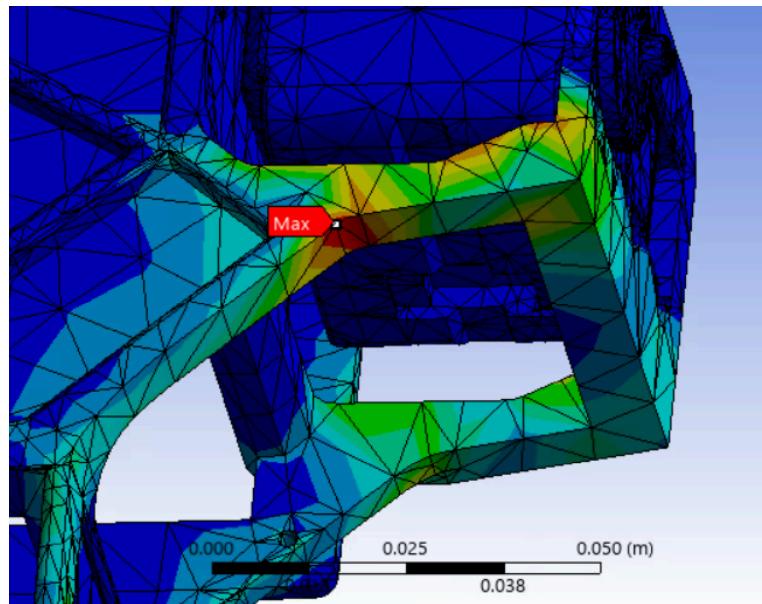


Figure 13: Zoomed in Ansys strain simulated results of the new chassis

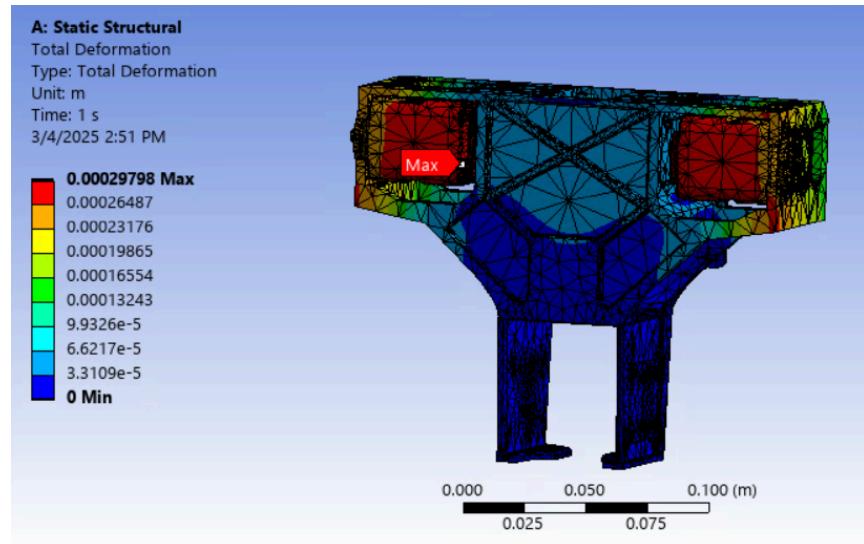


Figure 14: Ansys deformation simulated results of the new chassis

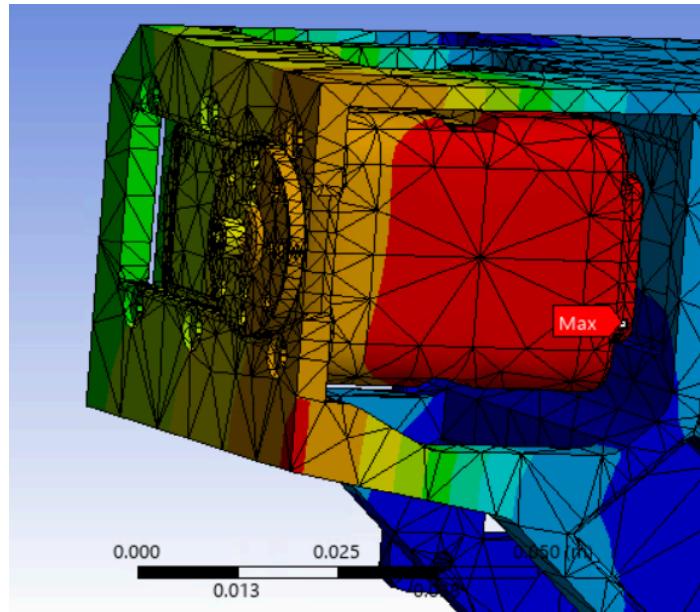


Figure 15: Zoomed in Ansys deformation simulated results of the new chassis

	Max Stress (MPa)	Max Strain (m/m)	Max Deformation (m)
Old Design	25.223	.05208	.0053566
New Design	47.18	.0021453	.00029798

Table 1: Max stress, strain, and deformation for the old and new chassis design

Conclusions

Given the results shown in Figures 4 through 15 and in Table 1, there has been a noticeable improvement in the structural integrity of the chest design. So much so that we will be finalizing this design as the chest of the robot for this year. The pressure points shown are nowhere near as problematic as they were in the original design, and in our experience, they will be negligible when the chest is fully installed.

References

- Sutherland, A., Michels, W., McCusker, A., & Gupta, S. (2024). *Design & Assembly of a 3D Printed Humanoid for At-Home Assistive Care*. : Worcester Polytechnic Institute.