Design and Analysis of a Vertical Thrust Stand for Liquid Rocket Engines

Vraj Patel[1], Luke Fritz[2], Luke Raque[3], Pierre Bougrat[4], Jacob Bunch[5], Zachary Moore[6], Keller Powell[7], Vasilios Ekimogloy[8], Jack Redden[9], Lucian Stelk[10]
The University of Alabama in Huntsville, Huntsville, AL, 35899, USA

Tartarus, the liquid rocketry project part of the UAH Space Hardware Club, has designed a thrust stand for mounting and recording data from hot-fire tests of liquid rocket engines. The thrust stand is rated for an engine with a thrust of 1000 pounds of force (LBF) with a safety factor of 3 in order to meet both the operating thrust of the "Prometheus" engine, which was designed by Tartarus and is scheduled to be tested on the thrust stand later this year, as well as accommodate future engines with minimal modifications required. The thrust stand is split into 4 subcomponents: the thrust stand frame, the flame diverter, the thrust structure, and the igniter fluids system. The thrust structure is used to mount the test article engine with 4 rail guides restricting horizontal movement so the engine can be mounted on 2 load cells. The "Prometheus" engine uses a torch ignition system which will be mounted along with its propellant tanks to the back of the thrust stand. Stress and displacement finite element analysis (FEA) was performed on the thrust stand frame as well as the flame diverter. Further details on the subcomponent design as well as the thrust stand structural calculations will be covered in the paper.

I. Introduction

The vertical thrust stand outlined in this paper was created for use by Tartarus, a team within the Space Hardware Club of the University of Alabama in Huntsville (UAH). The project is mainly focused on the testing and validation of research done to create the engine, known as Prometheus, before applying this knowledge to create further iterations. The Propulsion subteam handles the modeling and development of the engine. This subteam sets propellant mass flow and injector inlet pressure requirements, which are then passed onto the Fluids subteam. This subteam is mainly concerned with the design and manufacture of all required propulsion ground support systems, such as the Modular Fluids System and the Engine Thrust Stand. The main goal of the Engine Thrust Stand, known internally as the ETS, is to be strong enough to safely support the projected thrust of 280 lbf created by the vertical firing of the Prometheus engine, which uses liquid oxygen (LOx) and ethanol as propellants. Additionally, the thrust structure must be mobile and able to be transported easily due to the inability to test in-house on the university campus, including the need for the structure to be transported safely and without any breakage. With that, the structure is required to be easy to assemble as well as adaptive to the environment in which the engine will be fired. The design, development, and analysis of the Engine Thrust Stand have been concluded to address these issues.

---

[1] Undergraduate, Mechanical and Aerospace Engineering Dept, vpp0004@uah.edu, Student Member, 1603816.
[2] Undergraduate, Mechanical and Aerospace Engineering Dept, lwf0003@uah.edu, Student Member, 1604565.
[3] Undergraduate, Mechanical and Aerospace Engineering Dept, lrr0011@uah.edu, Student Member, 1604908.
[4] Undergraduate, Mechanical and Aerospace Engineering Dept, pgb0008@uah.edu, Student Member, 1605022.
[5] Undergraduate, Mechanical and Aerospace Engineering Dept, jwb0057@uah.edu, Student Member, 1810986.
[6] Undergraduate, Mechanical and Aerospace Engineering Dept, zdm0009@uah.edu, Student Member, 1422186.
[7] Undergraduate, Mechanical and Aerospace Engineering Dept, kbp0020@uah.edu, Student Member, 1810961.
[8] Undergraduate, Mechanical and Aerospace Engineering Dept, vekimogl@uah.edu, Student Member, 1823356.
[9] Undergraduate, Mechanical and Aerospace Engineering Dept., jcr0204@uah.edu, Student Member,1745440.
[10] Undergraduate, Mechanical and Aerospace Engineering Dept, ljs0027@uah.edu, Student Member, 1604671.

II. System Requirements

In order to achieve the goal of making a transportable, durable, and reusable system for testing purposes, the vertical thrust stand is built to these following requirements: Firstly, the vertical thrust stand is built to withstand thrust up to 1000 lbf with a safety factor of 3 for use with the Prometheus engine and later iterations, allowing Tartarus the ability to test without the need for a total redesign of the system. Secondly, the stand must be designed to fit within a standard truck bed to allow the system to be easily transported without requiring any specialized equipment. Next, the flame diverter within the test stand must be built to withstand 3000° Fahrenheit as well as an expected 280 lbf of thrust from the Prometheus engine. Fourth, the stand must be built with the entire ignition system integrated into its frame, allowing the entire system to be more compact and easily transported, as well as able to be placed a safe distance of 7 feet away from the trailer. Lastly, due to a limited budget, the total cost of the vertical thrust stand must not be in excess of $2000.

III. Design Overview

A. Main Structure

As mentioned in the system requirements section, the entire Engine Thrust Stand assembly must be compact and portable, yet also be able to safely withstand the forces associated with engine testing. The overall vertical configuration was chosen to avoid any propellants pooling in the engine and causing a detonation wave, also known as a hard start. To limit possible points of failure, welded joints were avoided due to concerns with manufacturing quality and consistency. Because of this, the Engine Thrust Stand's main structure consists of lightweight but strong 2-inch square steel tubing connected using custom-fabricated angle brackets. Two tubing sections arranged in an X form a 42.5-inch by 42.5-inch base, and 4 main axial members are angled 15 degrees inwards. These sections connect to the engine cage at the top, which will be discussed in later sections. The overall height of the structure is 62.5 inches from the ground to the top octagonal plate. In addition, lateral 1-inch tubing is mounted roughly halfway up the axial members. The overall trapezoidal shape and cross bracing provide lateral stability to mitigate any possible effects from engine thrust load oscillation, wind loads, or any other undesirable instability. To provide further stability, the structure will be tied down to the ground using steel cables. These cables attach to the structure using eye bolts located at the midpoints of each axial member, and they will attach to the ground using eye bolts in the testing ground provided by Cecil Spaceport.

Fig. 1 CAD Model of the Engine Thrust Stand

B. Engine Cage

The engine cage is the substructure of the ETS in which the Prometheus engine itself is housed. Additionally, it also contains the fluid system components for the built-in torch ignition system. This assembly consists of two octagonal plates, which are 15 inches from end to end, with eight 2-inch square steel tubing sections positioned vertically between the two plates. Every side of the cage will have a ¼-inch thick steel blast shield, except the front side, which will stay open for better viewing in the field. The vertical members will be attached to the octagonal plates using custom-made right-angle brackets. The Prometheus engine itself will be mounted to 4 custom brackets made of cut angle iron sections, which will all be connected to the engine cage using rail guides to allow for vertical movement. Two of these brackets will connect to two S-type load cells bolted to the underside of the top plate, while the other two brackets are not part of the main load path but instead provide extra stability. Finally, the entire engine cage will attach to the 4 main axial members using custom 75-degree-angled brackets.

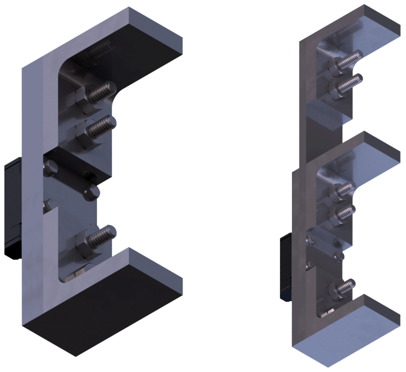


Fig. 2 Non-Load Cell and Load Cell Variants of Engine Brackets

C. Igniter Fluids Panel

In order to ignite the propellants inside the chamber, a specialized ignition system is required. Due to the use of ethanol for film cooling, a much higher temperature from the ignitor is required in order to ignite the propellants. The team decided to use commercial off-the-shelf propane and gaseous oxygen (GOx) tanks piped into an in-house machined copper ignitor body with a remotely activated spark plug. The ignitor is secured on top of the Prometheus engine, with an octagonal panel above containing the tubing, and a panel on the back side of the ETS holds the propane and GOx tanks.

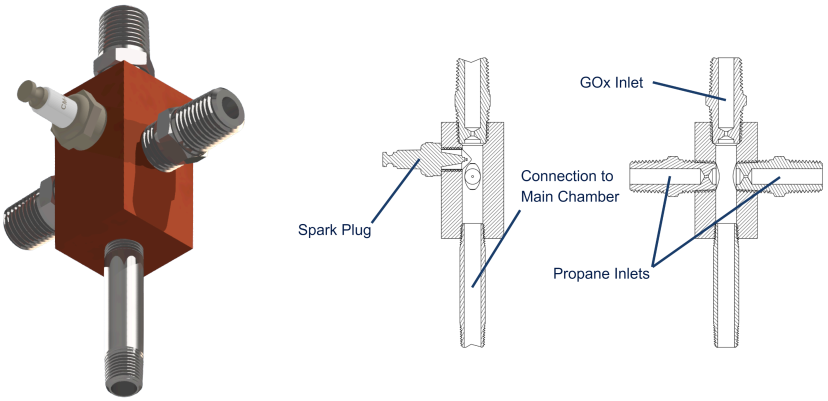


Fig. 3 CAD Model and General Schematic of Ignitor

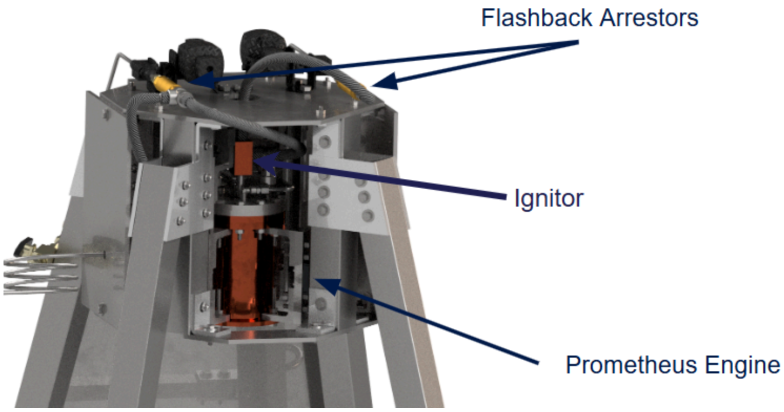


Fig. 4 CAD Model of Igniter Shown Within the ETS

A maximum burn temperature of 1500K for a 5-second run time was predicted by performing chemical equilibrium analysis, thermal FEA, and CFD simulations of the ignitor. To prevent the heat and pressure from the combustion chamber in the engine backtracking into the ignitor, each run valve is followed by a flashback arrestor. These flashback arrestors prevent any backflow above 50 psi, which will keep the ignitor, along with the propane and GOx tanks, safe from experiencing any of the conditions present within the combustion chamber.

As shown in the P&ID in Figure 5, the run valves are pneumatically actuated to ensure the safety of the members and quick, reliable actuation. Additionally, there are manual drain valves on both the propane and the GOx side to prevent any excess buildup between the tanks and valves.

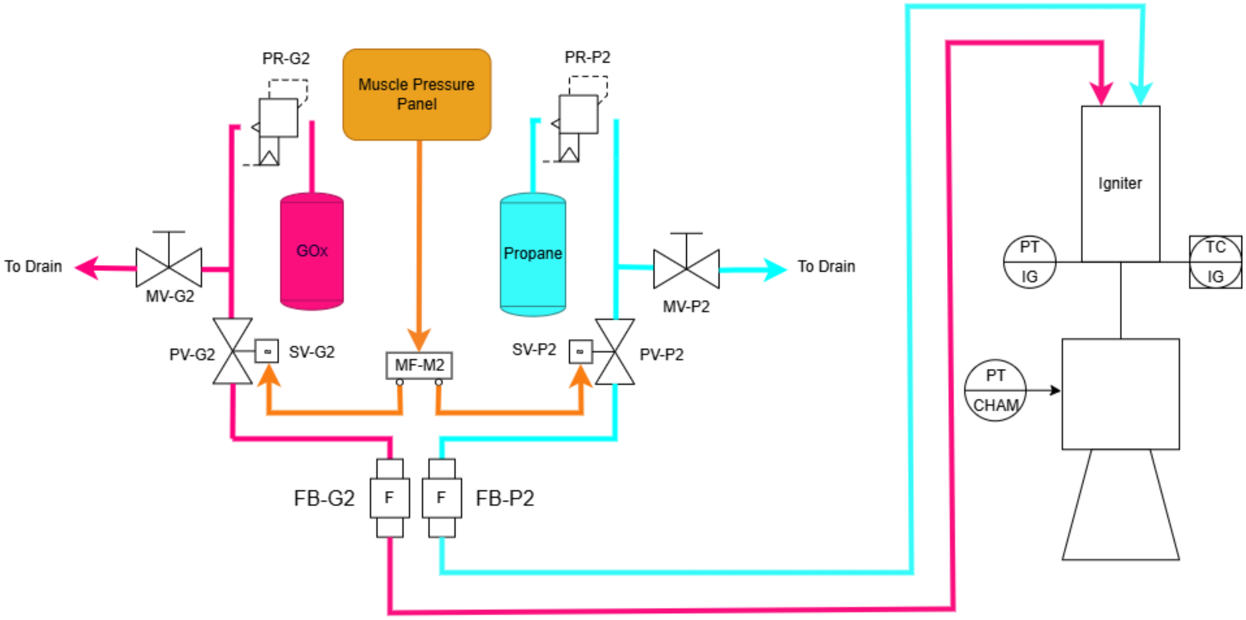


Fig. 5 P&ID of Ignitor

D. Flame Diverter

The flame diverter will be made using a ¼ inch thick steel diverter plate, mounted at an angle to redirect the flame away from the trailer and other hardware. Two L-beams are attached to either side of the plate to ensure that the flame stays away from the equipment, and a layer of concrete will be poured on top of the plate for added protection. The plate will be mounted directly to the bottom of the test stand via four supports made of 1-inch steel tubing, held together with 0.2-inch brackets.

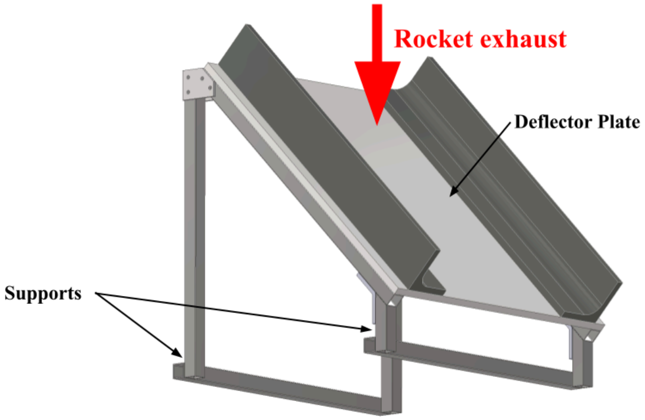


Fig. 6 Flame Diverter CAD

IV. Structural Analysis

Extensive structural analysis was performed on the Engine Thrust Stand geometry during the design phase to ensure that it would be capable of withstanding expected thrust forces with a safety factor of 3. Hand calculations were initially performed as they provided a quick tool to ensure design requirements were met. However, the complexity of the ETS required many simplifying assumptions to be made in order to perform hand calculations on separate subsections of the overall structure. This resulted in calculations that were useful, but ultimately did not accurately capture the stresses imposed on the structure during complex loading conditions.

To start, the stress in the main axial members was calculated by determining the component of the thrust force going through each component aligned with the members themselves. Since the tubing is tilted inward 15 degrees, the normal stress along the axial members was found using Equations 1 and 2 shown below.

$$F_{axial} cos\theta = \frac{1}{4}\left(F_{thrust} - F_{weight,\, engine}\right) \rightarrow F_{axial} = \frac{F_{thrust} - F_{weight,\, engine}}{4cos\theta} = \frac{280lbf - 68.27lbf}{4cos(15)} = 54.8\ lbf \qquad (1)$$

$$\sigma_{axial} = \frac{F_{axial}}{A} = \frac{54.8\ lbf}{2in^2 - 1.8in^2} = 72.105\ psi \qquad (2)$$

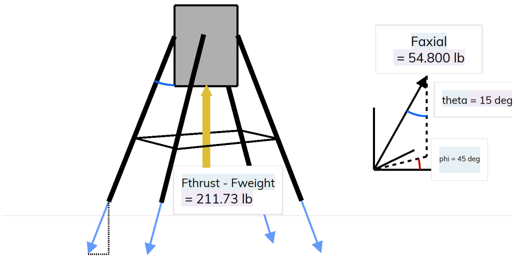


Fig. 7 Simplified Schematic of ETS Used for Calculations

Since the structure is symmetric overall, it was assumed that lateral bending loads would cancel each other out. The cross tubing at the base was surmised to be primarily under bending deformation curling upwards, also known as hogging. Since the applied axial force is angled inward, the compressive force along the base was factored in also. In this scenario, the point of maximum stress would then be close to the center of the cross, where the two sections of tubing meet at right angles with each other. The distance from the point of connection with the axial members to the midpoint of the base was measured to be 23.671 inches. Using this, the bending stress at this point was found using Equation 5.

$$\sigma_{bend,\, base} = \frac{-Mc}{I} = \frac{-(54.8cos(15)lbf*23.671in)*1in}{(\frac{1}{12}*2in*2in^3)-(\frac{1}{12}*1.8in*1.8in^3)} = -2732.550\, psi \qquad (3)$$

$$\sigma_{axial,\, base} = \frac{-F_{axial}sin\theta}{A} = \frac{-54.8sin(15)lbf}{2in^2-1.8in^2} = -18.662\, psi \qquad (4)$$

$$\sigma_{base} = \sigma_{bend,\, base} + \sigma_{axial,\, base} =-2732.550\, psi - 18.662\, psi = -2751.212\, psi \qquad (5)$$

The flame diverter was simplified to be pin-connected at the base, with the total thrust force acting purely vertically at the point of the face located directly below the engine chamber outlet. As such, the values of the upward reaction forces were found first and then used to calculate the resulting stresses in the members. The fact that each reaction force is symmetrically split between a pair of support legs was also accounted for in the following system of equations, Equations 6 and 7.

$$2Ay + 2By = F_{thrust},\ 18in * 2By - F_{thrust} * (18in - 8.353in) = 0 \rightarrow Ay = 64.968\, lbf,\ By = 75.032\, lbf$$

$$\sigma_{diverter,\, a} = \frac{-Ay}{A} = \frac{-64.968lbf}{1in^2-0.87in^2} = -267.247\, psi \qquad (6)$$

$$\sigma_{diverter,\, b} = \frac{-By}{A} = \frac{-75.032lbf}{1in^2-0.87in^2} = -308.648\, psi \qquad (7)$$

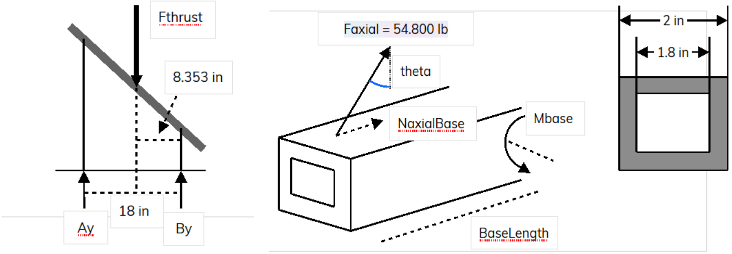
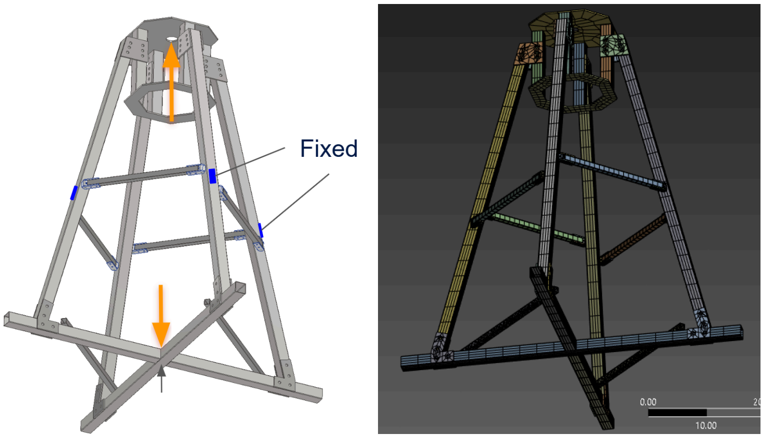


Fig. 8 Flame Diverter and Base Tubing Schematics

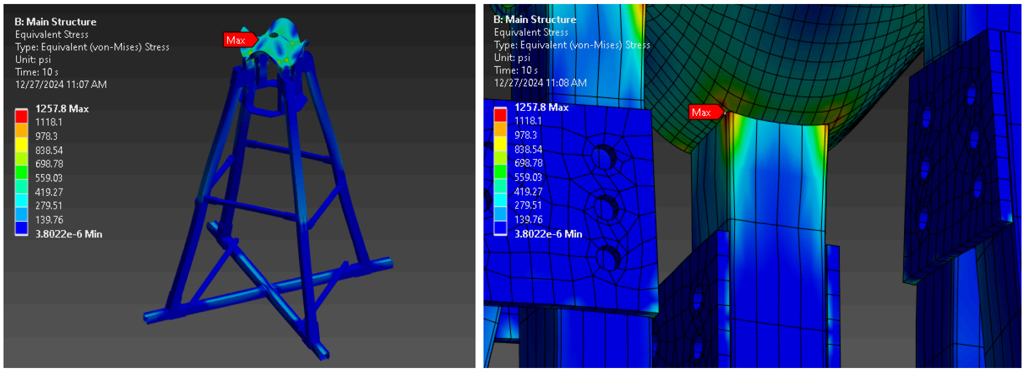
In addition to the hand calculations, finite element analysis (FEA) was also performed to gain a higher fidelity understanding of the stresses the ETS would face and relevant stress concentrations. ANSYS Student's Static Structural package was used to perform this analysis. Due to the ANSYS Student License max cell limit of 32000, the structure was analyzed in three separate subassemblies: a simplified main structural frame, the engine cage assembly, and the flame diverter. All these geometries were meshed in ANSYS, and these meshes utilized plane quadratic elements for better performance in representing complex geometries and bending deformations. Local sizing was also extensively used to locally refine the mesh and more accurately capture material behavior near stress concentrations. These local sizing mesh refinements were done iteratively within the computational constraints of the student license. Bolts were removed from the simulation, and bolted joints were assumed to be fixed, a necessary assumption that may have resulted in some inaccurate solutions. In addition, all of the following simulations were performed using the material properties of A36 structural steel built into ANSYS, which uses a Young's Modulus of 29000 ksi and a yield stress of 36 ksi at 71.6°F ambient temperature. Total deformation, Von Mises stress, and maximum principal strain results were primarily analyzed for all the simulations. Finally, two cases of each subassembly were simulated: one with a nominal 280 lbf thrust load and one with a 1000 lbf thrust load.

A. Main Structure FEA

The main structure assembly FEA aimed to provide a general overview of how the overall frame would deform during hot fires. This assembly was essentially a bare-bones version of the top-level ETS assembly; it excluded the flame diverter, igniter fluids panel, Prometheus engine assembly, and the higher fidelity engine cage assembly. Specifically, the main structure removed the load cells, engine mount brackets, engine carriage rails, and internal brackets. As discussed previously, the ETS will be placed on a sufficiently flat ground roughly 7 feet away from the Mobile Test Stand trailer and be tethered to the ground using steel cable and eye bolts at four midway points on the main axial members. With these assumptions in mind, the following constraints were applied to this FEA case: To simulate these tie-down points, two cells at the midpoint of each axial member were manually selected to be fixed. A compressive normal force was also applied to the base. Finally, a distributed force load equal to engine thrust was applied to the top octagonal plate. It should be noted that this is a simplifying assumption as, in reality, the full thrust load will be going through two concentrated points at the top plate, where the load cell and engine bracket assembly mount to the engine cage. Local mesh refinement was applied to the top and bottom octagonal plates, lateral 1-inch square tubing, and upper 15-degree custom engine cage brackets.



Figs. 9 and 10 Main Structure Constraint Assumptions and Mesh

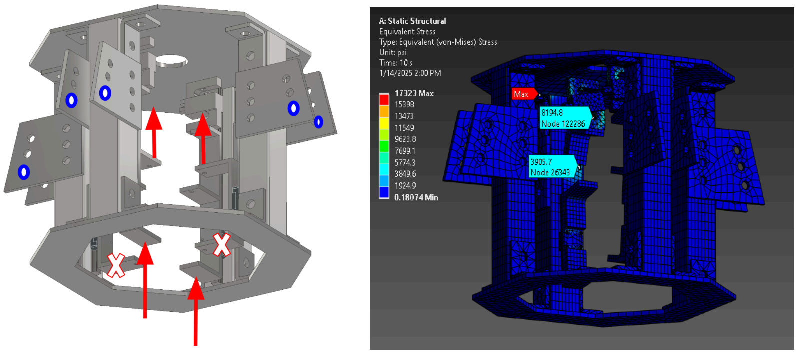


Figs. 11 and 12 Results of 280 lbf Load Case

As mentioned before, due to the simplification of the geometry and the assumptions associated with the constraints applied, the deformation and stress patterns seen on the top octagonal plate specifically are most likely not fully accurate. In addition, extensive local sizing was performed at the connection point of one section of the engine cage vertical tubing and the top plate due to exhibited high stress concentrations on one side of the symmetrically loaded structure. As a result of ANSYS structural solver numerical limits, the brackets that connect the plate and the cage vertical members had to be omitted, meaning that what is a bolted bracket joint, in reality, was treated as a fixed right-angle joint. The maximum expected stress at this point is 1257.8 psi. With this aside, the overall structure exhibited very low overall stress, peaking around 400 psi for the 280 lbf case near the tie-down points and base. For the 1000 lbf case, stress concentrations are near the fixed tie-down points, and the stress of the base was around 1 ksi.

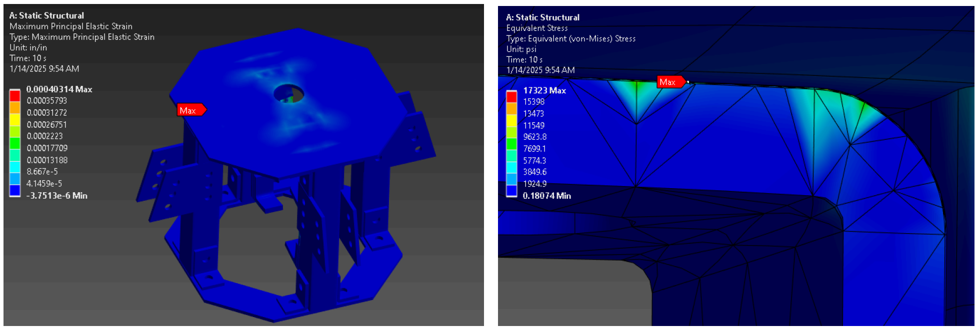
B. Engine Cage FEA

Following the main structure, a higher fidelity simulation of the engine cage geometry alone was also performed. Again, due to numerical constraints, the two engine brackets and rail assemblies that are not connected to the S-type load cell were omitted. These two components are primarily there for lateral structural stability of the engine during hot-fires and are not critical components that are directly in the load path. Therefore, since primarily vertical loads are expected during nominal use, omitting these from the simulation should not greatly affect the accuracy of the overall results. For this simulation, one pair of holes on each of the four 15-degree brackets was fixed, and vertical thrust loads were applied to the faces of the engine brackets that the chamber will directly mount to. In addition, the brackets themselves were constrained in the lateral direction such that they would only be able to move vertically. Local mesh refinement was done on the top plate, load cell geometry, and the rounds on the brackets that connect the top octagonal plate to the vertical members of the engine cage.



Figs. 13 and 14 Simulation Constraints and Structural Mesh

The results of this simulation are shown below in Figures 15 and 16. As expected, the primary stress concentrations are on the top plate brackets and the engine brackets. The engine brackets are expected to take most of the load of the structure and, due to the nature of their lateral constraint, will be in a high compressive and bending stress environment. Simulations show a peak stress of roughly 4 ksi near the upper angle iron piece of the bracket assembly. However, there is also a notably high stress of 17 ksi on the inner edge of one of the top brackets. This concentration is most likely due to the bracket being treated as fixed to the structure when in reality, it is a bolted joint. ANSYS treats interfaces between parts like these as bonded contact regions, where loads and displacements are transported across the faces, but the nodes themselves are not shared in any way. No sliding or separation between the two faces is allowed at all. Moreover, this unusually high stress only occurs on one of the brackets, even though this is a symmetrical structure. In addition, the top octagonal plate's stress distribution is much different from what the main structure analysis indicated. This engine cage simulation took into account the brackets fixing the top plate to the vertical tubing and also applied the load through a more realistic load path with the combined thrust force going through the engine brackets and S-type load cells. As a result, the stress levels of the top ¼ inch steel plate are remarkably low, around 1 ksi at most, and are not an area of concern.

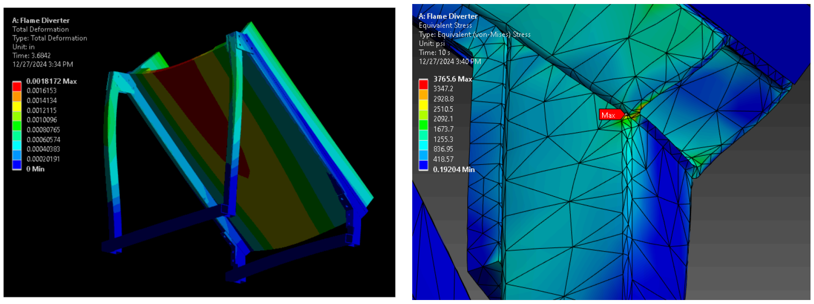


Figs. 15 and 16 280 lbf Engine Cage FEA Results

The same process was done for the 1000 lbf case, which exhibited similar stress distributions, albeit with greater magnitudes. Simulation results show the bracket experiencing a peak stress of 13.8 ksi. The current iteration of the Engine Thrust Stand is not expected to support thrust loads anywhere near this load case without extensive modification. Particularly, the top brackets, octagonal plate and engine brackets must be thickened to account for the increased stress. The S-type load cells would also be chosen to ensure that they are operating well within manufacturer-recommended loads.

C. Flame Diverter FEA

In comparison to the other sub-assemblies of the Engine Thrust Stand, the flame diverter geometry and constraint case are relatively simple. Local mesh refinement was done on the top and bottom custom brackets due to stress concentrations near the bolt holes and on the inner fillet of the bottom face of the angled tubing. This was done to better understand the stress and deformation behavior of these bonded faces. The isolated diverter structure was constrained by applied fixed conditions on the base faces of the vertical 1-inch square steel tubing. The thrust load was then applied on the angled diverter face as a distributed vertical load. The results of the 280 lbf case are shown in Figures 17 and 18 below.



Figs 17. and 18 280 lbf Flame Diverter FEA Results

The 280-lbf simulation results showed a typical sagging deformation of the diverter plate, with a maximum deflection of 0.0018 inches occurring at the midpoint of the top edge. The 1-inch tubing's inner fillet exhibited a maximum stress of 3.8 ksi. Again, the model assumed a bonded contact between the two tubing sections even though they are bolted via a bracket in reality. The 1000 lbf case exhibited similar results, with a max stress of 11.87 ksi at the same location as above, which still fulfills the factor of safety requirement.

Overall, structural analysis indicates that the Engine Thrust Stand will safely be able to withstand the loads required of it for Prometheus engine testing. There are some areas of concern within the design, most notably the top octagonal plate brackets and the engine brackets. To confidently certify the structure for 1000 lbf, slight modifications to these components may be needed. These modifications, if necessary, will be fairly straightforward and will not require any major design changes. Furthermore, real-world testing will give a better understanding of whether or not minor design refinements are needed for a relatively short firing duration of 5 seconds. FEA analysis was extensively used to gain a better understanding of the stresses the structure would undergo during engine testing and to assist in the identification of areas of concern. The simulations showed reasonable results in terms of deformation and stress concentration information and aided in the redesign of the engine bracket assembly.

V. Conclusion

The Engine Thrust Stand is a structure created to allow the vertical firing of the Prometheus engine, a combined ethanol-LOx liquid rocket engine capable of producing up to 280 lbf. To do so, the ETS was constructed out of low-carbon steel and is able to withstand the loads produced by Prometheus, up to a maximum of 1000 lbf, with minimum modifications.

The Engine Thrust Stand is currently being worked on and will be completed soon, and it will be put to the test when the Prometheus Engine is hot-fired this May. The data gathered from the engine fire and from the performance of the thrust structure will help guide the team in the development of the next engine, which is already in the early stages of development. Any changes needed or lessons learned will be implemented in the next test structure, continuing the iterative development of the Tartarus program. The future goals of the project are to work with other teams within Space Hardware Club to create a flight vehicle in tandem with engine development for the purpose of competing in the Spaceport America Cup (SAC) or another similar competition. The passing down of knowledge is the key to this process, as the new generation of Tartarus members will be the future of the project.

Acknowledgments

The authors would first and foremost like to thank Dr. Gang Wang and Dr. Richard Tantaris for their mentorship to The University of Alabama in Huntsville's Space Hardware Club. We would also like to thank Dr. Xu for representing AIAA at The University of Alabama in Huntsville. We are also very grateful to the MAE department at UAH and Mr. Jon Buckley in particular for the use of their machine shop. The Alabama Space Grant Consortium and the University of Alabama in Huntsville have graciously given us the resources necessary to fund this project through the Space Hardware Club. A special thanks to Alloy Valves and Control Inc. for giving insightful technical feedback and providing cryogenic valves for the Modular Fluids System. Finally, we would like to thank all members of Tartarus for their continued support and dedication to the project.

References

[1] Hibbeler, R. C., Mechanics of Materials, 11th ed., Pearson, London, 2022, Chaps. 6, 8.

[2] Matweb Material Property Data "ASTM A36 Steel bar, UNS K02600"

[3] Frenklach, M. & Wang, Hai & Goldenberg, M. & Smith, Gregory & Golden, D. & Bowman, C. & Hanson, R. & Gardiner, W. & Lissianski, Vitali. (2000). GRI-Mech—An Optimized Detailed Chemical Reaction Mechanism for Methane Combustion.