Biomechatronic Hip Exoskeleton Team (BHET)

Individual Analysis:

Deflection and Stress on Hip Exoskeleton Frame

KEEGAN RAGAN

2019-2020

Project Sponsor: The NAU Biomechatronics Laboratory and W.L. Gore & Associates, Inc.

Sponsor Mentor: Leah Liebelt

Instructor: Sarah

Introduction

The purpose of this report is to conduct a finite element analysis (FEA) on the frame components of the biomechatronic hip exoskeleton designed for the BHET capstone project. Pictured below is the current CAD design for the hip exoskeleton.

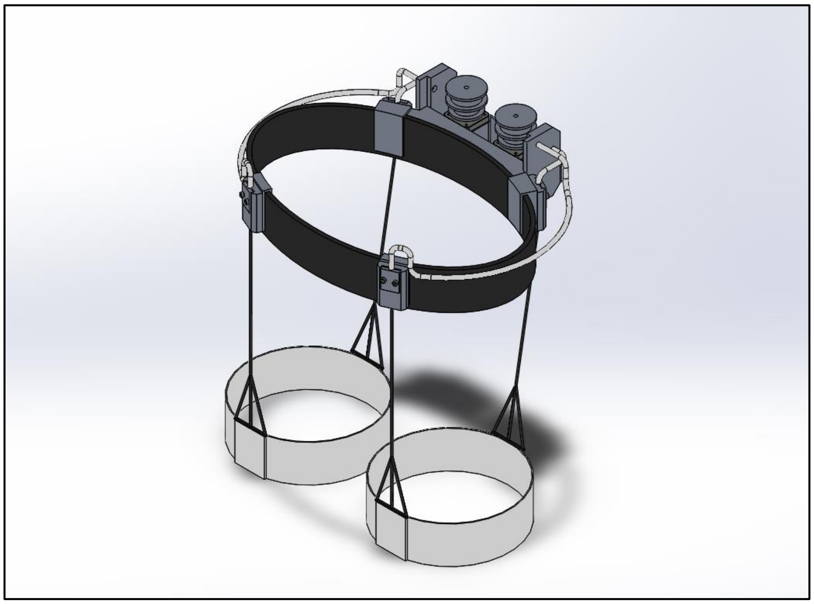

Figure 1: Front isometric view of hip exoskeleton

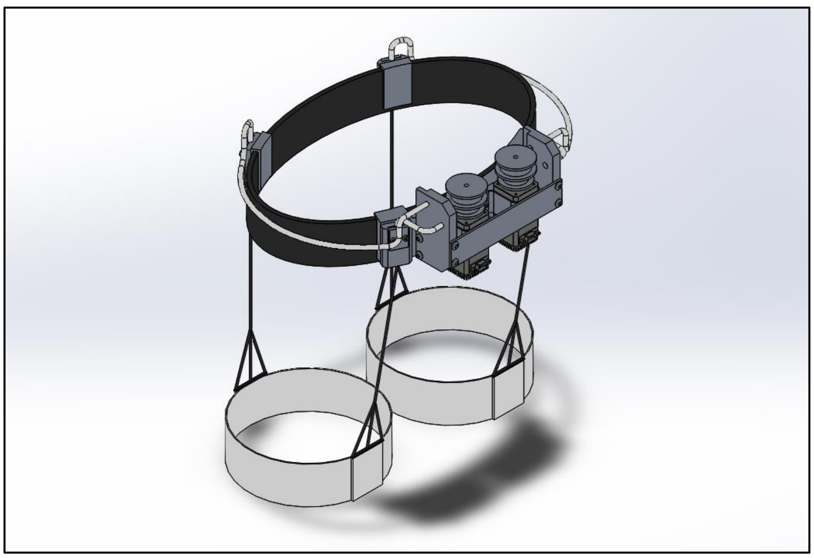

Figure 2: Rear isometric view of hip exoskeleton

The parts of the above design that are expected to experience the greatest amount of force are the four cable bosses and the two motor mounts. These components of the design will be the subject of the analysis. The below figure shows the components in question.

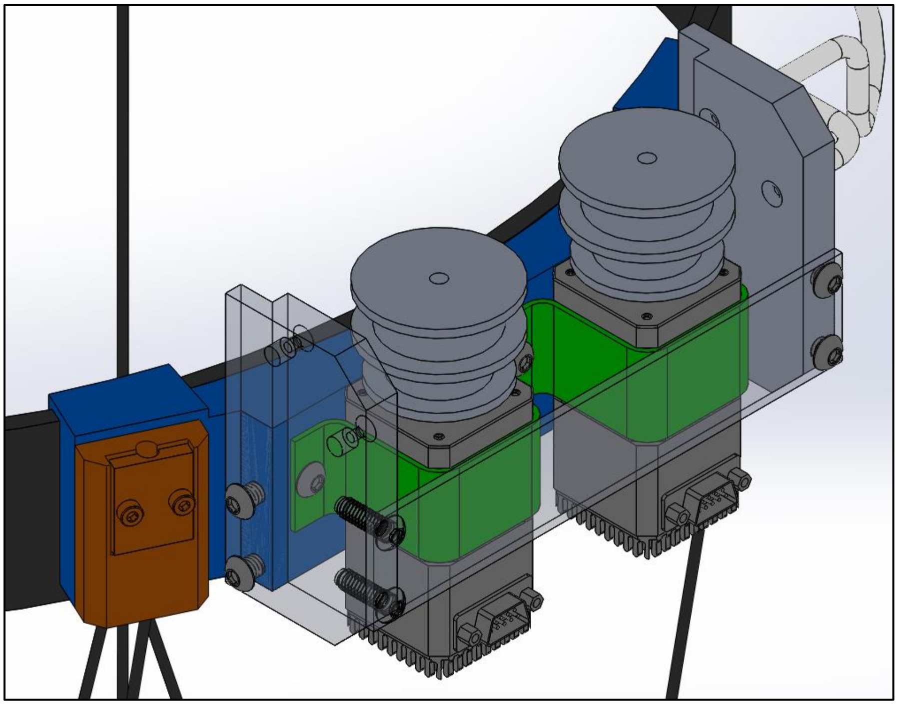

Figure 3: Detail view of motor mounts (highlighted green), motor brace (highlighted blue) and cable boss (highlighted orange)

The main concerns about the portion of the design detailed in Figure 3 is deflection or shearing of the motor mounts and displacement of the cable bosses. The FEA analysis described in the below sections will test for this phenomenon and inform any necessary design changes. More thorough details of the design described in the above figures can be found in the engineering drawings attached to this report.

Assumptions

The following assumptions will be made during the FEA analysis:

1. The motor mounts are fixed at the location of the two bolts that connect them to the backplate.
2. The rear cable bosses will be fixed at the backplate.
3. The front cable bosses are assumed to experience similar forces as the rear cable bosses. Therefore, a single analysis will represent all four cable bosses present on the exoskeleton.
4. All components in this analysis are made of 6061-T6 aluminum. Material properties are taken from SolidWorks. These values are shown in the below table.

Table 1: Material Properties of 6061-T6 Aluminum

Property	**Value**	**Units**
Elastic Modulus	6.90E+10	N/m^2
Poisson's Ratio	0.33	N/A
Shear Modulus	2.60E+10	N/m^2
Mass Density	2700	kg/m^3
Tensile Strength	3.1E+08	N/m^2
Compressive Strength		N/m^2
Yield Strength	2.75E+08	N/m^2
Thermal Expansion Coefficient	2.40E-05	/K
Thermal Conductivity	166.9	W/(m·K)
Specific Heat	896	J/(kg·K)

All the above assumptions assume an ideal case where all components are completely rigid when connected to the harness. This will likely not be the case with the real exoskeleton. Therefore, the FEA analysis conducted in this report will only serve to inform design decisions. It is likely that more design changes will be made upon testing the full-scale prototype. Additionally, because no full-scale tests have been run there are no real-world values to input into FEA models. Therefore, force values used in the FEA model will be estimated. These estimations will still allow for problematic stress concentrations in the design to be identified and corrected.

Background on Finite Element Analysis

Finite element analysis (FEA) is a common engineering analysis technique that is typically used in conjunction with computer assisted design (CAD). In FEA analysis a structure (or CAD model) is broken into finite, well defined, elastic substructures called elements [1]. In conjunction with the properties of the material the structure is made from, these elements can be used to calculate a variety of values for analysis. The below figure demonstrates the concept of breaking a structure into elements.

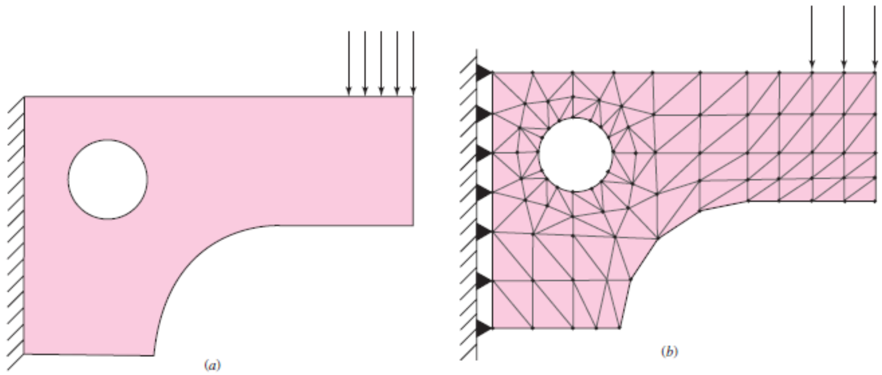

Figure 4: Idealized model (a) and finite element model (b) [1]

Figure 4 shows how the idealized model is broken up into elements. The combination of all the elements in an FEA model is referred to as the mesh. The density of this mesh corresponds to the error in the results of the analysis. A less dense mesh results in less elements and typically more error, particularly in a structure with more complex geometry. Often mesh density will vary throughout a structure, with higher density meshes localized to more geometrically complex portions of the structure. This typically cuts down on the compute time of the mesh. There are other components to the quality of mesh as well, such as the shape of the elements. The FEA analysis in this report used triangular elements with a constant mesh size, for simplicity.

FEA Models

The below sections describe the FEA analysis conducted on individual components of the hip exoskeleton.

Cable Boss

The cable boss acts as a mounting point for the Bowden cables utilized in this design. The cable housing will be clamped into the boss, which allows for it to terminate at this point so the cable can run down to the user's knees. The point where the housing interfaces with the cable boss will experience a force as cable is pulled in and let out of the housing. This was simulated in FEA by fixing the rear face of the cable boss and applying 10 kilograms of force to the cable housing interface. The below figure shows the results of this analysis.

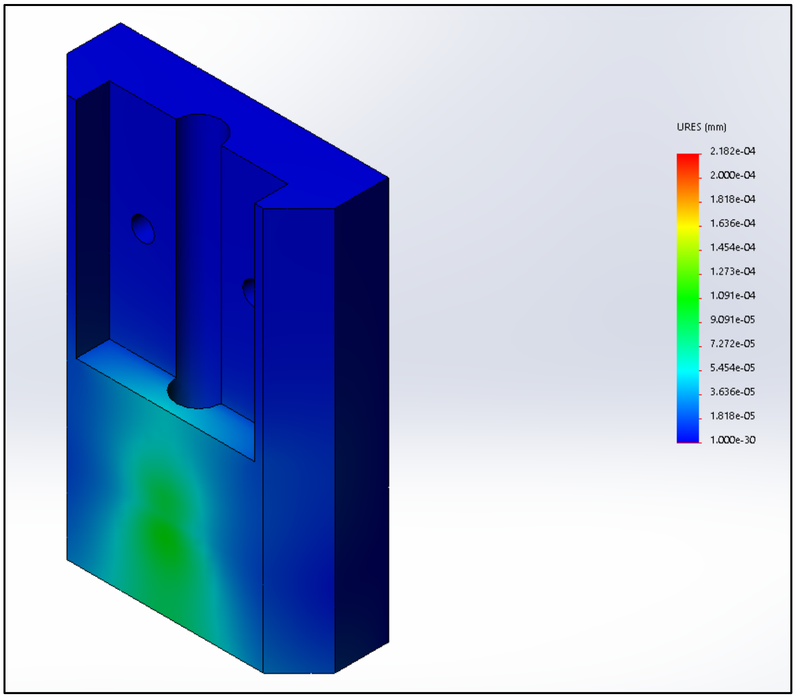

Figure 5: Displacement on cable boss with 10kg of force applied.

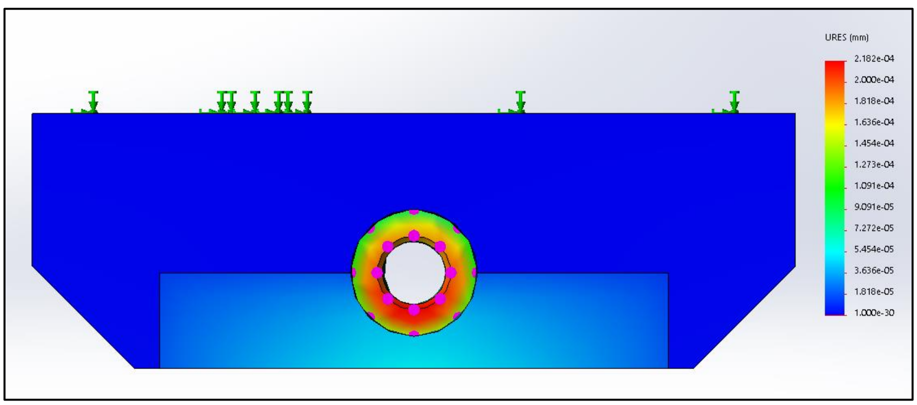

Figure 6: Displacement on cable boss with 10kg of force applied (Top).

As shown in the above figures, the most significant displacement occurs where the cable housing contacts the cable boss. However, the maximum projected displacement at these forces is $2.182 * 10^{-4}$ millimeters, which is an acceptable amount. At this point the design of the cable boss will not be altered prior to construction of the prototype.

Motor Brace Assembly

The motor brace assembly acts as a platform that stabilizes both the cable bosses and motors. Ensuring that this assembly experiences as little deformation as possible is critical. The same 10kg force applied in the previous analysis was applied again in this analysis. Additionally, a 9lbf-in torque was applied to the motor brackets. This value was taken from a motor quoted from McMaster-Carr [2]. Shown below are the Von Misses stress and displacement stress charts for the motor brace assembly.

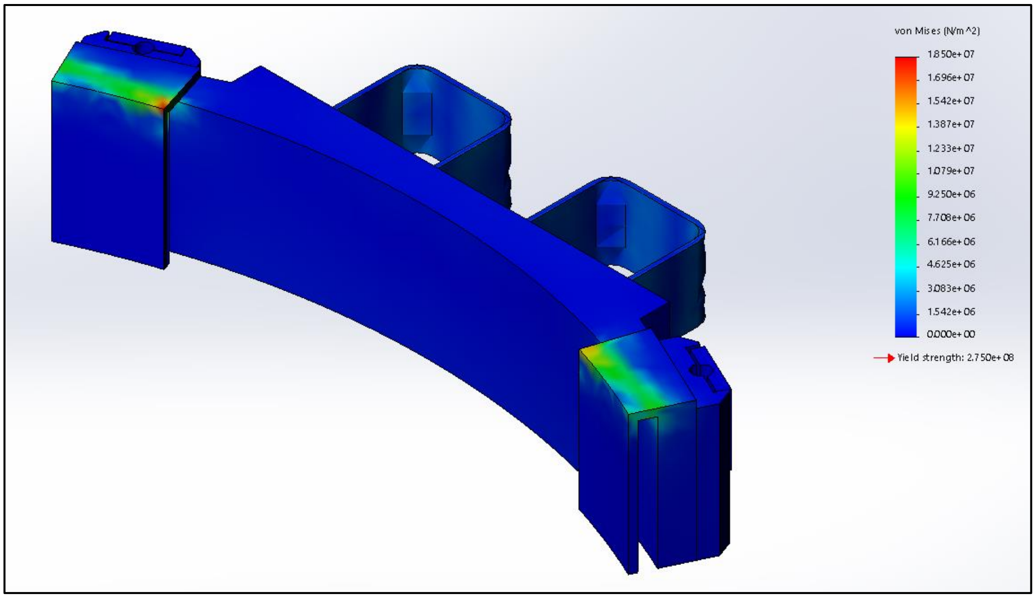

Figure 7: Von Mises stresses on the motor brace assembly in Nm^2

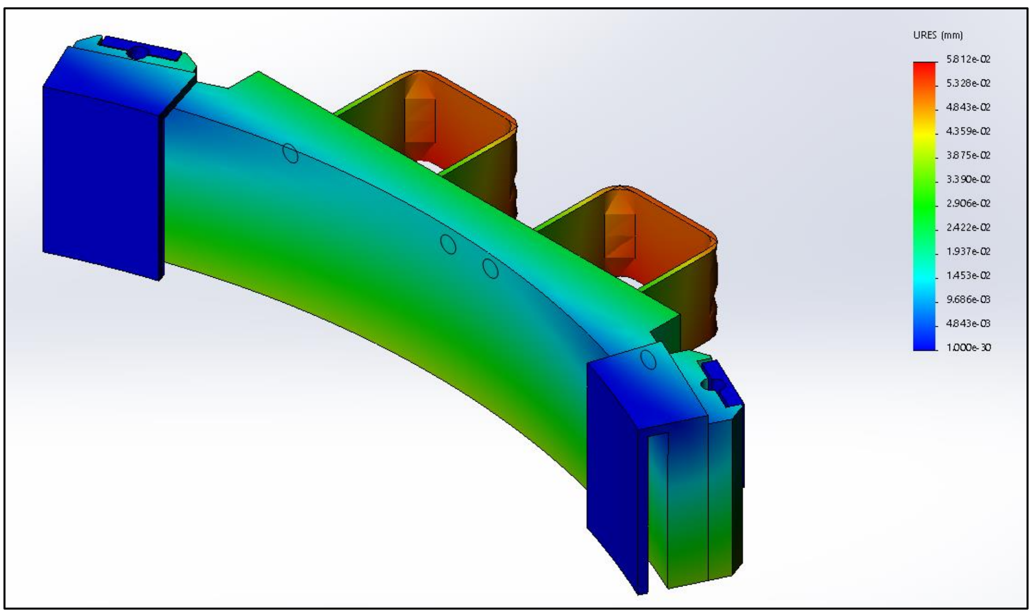

Figure 8: Displacement on the motor brace assembly in mm

As shown in Figure 7, there are significant stress concentrations at the belt clips in this version of design. Additionally, Figure 8 shows that there is a high concentration of displacement on the motor mounts. To accommodate for this, the design needs to be amended. Shown below are the Von Misses stress and displacement stress charts for the design revision after the initial FEA analysis.

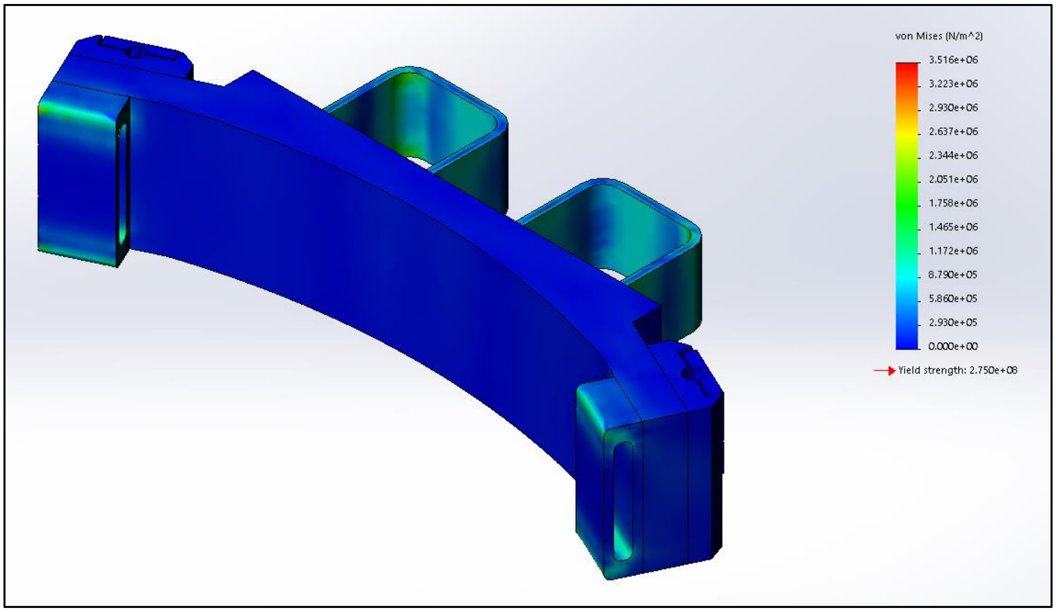

Figure 9: Von Mises on the version 2 motor brace assembly in Nm^2

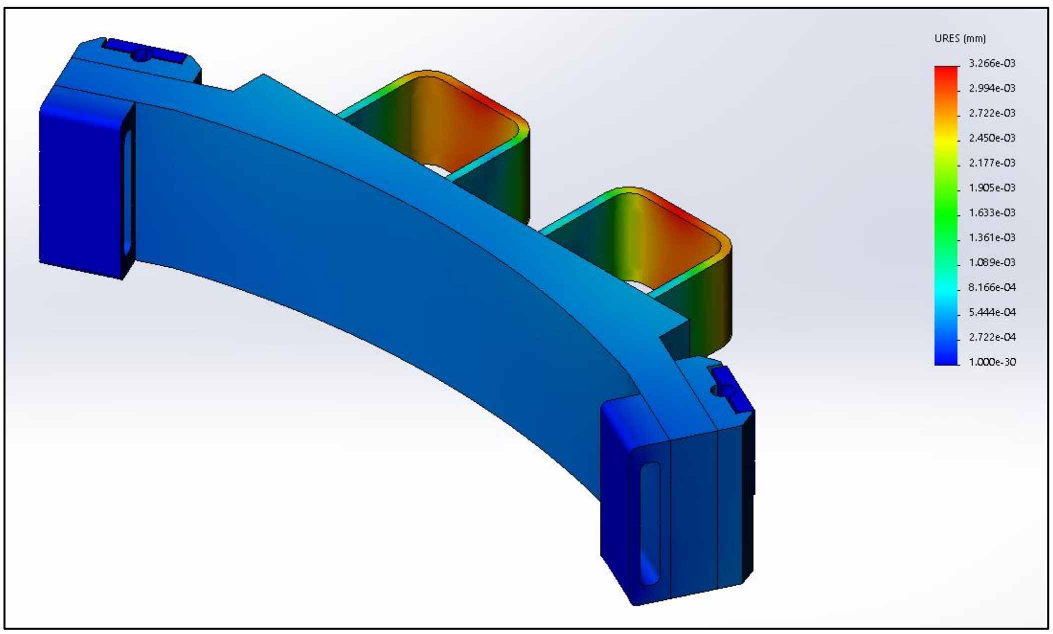

Figure 10: Displacement on the version 2 motor brace assembly in mm

In this improved design (version 2) the thickness of the motor mounts was increased from 0.1 inches to 0.125 and the integrated belt clips were replaced with separate replaceable versions that had thicker material. Comparing figures 7 and 9 shows that the version 2 improvement reduced stress concentrations at the belt clip portion of the design. However, the mounting brackets for the motors could still be problematic based off the FEA analysis. Further design improvements will need to be implemented to ensure that the electric motors are mounted appropriately

Conclusions

Conducting FEA analysis on the components of the hip exoskeleton design allowed for improvements to be made. Key of these was better optimization of the motor brace assembly. The integrated clips in the original design showed to have significant stress concentrations in the belt clip portion of the component. To reduce stress concentrations, non-integrated belt clips with a more robust design were implemented. This improved the results of the FEA analysis. However, the motor mounts are still showing significant amounts of deflection. This is a critical component in the design, so real-world testing on motor mounts is strongly recommended. Accuracy of the FEA models shown in this report will also be improved when force and torque values are obtained from testing the prototype exoskeleton.

Works Cited

[1] R. G. Budynas and J. K. Nisbett, "Chapter 19: Finite Element Analysis," in *Shigley's Mechanical Engineering Design 9th ed*, McGraw Hill, 2011, pp. 955-972.

[2] McMaster-Carr, "Speed-Control Motor," [Online]. Available: https://www.mcmaster.com/7200k1. [Accessed 29 November 2019].