College of Engineering
Mechanical Engineering Department

Final Design Report

ME 490W Engineering Design - Senior Project I
Project 18
Team MAM05-18-B
Race Car Ergonomics Jig (Phase II)

Aztec Ergonomics
Jakob Bravo, Zach Caceres-Batista, Sean Hedgecock, Alexandria Ng, Victor Velazco

Report Submission: 12.11.2024

Narrative:

In the broader context of our final design report, my individual pages serve as a technical "deep dive" into the structural integrity and mechanical feasibility aspects of our ergonomic jig. While the full report examines various subsystems, ranging from materials selection and ergonomics to cost analysis and potential manufacturing processes, my portion shows the engineering analyses that ensure the design will withstand realistic loading conditions. By discussing finite element analyses, free body diagrams, stress distributions, and factors of safety for critical components like the seat exo-skeleton and pedal box mount, my individual pages help the reader understand theoretical concepts and their practical applications. In other words, my contributions to the final design report ensure that the discussions in other sections of the report about comfort, adjustability, and life-cycle are all based on engineering principles.

When viewed in the context of the final design report, my individual pages accomplish two key objectives. First, they validate that the structural subsystems can meet the intended performance requirements. This confirmation lends credibility to other sections that detail the user experience, manufacturing steps, and future improvements. Second, they demonstrate that our team's conceptual ideas have been backed by analytical and computational methods. As a result, decision-makers, potential users, and anyone reading the report can trust that the final product's form and function rest on a sound engineering foundation, calculations, and analyses.

The revisions I made to these pages were largely informed by feedback from the professor, instructional assistants, writing consultants, and my peers. Early drafts, while technically accurate, were dense and sometimes difficult for a non-specialist audience to follow. The writing consultant recommended breaking up lengthy sentences and reorganizing complex ideas into shorter, more streamlined statements. Acting on this tip, I reduced overly technical phrasing in places where a simpler explanation would suffice. Likewise, the professor encouraged clearer transitions between paragraphs to ensure that each new section flowed logically from the one before it. I achieved this by adding topic sentences that set the stage for the subsequent technical details.

My peers and instructional assistants also suggested incorporating more visual references, such as figures and diagrams, to reinforce the written explanations. Though these figures were already present, I refined the accompanying captions and descriptions so that the imagery supported, rather than merely supplemented, the text. My inclusion of references to the appropriate figures and appendices in the text allows readers to easily locate the associated visual representation or calculations that provide support for the analyses and outcomes, and allows readers to gain a deeper understanding of the technical aspects of the report. I also clarified the boundary conditions and loading scenarios in my FEA discussions to help readers understand not just the results, but also the assumptions behind them. In sum, by focusing on clarity, organization, and directness, I used the feedback I received to produce a more cohesive, reader-friendly, and technically sound contribution to our final design report.

Individual Pages:

2.4 Engineering Analysis

The engineering analysis conducted for the ergonomic jig centers on verifying the structural integrity of its critical components, including the seat mount module, the modular pedal box mount, and the overall frame structure. The main purpose of the entire analysis is to share the story of how our calculations began and the path taken to find the most important information that impacts the decisions behind our design. Standard analytical methods, combined with numerical approaches employing both SolidWorks Simulations and Ansys Finite Element Analysis (FEA), were used to evaluate the jig's performance under expected operational loads. Edge case load scenarios and boundary conditions were also considered to gain insight into how stress distributions and overall factors of safety (FOS) would respond when the jig is subjected to extreme conditions.

Seat Mount Analysis

2.4.1 Design Configuration and Load Distribution

In the final seat mount configuration for our project, the telescopic punched tubing rail is anchored at both ends with pivot joints and integrated into a seat exo-skeleton. This design provides a more uniform load distribution than earlier concepts, like the cantilevered or single-pivot setups, which tended to concentrate bending moments and introduce high stress peaks. Under the guidance of Professor Ayala, this approach makes sure that the load is supported at both ends of the rail, minimizing localized stress concentrations. The seat exo-skeleton, bolted beneath the seat with a four bolt pattern, incorporates two internal T-matic connectors and a gusseted corner bracket. The main idea is to strengthen the connection between the seat and the frame, as shown in Figure 13. These components channel the load along well defined paths and help ease the loads on the pivot assembly itself.

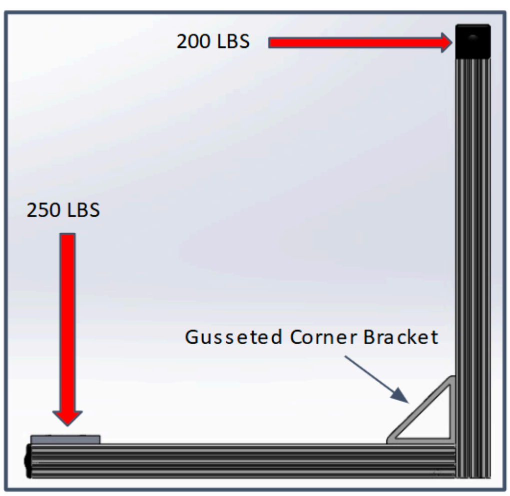

Figure 13: Schematic and Load distribution of the Seat Exo-Skeleton

2.4.2 Free Body Diagram, Shear, and Bending Moment

To confirm the structural integrity of the seat exo-skeleton, initial analyses began with a free body diagram, also known as an FBD, and corresponding shear and bending moment calculations. The lower portion of the seat exo-skeleton, resting on the universal pivot joint, was isolated for this purpose. In Figure 14, Point A is subjected to a 250 lb vertical load while Point C remains fixed. Point B represents another boundary condition involving the 15 Series Universal Standard Structural Pivot and associated pivot tee plates, modeled as a pinned support. After performing the calculations (detailed in the Appendix), the universal pivot joint was found to experience a 616.07 lb shear force and a 2875 lb-in bending moment at the gusseted corner bracket location, as illustrated in Figure 15. Using the bending stress relation, it is evident that the pivot joint could encounter 3883.33 psi under the applied load, depicting the universal joint as an important component to further analyze.

Figure 14: FBD of Seat Exo-Skeleton

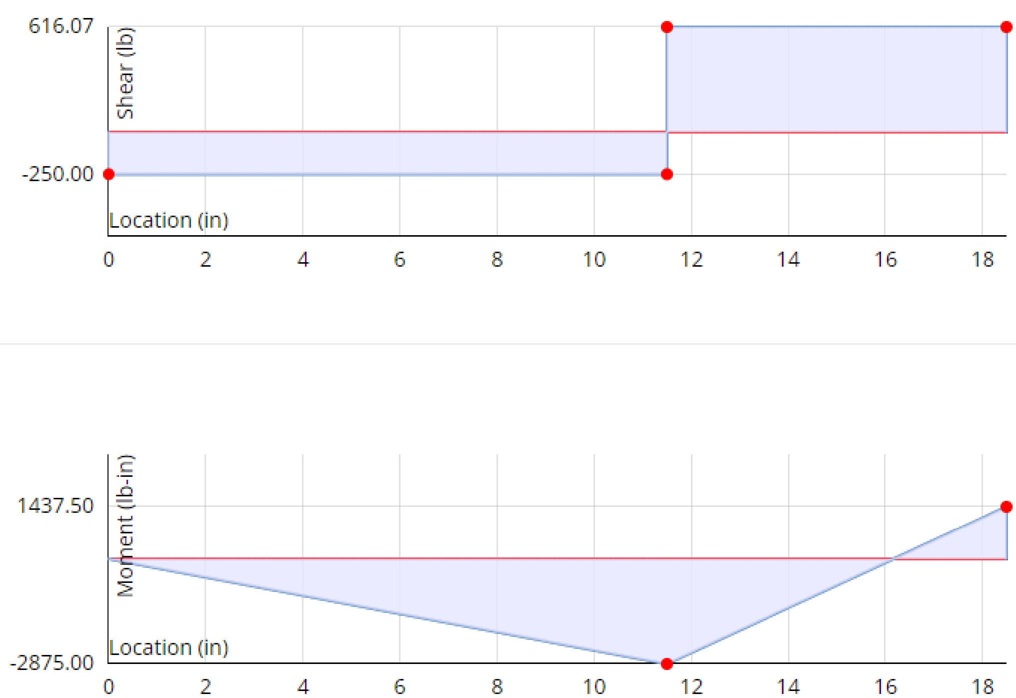

Figure 15: Shear and Bending Moment Diagram for Universal Pivot Joint

2.4.3 Finite Element Analysis Validation

With these initial analytical insights in hand, an FEA study was performed to validate the stress distributions and confirm the universal pivot joint's capacity. The seat exo-skeleton material properties including 6061-T6 pivot joints and 80/20 6105-T5 aluminum extrusions, offer yield strengths of 40,000 psi and 35,000 psi, respectively. The FEA featured boundary conditions of two fixed hinges representing the universal pivot joint at the bottom rail and another pivot joint attached to the top T-frame of the seat exo-skeleton. The applied loads included the original 250 lb vertical force and an additional 200 lb horizontal force, simulating an extreme load case scenario. Figure 16 illustrates the SolidWorks simulation of this first Von Mises stress study.

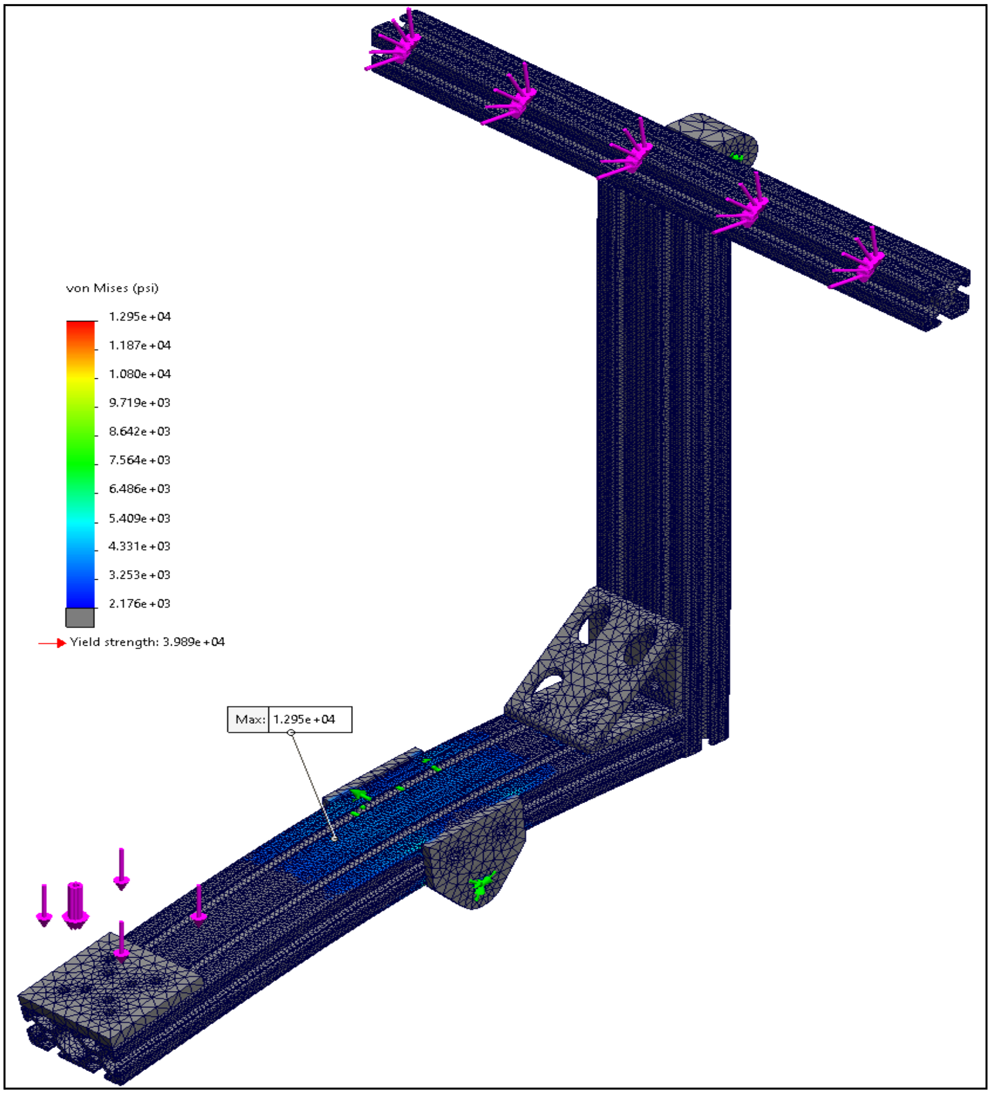

Figure 16: FEA of Seat Exo-Skeleton Von Mises Study 1

The results indicated that the maximum Von Mises stress reached approximately 12,950 psi, staying below the yield strengths of the materials used.

To fully understand the component's performance, a FOS study was also conducted. As shown in Figure 17, the minimum FOS reached about 3.08, providing a stable outcome above the yield point.

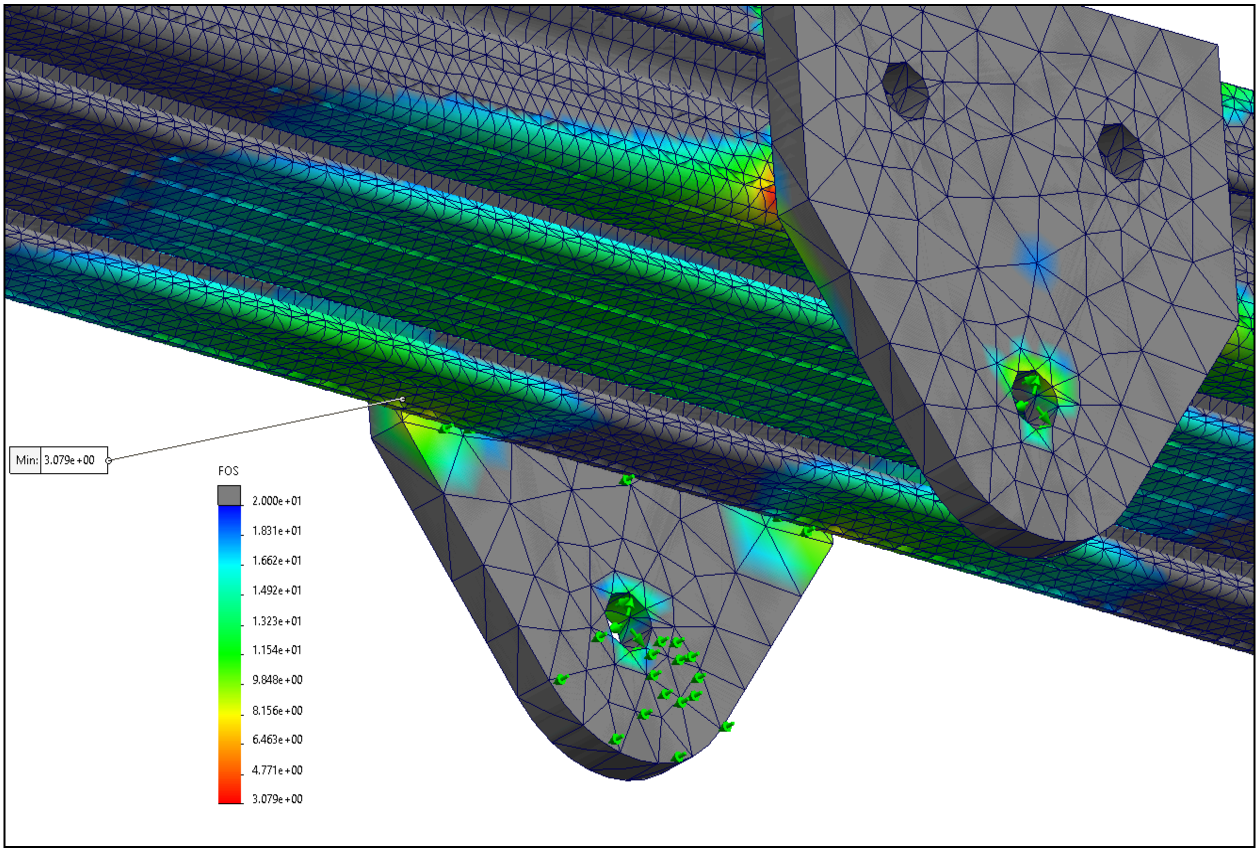

Figure 17: FEA of Seat Exo-Skeleton FOS Study 1

The FOS study shows that the primary area of stress concentrations will occur toward the outer surface of the 80/20 extruded aluminum rather than inside the channels. This is interesting because it shows proof the centroid plays a part in focusing the largest stresses at the farthest points from it. In mechanics of materials, this is consistent with the principle that bending moments create higher stresses at locations farther from the neutral axis (known as the centroid), where the moment of inertia is greater. The FEA further supports this by portraying higher FOS values near the centroid and reduced FOS closer to the outer edges. This validates the relationship between geometry, centroidal distance, and stress distribution. With this in mind, the universal pivot joint and the seat exo-skeleton as a whole will maintain their structural integrity.

To gather more data on this subject, a second FEA study was conducted with a fixed bottom rail to examine how a strictly horizontal load would translate across the seat exo-skeleton. A sinusoidal force distribution was used in SolidWorks to produce a more realistic representation of load transfer, making sure that no single point would take on an unrealistically high portion of the stress. With a 200 lb horizontal load applied under these conditions, the study revealed that the concentrated Von Mises stress of about 3,461 psi takes place near the lower portion of the vertical rail, as seen in Figure 18. This second study displayed the gusseted corner bracket area as a centralized stress region.

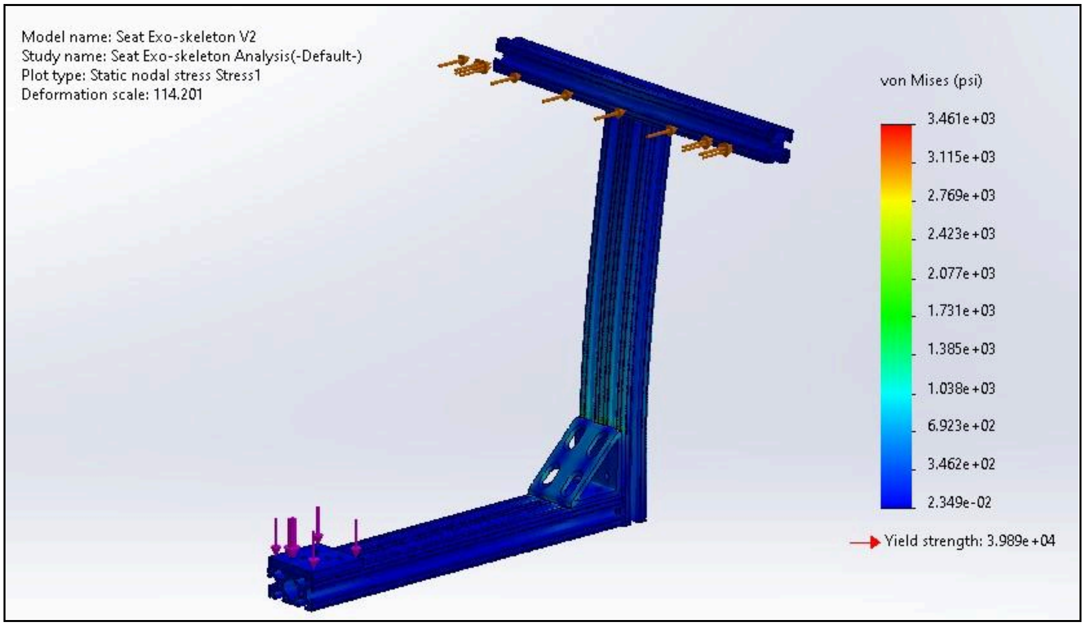

Figure 18: FEA of Seat Exo-Skeleton Von Mises Study 2

Using Ansys in Figure 19, the corner bracket was isolated and subjected to both a 200 lb horizontal load and a 250 lb vertical load, with the geometry fixed at all four screw holes on each plane. This isolated analysis allowed a more detailed examination of how stresses propagate through the bracket and into the mounting interface. Once again, the methodology made sure that boundary conditions realistically approximated how the bracket would function in the assembled jig.

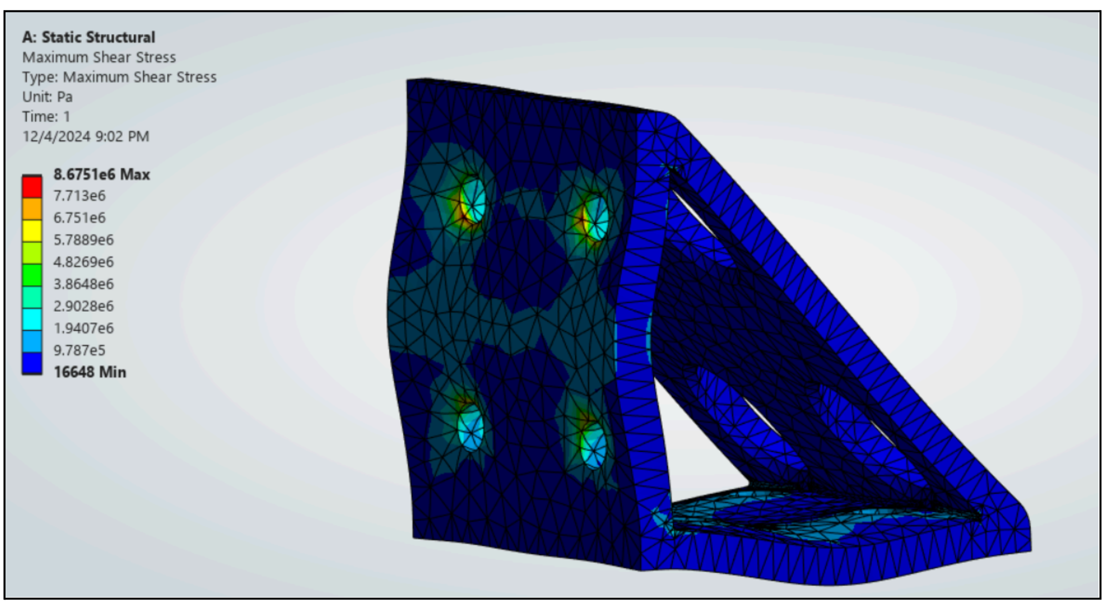

Figure 19: FEA of Gusseted Corner Bracket Maximum Shear Stress Study 2

After isolating the corner bracket, the results showed that the primary areas of peak stress appear around the bolt holes and critical interface regions. Previous analysis, shown in Figure 18 for the

Maximum Shear Stress Study 2 scenario, confirmed that the resulting shear stress of approximately 1,258 psi remains under the 40,000 psi yield strength of 6061-T6 aluminum. The minimum FOS was found to be approximately 32.12, demonstrating its ability to be a solid component of the assembly.

The improved, pivoted telescopic rail and exo-skeleton arrangement promotes a more uniform stress distribution compared to the previous Unibearing based design. Distributing forces more evenly and employing sinusoidal bearing load applications, the front and rear mounting points share the applied loads, preventing the formation of critical stress concentrations in areas that could be detrimental. This helps maintain the structural integrity under operational conditions while preserving the flexibility to adjust seat angles and positions without compromising safety.

Pedal Box Mount Analysis

Under the revised design, the pedal box mount uses bolted side plates and stainless steel thumb screws (wingnuts) rather than the 10-18 anchor fasteners for securing the mounting plate to the T-slot aluminum base frame. Although the fastening method has been modified, the loading scenario remains consistent. An estimated maximum braking force of approximately 450 lbs is transmitted through the pedal box into a 6063-T6 aluminum mounting plate. Four 304 stainless steel screws share this load primarily in shear, resulting in per-screw shear stresses under the material's shear strength. For example, with each screw subjected to roughly 1,984 psi, the FOS against failure remains at 14.98, showing that the screws can reliably handle the applied load. The bearing stress in the aluminum plate stays around 1,125 psi, which is below the 31,000 psi yield strength of 6063-T6 aluminum, providing long term durability without permanent deformation. Refer to the appendix for the calculations.

To get a better idea of hypothetical load conditions, it is important to translate the driver's applied force at the pedal into the reaction forces and bending moments acting on the mount. Although the geometry of the pedal itself is not the subject of this analysis, the outcome of that geometry can be used as input. For instance, when a 450 lb brake application is considered, reaction forces on the mount can be hundreds of pounds, and bending moments can reach several hundred in-lb depending on where and how the load is transferred through the pedal and onto the mounting plate. These reaction forces and moments show that the mount is not only resisting a simple downward force, but a more complex combination of shear and bending.

To validate the calculations, FEA is conducted on the pedal box mounting assembly. The boundary condition setup is depicted in Figure 20. The FEA model includes the pedal box base frame, alloy steel screws, and the mounting plate. The brake load is applied as a concentrated force at the mounting plate to replicate the load path coming from the pedal box. Constraints are placed along the mating surfaces of the frame and plate, reflecting how the assembly is fixed in a real scenario. Material properties for aluminum 6063-T6 and stainless steel fasteners are used based on online databases, providing an accurate simulation environment.

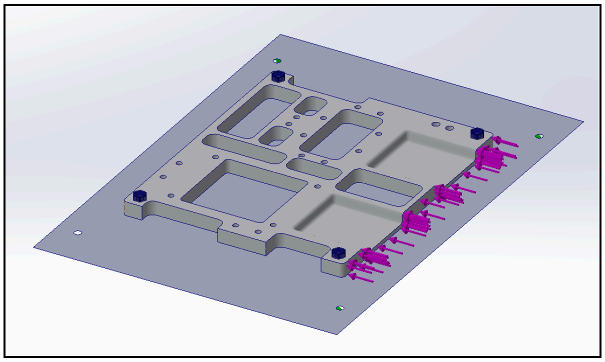

Figure 20: Pedal Box Mount Boundary Conditions

The 450 lb force was applied to the pedal box fixed on the mounting plate as seen in Figure 21 below . Under the simulated 450 lb braking load, the peak von Mises stresses remain below the yield strengths of both the aluminum and steel components. This provides insight into the stress distribution and concentrations. Since the observed stresses are modest compared to the material limits, and deformation is minimal, the mounting assembly's design is verified by both theoretical and computational methods.

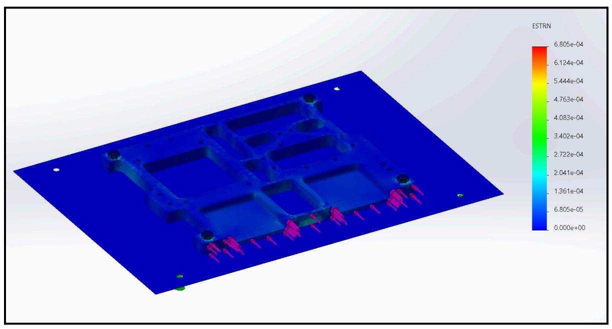

Figure 21: Pedal Box Mount FEA

Base Frame Structure Load Path Analysis

The improved base frame structure load path analysis reflects the changes made to both the connection method and the base frame's elevation. Unlike previous iterations, the frame is no longer raised off the ground by supports. Instead, the T-slotted aluminum extrusions rest directly on a level surface, removing a potential source of bending moments and reducing the complexity of the load path. This direct contact with the ground allows the frame to distribute vertical forces more uniformly. It also increases stability and minimizes the risk of unwanted deflection. Within this design, the previously used hinges have been replaced with internal 15 Series 5/16-18 butt fasteners and metal dowel guiding pins, displayed in Figure 22. The butt fasteners, made from aluminum alloy with a 35,000 psi yield strength, create a flush, rigid connection by engaging both ends of the T-slotted profiles. Even if the bending moments at the joint approach a few thousand lb-in, which are values from earlier calculations, the increased stiffness and yield strength of the 80/20 6105-T5 aluminum extrusions and internal fasteners creates more strength and precision.

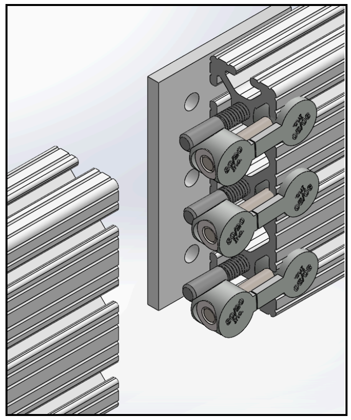

Figure 22: Base Frame Fasteners and Guiding Pins

The metal dowels, threaded on one end and composed of 18-8 stainless steel, serve as guiding pins to align the profiles. The alignment mitigates shear-induced misalignment, improving the load transfer between frame segments. An external flat plate with six bolt holes serves as a redundant load path, bearing part of the shear and bending loads. By distributing the applied forces through the internal butt fasteners, dowel pins, and external reinforcement plate, the assembly prevents deformations and reduces stress concentrations that could have a negative impact on structural integrity. With the frame resting directly on the ground, vertical loads are no longer transmitted through secondary support structures, simplifying the load path and creating better stability.

Summary and Design Impacts

What was revealed by this version of the design is that it provides several benefits and opportunities. First, the pivoted telescopic seat rail and exo-skeleton assembly offer a more uniform stress distribution than earlier configurations with its calculated Von Mises stress of 12,950 psi. The improved load paths reduce peak stresses at critical joints and connections, such as pivot points and corner brackets, leading to a factors of safety of 3.08 and 32.12 respectively. Also, the revised pedal box mount configuration, featuring bolted side plates and thumb screws, simplifies adjustability and maintains a high 14.98 factor of safety, meaning the assembly remains secure under high braking forces. Our main goal is to provide a measurement device that is not only strong, but can be safely used.

By taking out the hinges and support feet beneath the base frame and the introduction of internal butt fasteners and alignment dowel pins, the jig has a more streamlined load path. This simplification provides better stability, reduces the likelihood of misalignment induced failures, and allows for more direct translation of user-generated forces into the ground. These improvements all enhance the jig's performance while opening the door to potential future refinements. Overall, these measures bring a stronger, user friendly, and maintainable ergonomic jig that can evolve alongside future designs of AER.