Orion Crew Module / Service Module Structural Weight and Center of Gravity Simulator
&
Vehicle Motion Simulator Hoist Structure for Orion Service Module Umbilical Testing

Peter Ascoli
KSC
Major: Mechanical Engineering
OSSI-NIFS Summer Session
Date: 29 07 2014

Orion Crew Module / Service Module Structural Weight and Center of Gravity Simulator & Vehicle Motion Simulator Hoist Structure for Orion Service Module Umbilical Testing

Peter A. Ascoli*

The Cooper Union for the Advancement of Science and Art, New York, NY, 10003, USA

Michael H. Haddock†

NASA Kennedy Space Center, FL, 32899, USA

An Orion Crew Module/Service Module Structural Weight and Center of Gravity Simulator and a Vehicle Motion Simulator Hoist Structure for Orion Service Module Umbilical Testing were designed during a summer 2014 internship in Kennedy Space Centers Structures and Mechanisms Design Branch. The simulator is a structure that supports ballast, which will be integrated into an existing Orion mock-up to simulate the mass properties of the Exploration Mission-1 flight vehicle in both fueled and unfueled states. The simulator mimics these configurations through the use of approximately 40,000 lb_f of steel and water ballast, and a steel support structure. Draining four water tanks, which house the water ballast, transitions the simulator from the fueled to unfueled mass properties. The Ground Systems Development and Operations organization will utilize the simulator to verify and validate equipment used to maneuver and transport the Orion spacecraft in its fueled and unfueled configurations. The second design comprises a cantilevered tripod hoist structure that provides the capability to position a large Orion Service Module Umbilical in proximity to the Vehicle Motion Simulator. The Ground Systems Development and Operations organization will utilize the Vehicle Motion Simulator, with the hoist structure attached, to test the Orion Service Module Umbilical for proper operation prior to installation on the Mobile Launcher. Overall, these two designs provide NASA engineers viable concepts worthy of fabricating and placing into service to prepare for the launch of Orion in 2017.

Nomenclature

g	Earth's Gravitational Acceleration, ft/s^2
I	Moment of Inertia, in^4
J	Polar Moment of Inertia, in^4
S	Section Modulus, in^3
A	Area, in^2
c	Half Beam Depth, in
L_e	Equivalent Length, in
d	Diameter, in
t	Thickness, in
x	Position, in
m	Mass, lb_m

*Mechanical Design Engineer Intern, Structures and Mechanisms Design Branch, Kennedy Space Center, The Cooper Union.

†Mechanical Engineer, Flight Mechanisms & Flight Crew Systems Branch, NE-M3, Kennedy Space xCenter.

W	Weight, lbf_f
P	Applied Load, lb_f
F	Force, lb_f
T	Torque, lb_f in
M	Moment, lb_f in
V	Shear Force, lb_f
E	Modulus of Elasticity, lb_f/in^2
FS	Factor(s) of Safety, dimensionless
σ	Normal Stress, lb_f/in^2
σ_u	Ultimate Tensile Stress, lb_f/in^2
σ_y	Yield Tensile Stress, lb_f/in^2
τ	Shear Stress, lb_f/in^2
τ_u	Ultimate Shear Stress, lb_f/in^2

I. Introduction

The Kennedy Space Center (KSC) is home to roughly two thousand National Aeronautics and Space Administration (NASA) employees spread across a diverse group of organizations. When a NASA organization requires engineering design services, the Structures & Mechanisms Design Branch (NE-M2) can be utilized to produce the desired product. NE-M2 primarily produces design work in the area of umbilical systems, ground support equipment, and flight hardware handling and transportation.

Two mechanical design projects, an Orion Crew Module / Service Module (CM/SM) Structural Weight and Center of Gravity (CG) Simulator and a Vehicle Motion Simulator (VMS) Hoist Structure for Orion Service Module Umbilical (OSMU) Testing, were completed during a summer internship in NE-M2. Each project is individually outlined throughout this report.

A. Orion CM/SM Structural Weight & CG Simulator

1. Background

Recently, Ground Systems Development and Operations (GSDO) has requested NE-M2 to produce a design concept for an Orion CM/SM Simulator that will mimic the mass properties of the spacecraft to be used for Exploration Mission 1 (EM1), which will launch in 2017. Figure 1 shows a rendering of this spacecraft. In preparation for EM1, GSDO has chosen to use a mock-up of the crew module (CM) and service module (SM) to demonstrate that they can successfully transport the vehicle to the Multi Payload Processing Facility (MPPF) for Verification and Validation (V&V) activities and, once at the MPPF, to move the vehicle to its work stand for V&V tests. More specifically, the requested simulator is required to validate the unloading and movement of the modules to the CM/SM servicing stand using air-bearing pallets and the transportation procedure of the fully loaded crew and service modules from the MPPF to the Vehicle Assembly Building (VAB).

Figure 1. A rendering of the EM1 spacecraft, Orion, with annotated crew (1) and service (2) modules. Image Credit: NASA.

The CM/SM Structural Weight & CG Simulator needed to be designed from scratch. A full scale mock-up structure of the crew and service modules already exists, but it only accurately mimics the Outer Mold Line (OML) of the EM1 spacecraft, not its weight properties. For the purposes of this internship, I produced a completed engineering design concept and the accompanying analysis for an Orion CM/SM Structural Weight and CG Simulator. This contribution provides engineers at NASA KSC with a design worth fabricating that keeps GSDO on target to demonstrate the capabilities of their system(s) in 2016 and implement the necessary V&V tests that will ultimately lead to a successful launch of the Orion spacecraft for EM1 in 2017.

2. Design Constraints

The following bullet list outlines the design requirements, constraints and assumptions:

- The simulator will provide the capability to simulate the weight and CG of the Orion flight hardware short stack in the fueled and unfueled configurations; the mass properties for EM1 and Exploratino Mission 2 (EM2) spacecraft will be provided by GSDO.
- The existing CM/SM mockup stand, SM fairings, outriggers, avionics ring and Crew Module 2 (CM2) constitute the existing CM/SM Mockup Assembly in which the new mass simulator will be installed.
  - The weight and CG simulator must be designed such that it can be installed and secured within the confines of the existing CM/SM Mock-up Assembly (including CM2) without altering the elevation or OML of the existing short stack.
  - Any loads imparted by the new simulator to the CM/SM Mock-up Assembly must remain within allowable structural load limits of the Assembly.
- Table 1[a,1] lists the approximate weights and CG locations of the existing short stack mock-up components and the desired properties of the Simulator per the specifications of EM1 and EM2. The primary objective is to replicate the X_{CG} as its location has a direct correlation with the overturning moments. The minute offset in the Y and Z coordinates of the CG is not considered a driving design constraint as its close proximity to the vehicle centerline adds no discernible value in the context of satisfying V&V objectives.
- The simulator must be designed to withstand transportation loads and maintain Ground Support Equipment (GSE) Factors of Safety (FS) per KSC-DE-512-SM.
- The simulator will be designed to permit installation with cranes and/or forklifts (including hoisting provisions).

[a] X_{CG} datum located at the SM base. Y_{CG} and Z_{CG} datum located at the short stack vertical centerline.

Table 1. Weights and CG Locations

Component	Weight, lb_f	X_{CG}, in	Y_{CG}, in	Z_{CG}, in
CM2 Mock-up	8,860	243.00	N/A	N/A
SM Mock-up	4,400	93.47	N/A	N/A
EM1-Dry	39,412.0	187.05	0.47	-2.58
EM1-Fueled	60,210.0	167.88	2.50	-1.28
EM2-Dry	38,678.9	170.30	0.62	-2.88
EM2-Fueled	61,160.0	186.25	2.47	-1.30

- The SM OML fairing and avionics ring may be removed from the SM stand to permit installation of the simulator.
- Ballast/weights will have to be installed and secured inside CM2 to provide the required short stack CG.
- Ballast/weights will have to be installed and secured beneath CM2 and internal to the SM support stand structure to provide the required short stack CG.
- The simulator should be painted as it will be used in MPPF clean room environment.
- CM2 and SM stand assembly can be hoisted when bolted together.
- The safe working load (SWL) of the sling to lift CM2 is 26,400 lb_f.
- SM Mock-up stand (with simulator installed) will require a complete structural analysis.
- The Lockheed Martin Spacecraft Pallet will be available in time to allow Handling and Access V&V objectives to be satisfied using the CM/SM Mock-up Assembly (including the weight and CG simulator) in the MPPF.
- Lockheed Martin will provide the Interface Control Documentation (ICD) or equivalent engineering documentation defining the structural interface to the spacecraft pallet.

B. VMS Hoist Structure for OSMU Testing

1. Background

In preparation for EM1 in 2017, GSDO will test the new umbilical systems that will be mounted to the ML for the launch of Orion. These umbilical systems will be tested for proper operation at the Launch Equipment Test Facility (LETF) over the next two and a half years. The OSMU, one of these umbilical systems, provides Environmental Control Systems (ECS) and other services to the Orion SM. The various pipes and tubes that provide the services dangle off of the end of the OSMU arm and all terminate at an umbilical plate. In testing, this plate will be connected to the VMS[b] to ensure that all components of the OSMU respond appropriately to the launch and servicing conditions they will be exposed to when EM1 launches in 2017. Figure 2 presents a CAD rendering of this setup.

Because the umbilical plate naturally dangles off of the end of the OSMU structure there is need for a structure mounted to the VMS that can locate the 500 lb_f OSMU plate just inches away from the its connection point on the VMS. Workers will then manually connect the plate to the VMS via another structure that will be mounted to the VMS. Figure 2 shows the OSMU plate in the desired final location. As such, I designed a cantilevered tripod structure mounted to the VMS that places a pulley above the desired OSMU plate location and utilizes a hoist to reel in the plate.

[b]The VMS is a structure that uses hydraulic actuators to simulate rocket dynamics during liftoff.

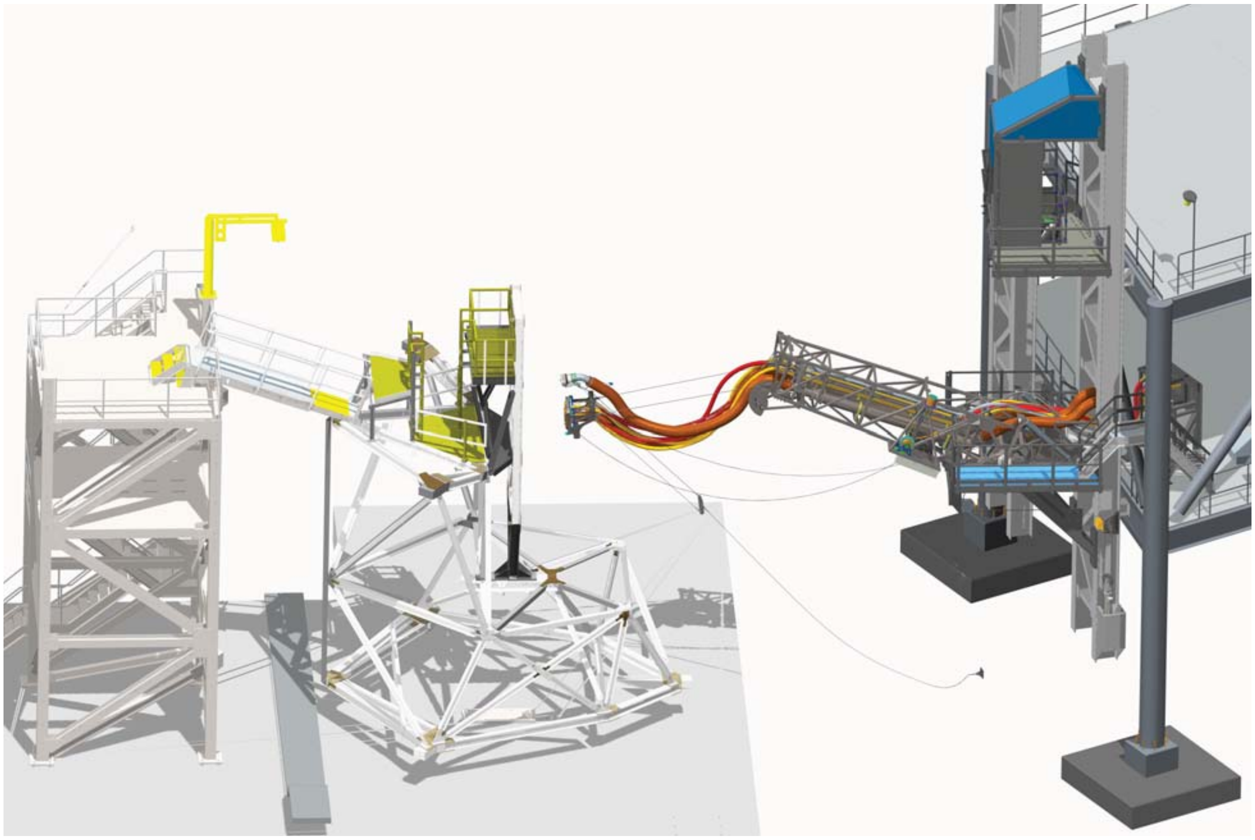

Figure 2. A Creo rendering of the VMS (left) and the OSMU attached to the LETF (right).

2. Design Constraints

The following bullet list outlines the design requirements, constraints and assumptions:

- The structure and its components must be designed to support a 500 lb_f load (the weight of the OSMU plate and servicing tubes).
- The pulley location must be adjustable up to 3 inches away from the nominal position that holds the OSMU plate at a 15° offset counter-clockwise from the vertical.
- The chosen winch/hoist must be 110-120V AC powered, be weatherproof, have approximately 50 feet of rope, pull in all of the rope in under 5 minutes, and adhere to the lifting specifications required by NASA-STD-8719.9.
- The structure must be designed to also withstand a $1g$ vertical acceleration load and a $5g$ vertical deceleration load (the OSMU plate will not be supported by the hoist structure during these accelerations).
- The structure must be analyzed with hand calculations and Finite Element Analysis (FEA) to verify that it complies with the required Factors of Safety and GSE design requirements.

II. Engineering Approach

NE-M2 utilizes hand calculations and basic FEA tools to validate design concepts. Once a design concept is completed the Engineering Design Analysis Branch (NE-M1) uses complex FEA models to verify critical hand calculations and analyze portions of the design where simple governing equations do not exist. As an intern in NE-M2, I utilized hand calculations and simple FEA models to size components for the two design projects. 3D Computer Aided Designed (CAD) models (part and assemblies) of the designs were then created using Creo Parameteric 2.0 software.

The set of equations outlined below form the general engineering design approach used to complete the two projects. Sample hand calculations of the Orion CM/SM Structural Weight and CG Simulator and the VMS Hoist Structure for OSMU Testing are located in the *Appendix* and presented in the context of the two designs.

For static structures Eq. (1) and Eq. (2) are paramount. Eq. (1) states that the sum of all external forces (on an entire structure, and on any individual component) must be zero. Eq. (2) states that the sum of all external moments must be zero. A violation of either of these equations would cause the desired static structure to become dynamic. To develop the two designs, Eq. (1) and Eq. (2) were applied to free body diagrams of individual components and groups of components to calculate the magnitude and direction of reaction forces due to imparted structural loads.

$$\Sigma\vec{F} = 0 \tag{1}$$

$$\Sigma\vec{M} = 0 \tag{2}$$

The resulting forces developed from Eq. (1) and Eq. (2) yield stresses within the components of the designs. Eq. (3) and Eq. (4) represent the most basic forms of tensile (normal) and shear (transverse) stresses respectively. The shape and material of structural components were chosen such that the applied forces resulted in allowable stresses.

$$\sigma = \frac{P}{A_{tensile}} \tag{3}$$

$$\tau = \frac{P}{A_{shear}} \tag{4}$$

The simulator and hoist structure are both large structures with many beams to transfer loads to ground. Eq. (5) and Eq. (6) represent the bending stress and shear respectively in a beam element. V and M were developed using 2D Beam Theory. Beams were chosen based on I, S, A and material so that the resulting stresses satisfied the required FS.

$$\sigma = \frac{|M_{max}|c}{I} = \frac{|M_{max}|}{S} \tag{5}$$

$$\tau = \frac{V_{max}}{A} \tag{6}$$

For beam members in compression, the member must be analyzed to ensure that it will not buckle according to Eq. (7). Subsequently, a beam is chosen based on its material and shape (I) to resist buckling.

$$P_{critical} = \frac{\pi^2 EI}{L_e^2} \tag{7}$$

Shear stress due to torsion and bearing stress are two other critical material conditions to analyze. A beam or weld may not have loads applied directly to their centerline (resulting in a torque). Therefore, Eq. (8) is used to ensure that the material can withstand the tendency to twist. Eq. (9) is utilized to ensure that the stresses imparted on a hole by a fastener, such as a bolt or pin, do not exceed their allowable value.

$$\tau = \frac{Tc}{J} \tag{8}$$

$$\sigma = \frac{P}{td} \tag{9}$$

All structures and equipment at NASA are built with a FS as specified in governing NASA standards.

Eq. (10) presents a variety of FS requirements, which each apply to specific applications. The simulator and hoist structure fall under GSE, which utilize FS of at least 2:1 on material yield strength and at least 3:1 on material ultimate strength. Components used in lifting applications must be designed to 5:1 on ultimate, bearing stresses are held to 1.5:1 on ultimate, and members in buckling are designed to 3:1 on the critical load.

$$FS = \frac{\sigma_u}{\sigma_{allowable}}, \frac{\sigma_y}{\sigma_{allowable}}, \frac{\tau_u}{\tau_{allowable}}, \frac{P_{critical}}{P_{allowable}} \tag{10}$$

Lastly, Eq. (11) calculates the center of mass. Eq. (11) was primarily used in the design of the simulator to determine the locations and magnitudes of the weights to be added to the existing mock-up assembly to satisfy the desired dry and fueled CG specifications.

$$\bar{x} = \frac{\sum_{i=1}^{n} m_i x_i}{\sum_{i=1}^{n} m_i} \tag{11}$$

III. Designs

A. Orion CM/SM Structural Weight and CG Simulator

1. *Design Development*

To design the simulator, the locations of the additional weights were determined first by comparing the CG of the short stack to the CG of the EM1 flight vehicle. Table 2[1] presents the location of the desired EM1 CGs[c] in the context of the short stack structure. For reference, Figure 3 presents screenshots of the short stack structure. While the CG of the fueled flight vehicle lies within the SM, the CG of the dry flight vehicle lies within the interfacing region between the SM and CM2, where no ballast can be placed. Subsequently, it was determined that the ballast must be placed inside both the SM and CM2 to accurately mimic the CG. Table 2 also shows that the ballast that must be added to transition the simulator from the *dry* to *fueled* state only needs to be placed within the SM. Therefore, the majority of the simulator was chosen to be designed to be secured inside the SM, while a smaller portion would be placed inside CM2.

Table 2. Elevation Datums

Component	X, in
Base of SM	0.00
Minimum Operable SM Elevation	12.98
EM1 Dry-to-Fueled Transition CG	*131.05*
EM1-Fueled CG	*167.88*
Maximum Operable SM Elevation	179.48
EM1-Dry CG	*183.84*
Lowest Point of CM2 Heat Shield	184.94
Short Stack Mock-up Assembly CG	*193.38*
CM2-SM Interface	201.13
Minimum Operable CM2 Elevation	213.00
Maximum Operable CM2 Elevation	285.02
Top of CM2	314.20

[c]The Y_{CG} and Z_{CG} are not shown in Table 2 because the displacements from the centerline were deemed to have a negligible effect on the value of V&V activities compared to the value of accurately representing the X_{CG} in transportation.

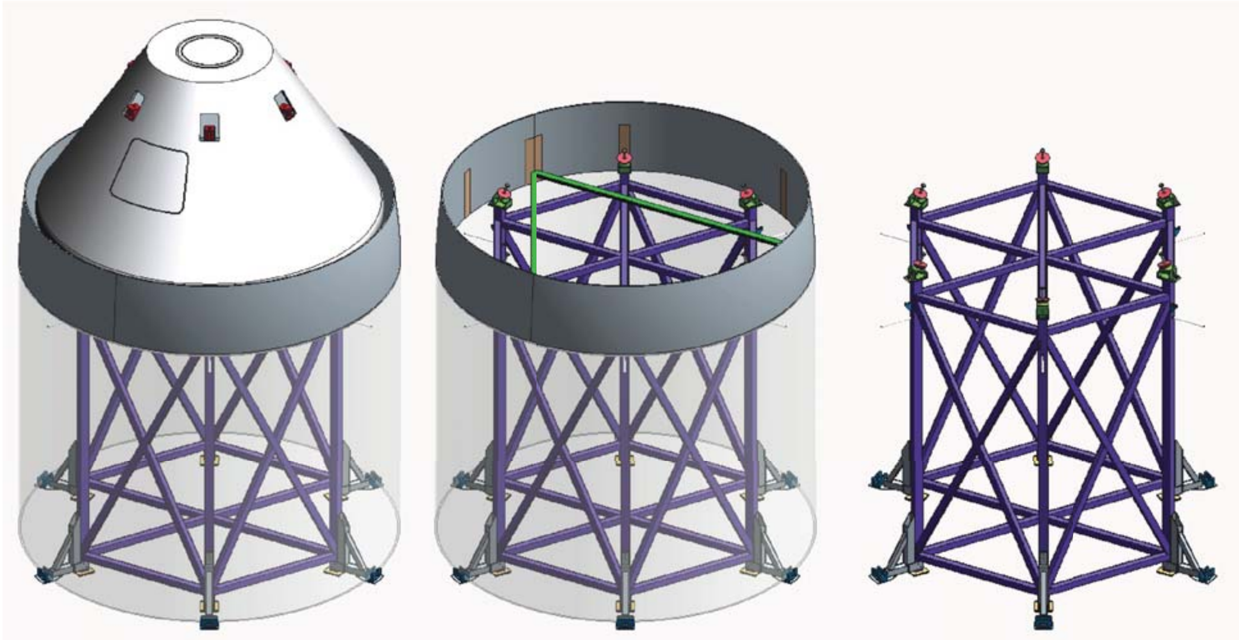

Figure 3. Screenshots of the short stack in Creo: (Left) CM2 and the SM (Middle) The SM (Right) The SM Structural Elements, the SM OML fairing and avionics ring are absent.

To proceed further with the design, the amount of ballast to be placed in CM2 needed to be determined and the structure of CM2 needed to be analyzed to ensure that it could withstand the addition of ballast required to simulate EM1-dry. Figure 4 presents the relationship between the SM ballast and CM2 ballast positions for four different ballast weights placed inside CM2. The blue line represents the maximum amount of weight (13140 lb_f) that could be placed inside CM2 that would permit the short stack loaded with the CM2 ballast to be lifted simultaneously. The trend shows that to provide the maximum flexibility of the X_{CG} of the simulator inside the SM requires maximizing the weight placed inside of CM2.

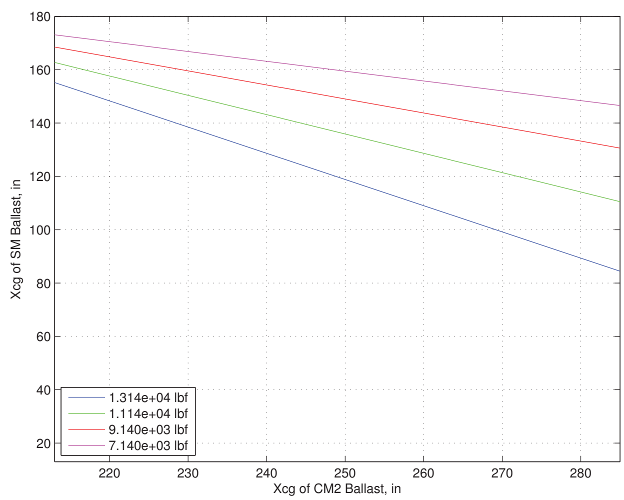

Figure 4. The relationship between the X_{CG} of the SM ballast and the CM2 ballast for different CM2 ballast weights (shown over the operable ranges of the two short stack components).

With the amount of ballast in CM2 chosen to be approximately 13,000 lb_f, the structure was analyzed to verify that the floor of CM2 could sustain the weight and that the short stack could successfully be lifted with the additional ballast. Structural analysis of the CM2 structure for lifting the short stack (without the new simulator) was already performed by NE-M2 and showed that the structure could withstand 40,000 lb_f

of tension through three of its legs and provided specifications on the Launch Abort System (LAS) Attach Fitting, eyebolts and bushings used in the lift.[2] Hand calculations and FEA results demonstrated that the LAS Attach Fittings, eyebolts, bushings, and CM2 structure could withstand the new maximum lift load of 26,400 lb_f. Hand calculations of bending in the floor beams verified that 13,000 lb_f could safely be added to CM2 as a distributed load. Additionally, reports from the NASA Project Orion Flight Test Office Abort Flight Test summarizing CM2 proof-loading tests and FEA results confirmed that CM2's structure could support the additional weight.[3,4] Finally, NE-M1's analysis of the SM confirms that the stand can withstand an additional 13,000 lb_f of load.[5]

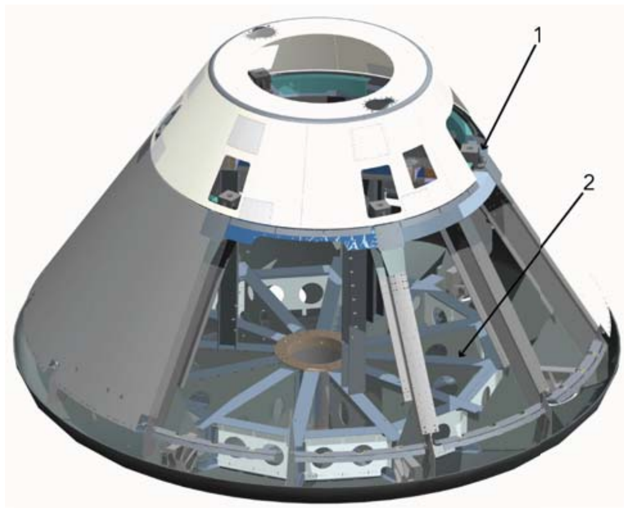

Figure 5. A Creo rendering of CM2's structural elements. Annotations: (1) LAS Attach Fittings (2) Floor Beam (1 of 12).

With CM2 analyzed, the simulator could now be thought of as three individual components. One ballast in the SM and one ballast in CM2 constitute the simulator to mimic the weight and CG of EM1-dry, and an additional removable ballast in the SM to transition the structure from EM1-dry specifications to those of EM1-fueled. Table 3[d] presents these three ballast weights and CGs.

Table 3. Weight and CG Targets.

Component	Weight, lb_f	X_{CG}, in
CM2 Dry Ballast	13140	239.000
SM Dry Ballast	13012	128.129
SM Fueled Ballast	20798	131.553

For easier transition between fueled and dry states, it was decided that water tanks would be placed inside the SM to create the transition ballast. Simple hand calculations for the corresponding volume of water demonstrated that mounting common-shaped water tanks or even oil barrels inside the SM would violate the OML if placed in the proper X_{cg} locations. Ultimately, it was determined that three or four triangular tanks, nearly the size of one of the six triangular openings in the SM's top and bottom radials, would provide the sufficient volume of water, could be placed at the necessary locations, and still leave room to place the remaining SM ballast.

Two initial concepts were modeled in Creo Parametric 2.0 to assess the feasibility of using three water tank transition ballasts. Figure 6 demonstrates the first concept. In this concept, triangular aluminum tanks will hold the water for the SM Fueled Ballast with the goal of installing additional water tanks for the SM Dry Ballast so that the whole unfilled structure could be lifted as one. However, calculations showed

[d]For the purposes of the initial design concept, the X_{CG} of the CM2 dry ballast was chosen to be halfway between the floor and ceiling to increase the flexibility of creating a CM2 ballast. At half the operable height, the ballast is neither too close to the floor nor the ceiling, which could impede its design.

the aluminum[e] structure to support the tanks was not strong enough to support the shear weight of the filled tanks with common structural beam shapes. The second concept was similar but used steel support structure instead of aluminum. At this point, the tanks and supports nearly made up all of the SM Dry Ballast meaning that lifting the simulator would no longer involve the CM2 attached to the SM, but rather, each would be lifted independently. However, the target weights and CGs would remain the same since they permit lifting CM2 and the SM independently with their respective simulator ballasts and lead to structural flexibility for designing the SM simulator.

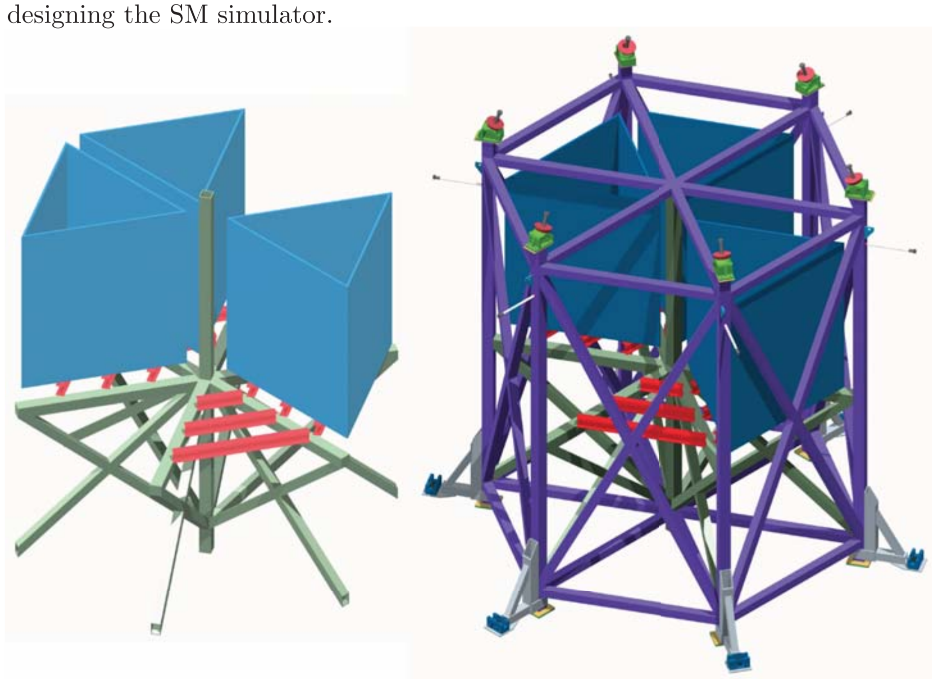

Figure 6. Creo renderings of the first simulator concept alone and installed inside the SM. Color Scheme: (Blue) Water Tanks (Red) Floor Support (Green) Structural Members, which connect inside the SM structure (Purple) SM Structure.

All concept development efforts ultimately led to a final design concept: Tanks and support structure installed inside the SM would constitute the SM Dry Ballast. Filling the tanks with water would create the SM Fueled Ballast. An additional weight, possibly a couple hundred steel tiles, inside CM2 would comprise the CM2 Dry Ballast.

2. Completed Design

Figure 7 shows an overview of the simulator design concept. The design consists of four aluminum water tanks, supported by a steel structure. Each tank rests on and is fixed to two steel square Hollow Structural Section (HSS) beams. These floor beams are attached to six radial beams (square HSS beams exactly the same size as the radial hexagonal pattern on the top and bottom of the SM), which efficiently transfer the load to square HSS diagonal beams mounted at the same joints. The diagonal beams react the load into the very bottom of the outer support beams of the SM and into one new central beam (part of the simulator), which finally transfer the load to ground. By transferring the loads through the very bottom of the SM, the simulator's impact on the structural integrity of the SM is minimized. The simulator (items 2, 3 and 4 in Figure 7) represents the SM Dry Ballast, while the water that will occupy the tanks constitute the SM Fueled Ballast. The tanks' support structure and the empty tanks combined neither fully accounted for the SM Dry Ballast weight, nor did it place the CG as high as originally desired. Consequently, additional steel ballast (item 3 in Figure 7), in the form of approximately forty 12"x12"x1" steel tiles were placed in tanks and supported near the top of the operable range of the SM to reach the desired weight and CG specifications of the SM Dry Ballast.

[e] Aluminum is a relatively light material while still exhibiting relatively high strength properties. It is used in designs when low weight requirements are desired.

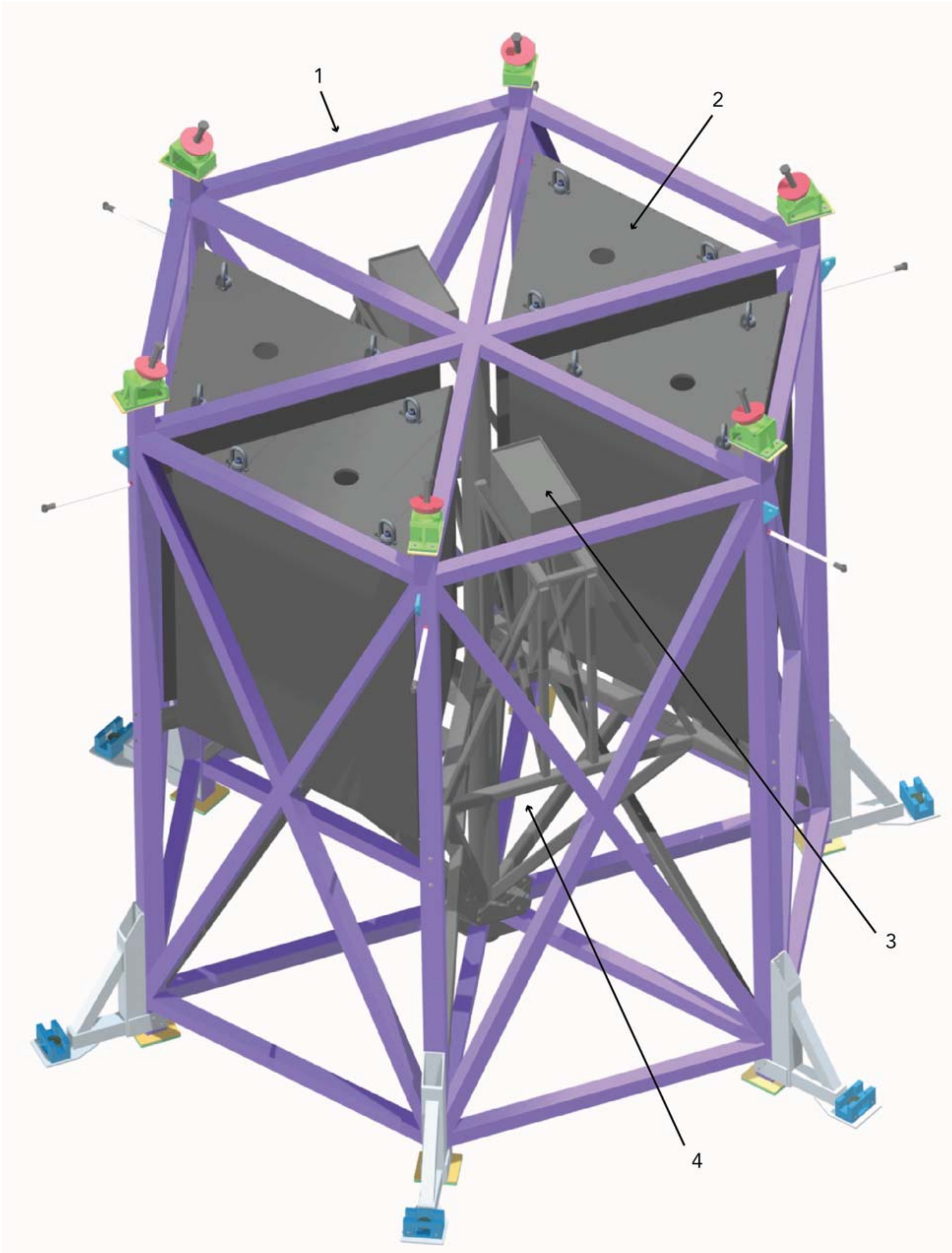

Figure 7. A Creo rendering of the Simulator installed inside the existing SM stand. Annotations: (1) Existing SM Stand (2) Water Tank (1 of 4) (3) Additional Steel Ballast (1 of 2) (4) Simulator Structure.

Figure 8 shows an overview of just the structural simulator elements inside of the SM. All elements are A500 structural steel square HSS, except for the cylindrical central column, which is round HSS. All of the elements are incrementally sized in 1 inch steps to allow for 0.35 inch (throat size) fillet welds[f] at each joint. The radials are 4"x4"x0.25", the tank supporting beams are both 3"x3"x3/8". The diagonal beams connecting the radials to bases of the outer and inner columns are 3"x3"x1/4" and the central cylindrical element is 7"x0.25". These structural steel shapes were chosen due their high strength, heavy weight, and ease of welding in order to create a robust structure that also serves as ballast weight. The pedestals that support the tile tanks have two 3"x3"x3/8" square HSS beams acting as the floor, while the remaining beams in the pedestal structure are 2"x2"x1/4" square HSS. Note the diagonal bracing in the pedestals that assist in resisting lateral transport loads applied to the steel tiles. The tanks that hold the tiles are 1/4 inch steel plate that are welded to the floor beams.

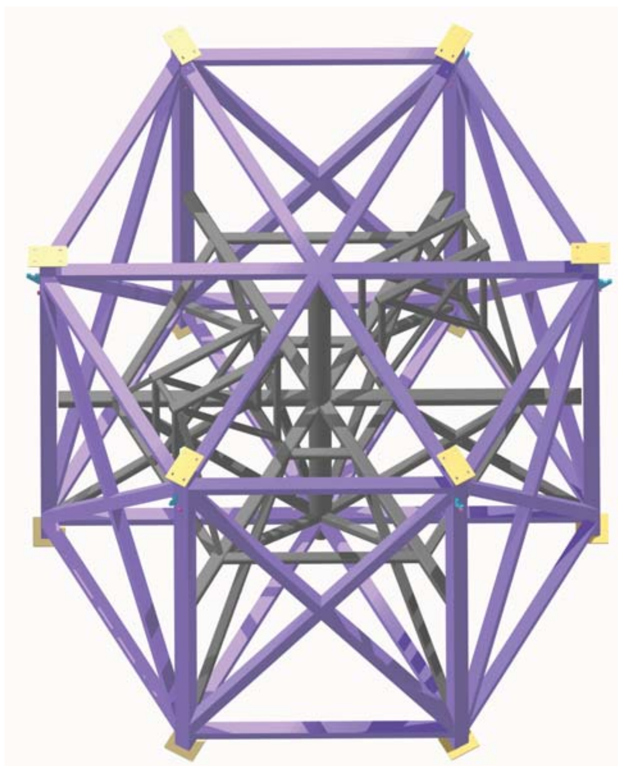

Figure 8. Creo rendering of the simulator structural elements (gray) inside the SM.

Figure 9 presents a top view of the simulator structure. Note that while the isometric view in Figure 8 looks extremely cluttered, the design is rendered to be very simple when viewed from another angle. The simulator radials lay directly in-line with the SM radials, as do the diagonal braces of the support structure. The beams that support the water tanks span the length of the triangular gaps and the pedestals are placed in two gaps while the water tanks will be placed on the remaining four floor sections.

[f]An offset of 0.5 inches on each side of the beam allow for this weld to be created.

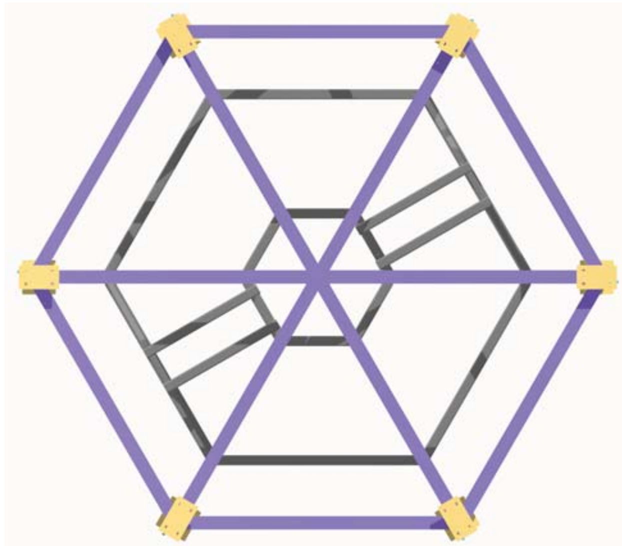

Figure 9. A top view Creo rendering of the simulator structural elements inside of the SM.

The four identical water tanks were designed with the purpose of placing approximately 20,000 lb_f of water 131 inches up from the short stack base. Figure 10 shows an isometric view of the water tank design (with and without the tank lid). The tank is constructed out of five 1-inch thick aluminum[g] 6061 plates. Three rectangular plates and a triangular base are welded together while the lid attaches to the walls through a series of 316 steel screws. The tank lid contains a centered 6 inch diameter hole to allow the tank to be filled with water. The tank should be filled with water up to two inches below the lid (75 inches of water) to achieve the appropriate weight and CG requirements. The lid also contains three hoist rings (McMaster Carr 3145T35) that will permit the tank (fully assembled but without water) to be lifted inside the SM. The dimensions of the tank OML create a 2 inch clearance on all three sides of the tank with the SM radial members.

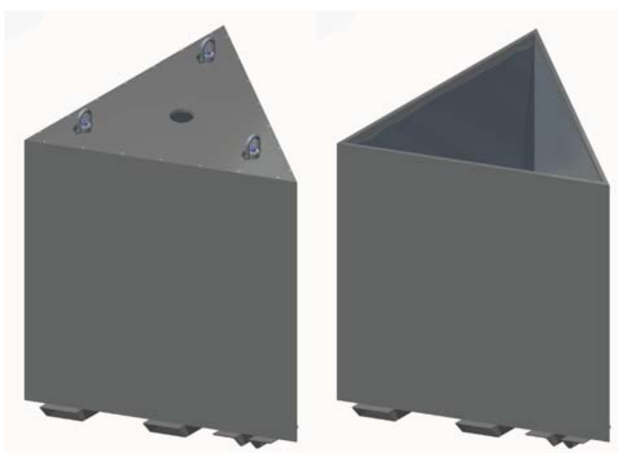

Figure 10. Creo renderings of the water tank sealed and with the water exposed.

The underside of the water tank shown in Figure 11 contains assembly, mounting, and draining provisions. The tank contains a standard ball valve (McMaster Carr 4786K210) to drain water from the tank. The tank also contains two angle brackets welded to the base. The aluminum angle brackets are placed three inches apart as to rest directly on opposite sides of one of the 3"x3"x3/8" tank support beams. Two 0.5 inch diameter 316 steel clevis pins connect the adapters beneath the tank support beam to secure the water

[g]Steel was considered as the tank material. Although steel is stronger, which would have permitted thinner tank walls, the weight of the tanks alone (without any support structure) would have surpassed the weight of the SM Dry Ballast.

tank in place and prevent tipping during transportation. Lastly, two 8"x4"x1/4" aluminum rectangular HSS tubes are welded to the bottom of the tank. The rectangular tubes are placed approximately 30 inches apart (centered about the CG of the water tank) and can be used to lift the tank with a forklift. Additionally, the rectangular tubing can support the empty tank when placed on the floor. These provisions significantly increase the ease of maneuvering the tank prior to installation. Note that the aluminum tubing and brackets each extend beyond the OML of the water tank. This extension is to let these elements serve as guides when placing the tanks inside the SM. The tubing will constrain the tank inside the triangular gaps (when being hoisted into the SM and when resting on the simulator support structure) while the taper will guide the tanks into place. Figures 11 and 12 show the rectangular tubing and brackets constraining a simulator water tank.

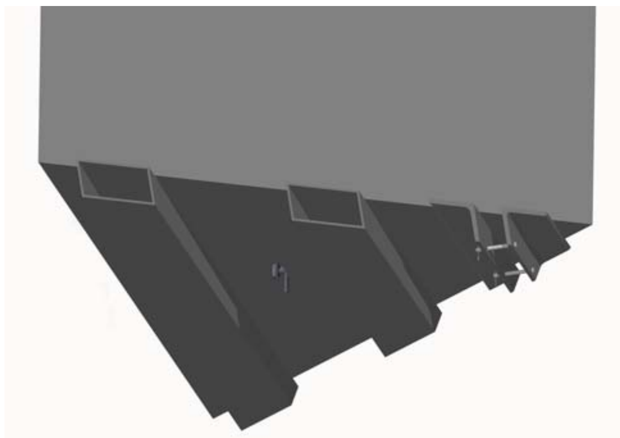

Figure 11. A Creo rendering of the underside of the water tank with exposed forklift, guidance, draining and mounting provisions.

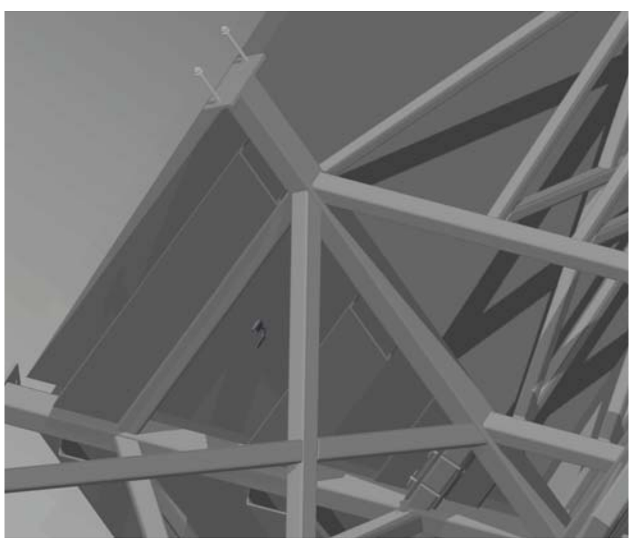

Figure 12. A Creo rendering of the water tank mounted to the simulator structure.

One final simulator feature worth noting is the SM-to-Simulator mounts. The simulator and SM are constructed out of two different metals, aluminum and steel. Although they are both common structural metals, they cannot be welded together. Consequently, the simulator is mated to the SM using a series of fasteners. One example is shown in Figure 13. The steel beams that constitute the simulator radials are cut 0.5 inches short of making contact with the SM in order to weld on steel plates. These plates contain holes that permit bolts to be placed through the plates and through the adjacent SM beam. Using at least two bolts allow the joint to provide moment resistance. Similar mounting provisions are in place where the round HSS connects to the bottom SM radials and where the diagonals connect to the bottom corners of the SM.

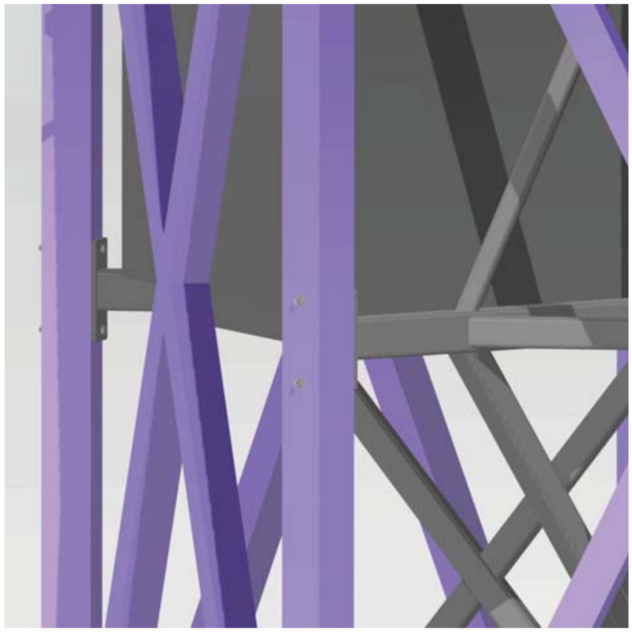

Figure 13. A Creo rendering of two plate and fastener sets that secure the simulator to the SM.

At this time the portion of the simulator residing inside CM2 is not complete. However, the weight and CG specifications of the SM Dry Ballast and SM Fueled Ballast have been determined, which dictates the CM2 Dry Ballast mass properties as shown in Table 4. The CM2 portion of the simulator will likely reflect a similar design to the tile tanks located in the SM portion of the simulator. The remaining weight can be created by using 318 steel tiles of 12”x12”x1” dimensions. Divided across 12 floor beams, that is 26 tiles per beam, or two substacks of 13 tiles per beam (nearly identical to the stacks in the SM). As such, similar tanks filled with tiles will likely be fixed to a structural pedestal that lifts the tiles approximately two feet off of the CM2 floor. The structure will require provisions that fix it to the existing CM2 structure. A similar angle bracket and pin combination may be used to secure the tanks and support structure to the vertical I-beams inside CM2.

Table 4. Final Weight and CG of Simulator Components.

Component	Weight, lb_f	X_{CG}, in
CM2 Dry Ballast	12842.47	243.773
SM Dry Ballast	13309.53	126.015
SM Fueled Ballast	20798	131.553

B. VMS Hoist Structure for OSMU Testing

1. Design Development

Before creating a design for the structure, a pulley and hoist were chosen. The pulley was chosen to be the VB 3500 from Jeamar Winches and the hoist was chosen to be the HD1200 from Columbia Winch & Hoist with a 1/4 inch wire rope. The pulley did not advertise a built in FS and thus the model was chosen based on its working load limit of 3500 lb_f, which for a 500 lb_f applied load meets the required FS of 5:1 for lifting equipment. The hoist was built to ANSI B30.7, which specified a FS of at least 3.5. Therefore, a model with a working limit load of 1200 lb_f meets the required FS of 5:1 for lifting equipment.

2. *Completed Design*

The final design, shown in Figure 14 features a tripod design that places the pulley at a nominal position 49 inches out from its connection point to the VMS. The structure is composed of three main beams; a 3"x2.5" wide flange beam and two 2"x2"x1/8" angle beams. The wide flange is centered on the VMS and the two angle beams extend symmetrically outward and downward to make contact with the outer frame of the VMS ladder. Figure 15 shows the structure mounted to the VMS. The angle beams extend down to the last available bolt pattern on the VMS to increase the efficiency of the beams to resist the vertical load applied to the pulley. The tripod design was chosen to efficiently use material in an effort to minimize the weight of the structure, which is also why the entire structure is aluminum 6061-T6 (sans fasteners). The wide flange beam was selected because it exhibited the high moment of inertia necessary to resist cantilever bending. The angle beams were chosen for being simple to attach to the wide flange while also containing a a relatively high moment of inertia necessary to resist buckling.

The three beams are connected via the Wide Flange-to-Angle Beam Adapter. This adapter bolts into the end of the wide flange and the ends of the angle beams to join the three at compound angles. Due to the compound angles of the flanges, the adapter will be machined on a 5-axis CNC mill out of a single block of aluminum. Large rounds are visible on the adapter and were left to both minimize stresses and machining time. The angle beams mounts to the VMS with similar adapters. Once again the flange attaches to the base plate at a compound angle so the 5-axis mill will be required to machine the two Angle Beam-to-VMS Brackets. The base of the adapter mimics the bolt pattern on the VMS ladder where it will be attached. The Wide Flange-to-VMS Brackets will not need as complex of machining jobs. The bracket is a standard 5"x5"x3/8" angle bracket that will mount the wide flange beam to both sides of the VMS ladder.

The Wide Flange-to-Pulley adapters at the front of the structure are bent pieces of aluminum plate with a series of holes, which allow the pulley to be shifted about its nominal position a total of 6 inches to account for some uncertainty in the desired plate position, weight, and CG properties. A 3/8" thick plate adapts the hoist to the wide flange at the VMS end of the structure. Four 1/4" thick diagonal aluminum plates are welded to the wide flange to support the hoist during normal use and during VMS deceleration. Finally, the design features all 316 stainless steel fasteners, except for the fasteners connecting the hoist plate to the wide flange where extra strength is needed and grade 8 steel fasteners will be used.

With this design, the OSMU plate can be lifted to it desired position in a matter of minutes. The hoist line can be reeled in and VMS actuated to perform its required OSMU tests, with all components of the hoist structure remaining attached[h]. Therefore, the testing efficiency is maximized from the standpoint of the hoist structure design. Hand calculations, FEA results, and additional renderings are located in the *Appendix*.

[h]The hoist could have been removed between tests if the structure could not withstand its deceleration loads, but testing would be much more efficient if the hoist could remain attached.

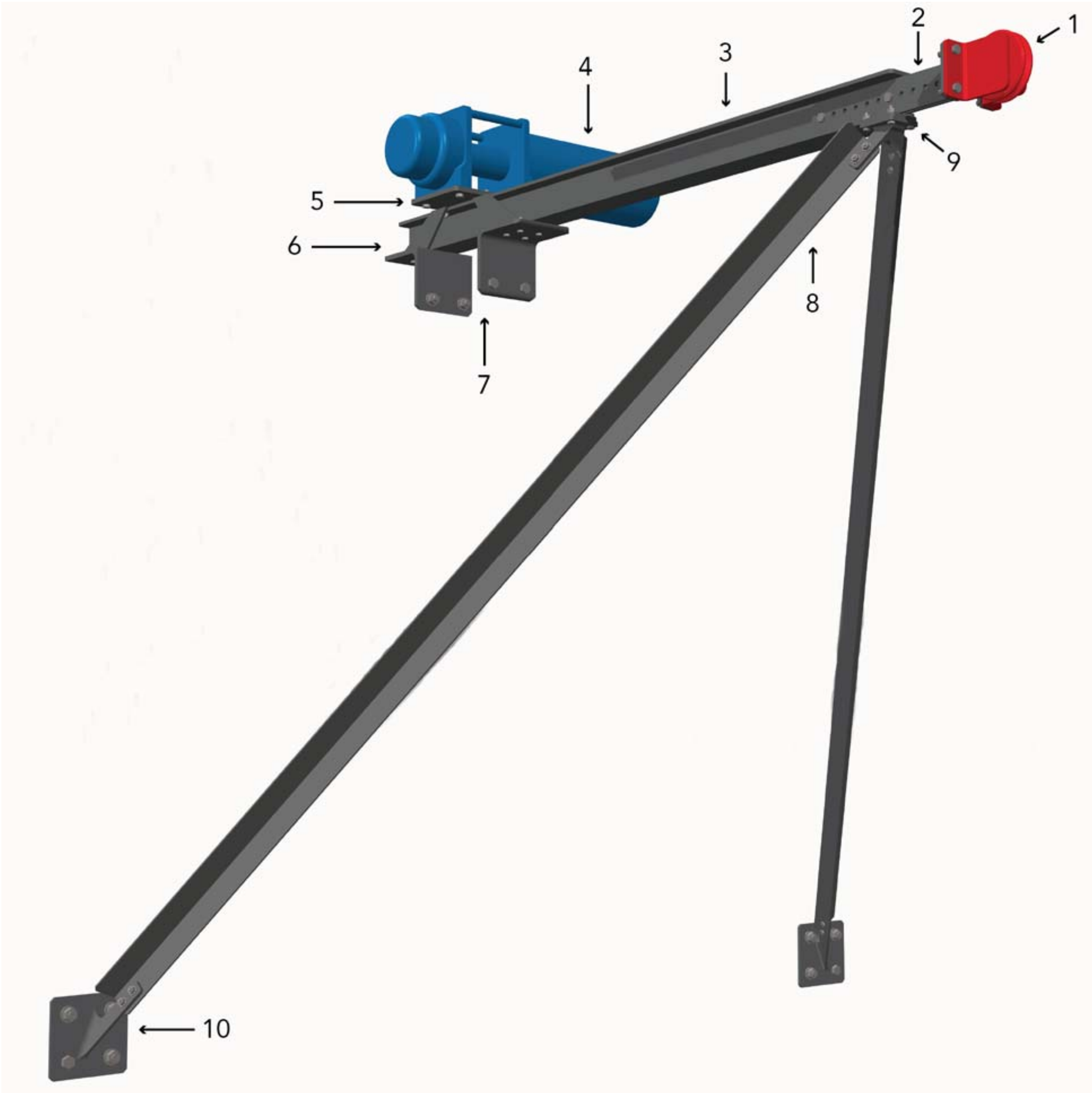

Figure 14. A Creo rendering of the designed structure. Annotations: (1) Pulley (2) Wide Flange-to-Pulley Adapter (3) Wide Flange Beam (4) Hoist (5) Hoist Mounting Plate (6) Wide Flange Strengthening Plates (7) Wide Flange-to-VMS Brackets (8) Angle Beam (9) Angle Beams-to-Wide Flange Beam Adapter (10) Angle Beam-to-VMS Adapter.

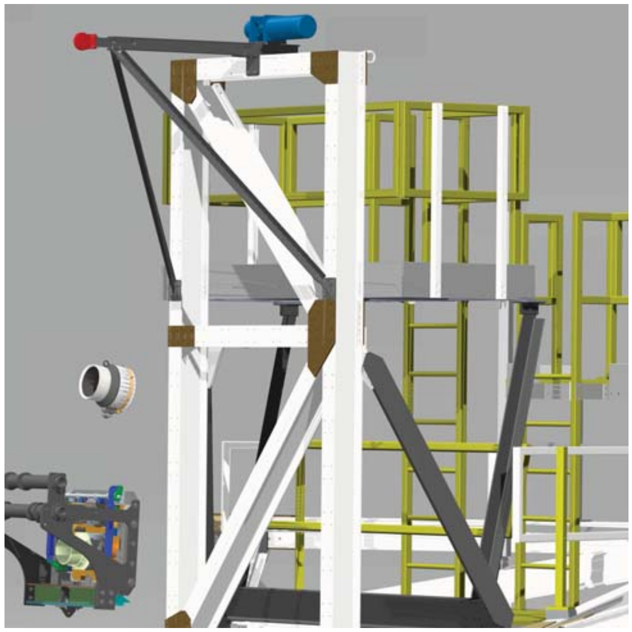

Figure 15. A Creo rendering of the designed structure attached to the VMS with the OSMU plate below.

IV. Conclusion

In conclusion, an Orion Crew Module/Service Module Structural Weight and Center of Gravity Simulator and a Vehicle Motion Simulator Hoist Structure for Orion Service Module Umbilical Testing were designed during my summer 2014 internship in KSCs Structures and Mechanisms Design Branch (NE-M2). The simulator uses structural steel ballast and water tanks to simulate the dry and fueled configurations of the EM-1 flight vehicle; filling and draining the four water tanks transitions the simulator between the two states. The simulator will be installed inside of the SM. The elements of the steel support structure will all be welded to each other and connected to the aluminum SM via a series of fasteners. The hoist structure is an aluminum cantilevered tripod structure that mounts to the VMS and uses a hoist to pull the OSMU plate into position to be attached to the VMS. The hoist structure is also designed to withstand the acceleration loads of the VMS. The simulator and the hoist structure were both designed to the required factors of safety for Ground Support Equipment. These two designs provide KSC engineers mature concepts worthy of fabricating and utilizing to advance NASAs preparations for launching Orion in 2017.

Appendix

A. Orion CM/SM Structural Weight and CG Simulator

1. *Hand Calculations*

Hundreds of hand calculations were used to verify that the simulator met the required FS. Some of these were literally done by hand while others were performed parametrically in MathCAD and/or Microsoft Excel. This section presents a few sample calculations to demonstrate the general process.

Eq. (11) was paramount to the design of the simulator and was used to determine the location of the three ballasts. A sample calculation to locate the SM Fueled Ballast is shown below.

$$\bar{x}_{SMFueledBallast} = \frac{\sum_{i=1}^{n} m_i x_i}{\sum_{i=1}^{n} m_i}$$

$$= \frac{m_{fueled} x_{fueled} - m_{dry} x_{dry}}{m_{fueled} - m_{dry}}$$

$$= \frac{(60210 lb_f)(167.88 in) - (39412 lb_f)(187.05 in)}{60210 lb_f - 39412 lb_f}$$

$$= 131.533 in$$

In the design of the simulator every member had to be analyzed to ensure they could withstand the imparted loads. An example is provided in the calculation below, which determines the normal stress due to bending in the longer beam directly supporting the water tank.

$$\sigma_{bend} = \frac{|M_{max}|}{S}$$

$$= \frac{PL}{12 S_{3"x3"x3/8"}}$$

$$= \frac{(4761 lbf)(63.59 in)}{12(2.3 in^3)}$$

$$= 1.0967e4 psi$$

The bending stress is then compared with the material strength to verify that it complies with the required FS. For A500 steel, ultimate stress is critical (meeting the FS on ultimate will inherently meet the FS on yield) and the required FS is 3:1.

$$FS = \frac{\sigma}{\sigma_{allow}}$$

$$= \frac{\sigma_{ult}}{\sigma_{bend}}$$

$$= \frac{58000 psi}{1.0967e4 psi}$$

$$= 5.28$$

This member exceeds the required 3:1 FS and therefore can sufficiently withstand the bending stress that will be imparted on it during use.

2. Finite Element Analysis

Creo Simulate was used to run FEA on two components for this design project. The first was run on the LAS Attach Fitting to investigate its response to an increase in the lifting load and the second was run on the simulator water tank to determine wall thickness.

FEA was performed on the LAS Attach Fitting using Creo Simulate to verify that it could withstand the new maximum loading condition. Figure 16 shows the results of this analysis. A lifting load of 8800 lb_f (the maximum lifting load contribution at one LAS Attach Fitting) was applied over the bushing area that made contact with the underside of the top hole. Fixed displacement constraints were applied over the washer area on the four attachment holes at the bottom of the fitting. The FEA results show that the fitting adheres to the required FS of 3:1 on ultimate[i].

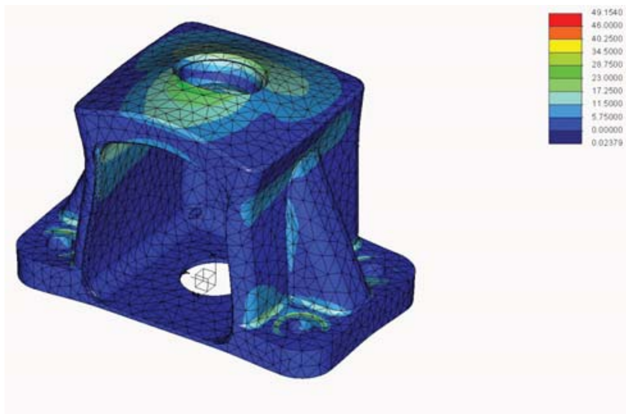

Figure 16. A Von Mises Stresses, ksi, fringe plot of the LAS Attach Fitting scaled such that the color red denotes an area of the material violating the required factor of safety.

Figure 17 presents the FEA results of a loaded simulator water tank and was used to size the wall thickness of the tank. The Finite Element Model (FEM) included the triangular Aluminum 6061-T6 tank with a uniform load of 2.8 psi applied to all interior surfaces. Although hydrostatic loads exhibit a linearly increasing pressure distribution as depth increases, uniform loads are significantly simpler to apply in Creo Simulate and the results led to a conservative design since the upper portion of the structure actually sees significantly less load. Consequently, a uniform load of the maximum hydrostatic pressure was used and the tank was fixed at its base. The FEA results show that a 1 inch wall thickness keeps the structure within the allowable stresses for welded aluminum (7.5 ksi). Therefore, aluminum plates can individually be cut and then welded together to create the tank.

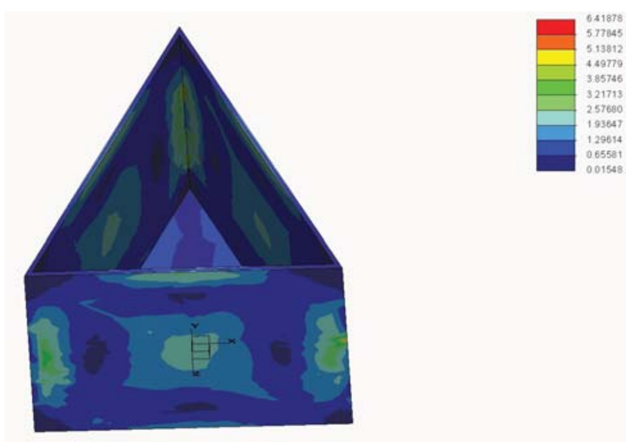

Figure 17. A Von Mises Stresses, ksi, fringe plot of the water tank scaled such that the color red denotes an area of the material violating the required factor of safety.

[i]Tiny hot spots of red are located inside the zero-displacement washer area (they may not be visible in this relatively low resolution graphic). However, these hot spots are merely a result of the FEA boundary conditions and would not occur in the physical material.

3. Additional Renders

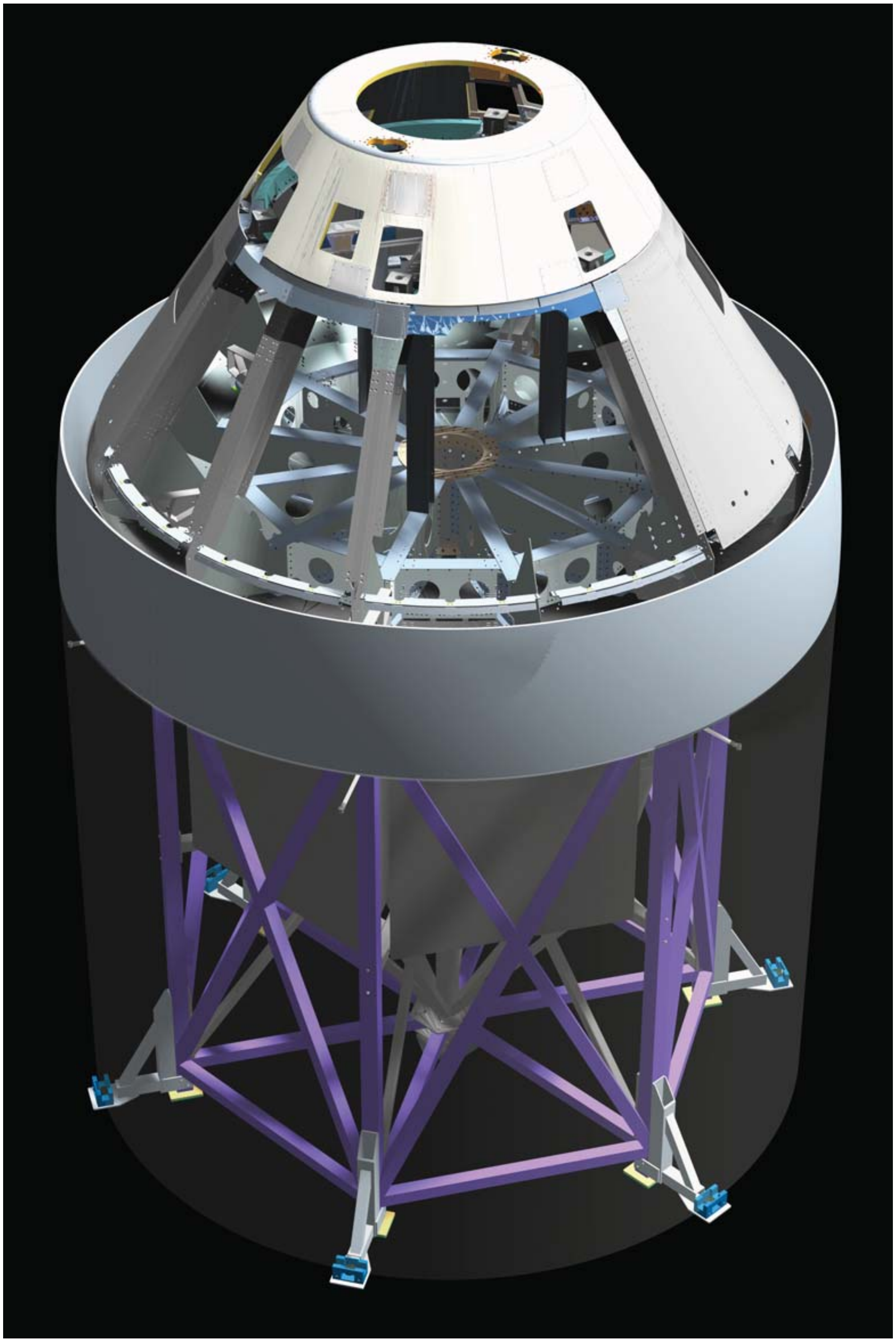

Figure 18. A Creo rendering of the simulator, SM and CM2.

B. VMS Hoist Structure for OSMU Testing

1. *Hand Calculations*

Similar to the simulator, many hand calculations were performed to verify that the hoist structure met the required FS. A sample calculation used to size one of the members is provided below.

The angle beams were chosen to resist buckling as defined by Eq. (7). A pin-pin connection assumption was made to remain conservative in the calculation. The calculation below solves for the minimum required I based on the required FS.

$$P_{critical} > 3P$$

$$\frac{\pi^2 EI}{L_e^2} > 3P$$

$$I > \frac{3PL_e^2}{\pi^2 E}$$

$$I > \frac{3(364.119lb_f)(\sqrt{(55in)^2 + (33in)^2 + (48in)^2})^2}{\pi^2(10.9e6psi)}$$

$$I > 0.065in^4$$

2. *Finite Element Analysis*

FEA methods were used to design multiple components of the hoist structure where hand calculations were deemed too complex and/or inaccurate. Specifically, Creo Simulate was used to design the Wide Flange-to-Angle Beam Adapter, the Angle Beam-to-VMS Bracket, the Wide Flange-to-VMS Bracket and to design the Wide Flange Strengthening Plates. While the figures below show final designs, it should be noted that many iterations of each component were modeled and analyzed until the FEA results showed acceptable stresses.

To design the Wide Flange-to-Angle Beam Adpater, Eq. 1 and Eq 2 were applied to a free body diagram of the whole structure and joints to solve for the load through the beam. The result yielded a 364.119 lb_f axial compression load through each angle beam. Creo Simulate was used to apply these loads through each flange in the form of bearing stresses and the model was fixed at its base. Figure 19 shows these loads and constraints, as well as the FEA results in the form of a Von Mises Stress plot. The FEA of the final design shows that the stresses fall within the allowable stress of Aluminum 6061-T6 (14 ksi). The holes on the base and on the flanges were analyzed using hand calculations to ensure that they met the FS for bearing stresses.

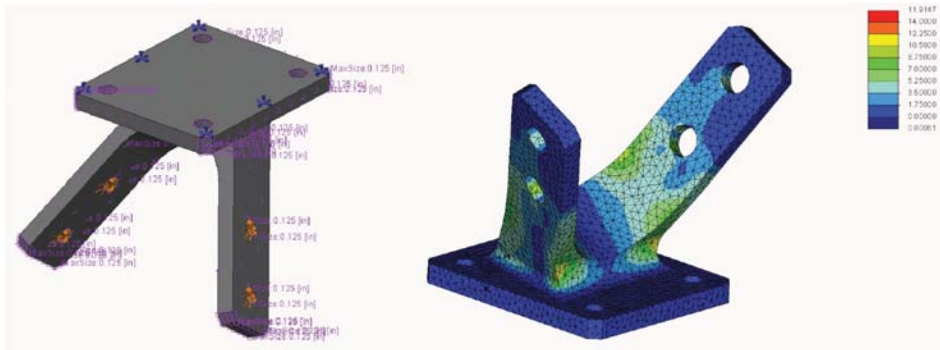

Figure 19. FEM (left) and FEA results (right) of the Angle Beams-to-Wide Flange Adapter. The Von Mises Stresses, ksi, fringe plot is scaled such that the color red denotes an area of the material violating the required factor of safety.

Similar to the Wide Flange-to-Angle Beam Adapter, the Angle-to-VMS bracket sees an axial compression of 364.119 lb_f. The load was applied in the form of bearing stress and the bracket was fixed at the bottom, as shown in Figure 20. Figure 20 also shows the FEA results. The Von Mises Stesses fringe plot of the final design shows the maximum stress at approximately 11 ksi, which is within the allowable stress range. Bearing stresses were verified with hand calculations after the completing FEA.

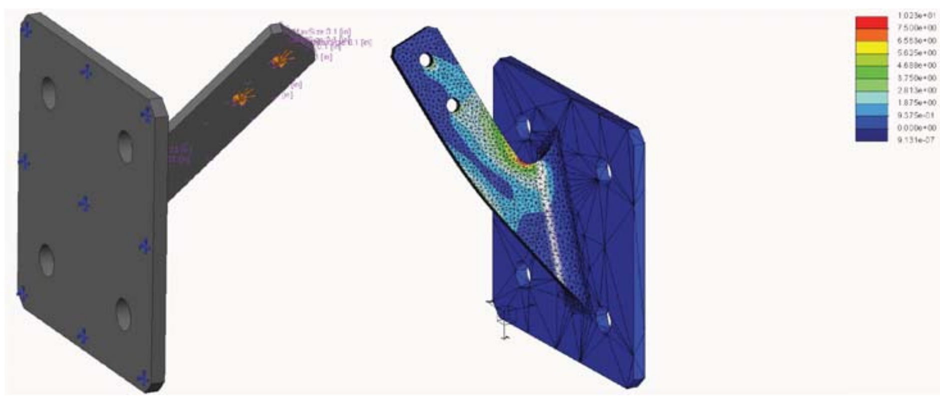

Figure 20. FEM (left) and FEA results (right) of the Angle Beam-to-VMS Bracket.

To design the Wide Flange-to-VMS Bracket, the loads imparted by the 5g deceleration were used because they were larger than the loads imparted on the bracket during normal use (hoisting the OSMU plate). Specifically, placing the hoist directly above (but with an off-center CG) resulted in a large force-couple imparted on the horizontal flanges of the bracket during deceleration. Figure 21 shows the loads and constraints of the FEM model. The force-couple was applied in the form of an upwards-acting 1368 lb_f and a downwards acting 1068 lb_f acting on the washer area of the top pairs of fastener holes. An additional 472 lb_f was applied as bearing stresses at the same holes to model the reaction force needed at this point to secure the rest of the hoist structure from ripping off of the VMS during deceleration. Finally, the bracket was fixed at the two holes in the vertical plane to mimic the attachment to the VMS. Although the FEA results in Figure 21 show an area of red near two fastener locations, this stress concentration is ignored since it is merely a results of the zero-displacement constraints and will not occur in the the physical material. Consequently, the bracket can successfully remain within the required factors of safety given the imparted loads.

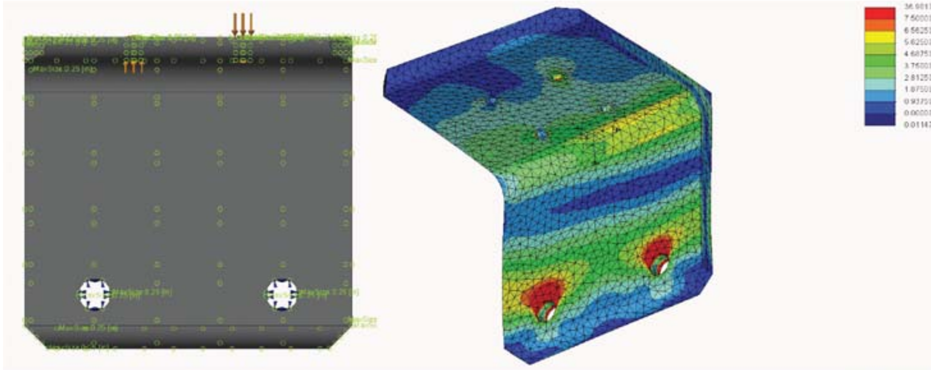

Figure 21. FEM (left) and FEA results (right) of the Wide Flange Beam-to-VMS Bracket. The Von Mises Stresses, ksi, fringe plot is scaled such that the color red denotes an area of the material violating the required factor of safety.

FEA methods were used to appropriately strengthen the wide flange beneath the hoist to resist the force-couple from the hoist during the deceleration of the VMS. Without strengthening and the force-couple applied to the wide flange, hand calculations show that the wide flange fails. Consequently, FEA was used to *size* plates that will be welded onto the wide flange beam beneath the hoist to strengthen the structure. The FEM in Figure 22 shows the application of the force-couple and the wide flange fixed at the fasteners that connect it to the angle brackets. Once again, unrealistic stress hot spots occur at the holes where loads and constraints are applied so they are ignored. Therefore, the FEA results in Figure 22 prove that welding on four diagonal plates (providing a direct path from the point of load application to *ground* and creating a resistance to torsion) successfully resist the imparted loads by keeping stress below 7.5 ksi, the maximum allowable stress for welded Aluminum 6061-T6.

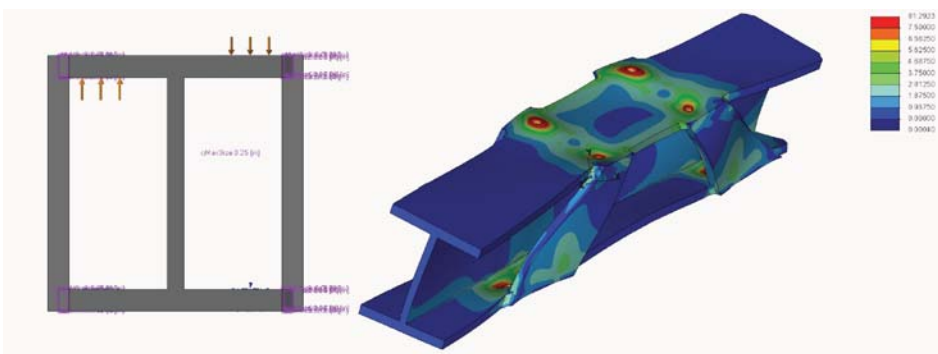

Figure 22. FEM (left) and FEA results (right) of the Wide Flange Strengthening Plates. The Von Mises Stresses, ksi, fringe plot is scaled such that the color red denotes an area of the material violating the required factor of safety.

Compared to the other components that required FEA analysis, the design of the strengthening plates required significantly more FEA models to obtain a successful design. Figure 23 shows two other options that were thoroughly tested, yet rejected. The right shows plates inserted into the beam that make contact with the web and both flanges. However, even at very short distances to the fastener holes, the imparted moment on the material is large enough to cause failure. The design on the left shows long plates spanning the outside of the beam (similar to the previously discussed diagonals). However, the fringe plot shows that there is very little stress throughout most of the plate, meaning that most of the plate does not lie in the load path and therefore is an inefficient design.

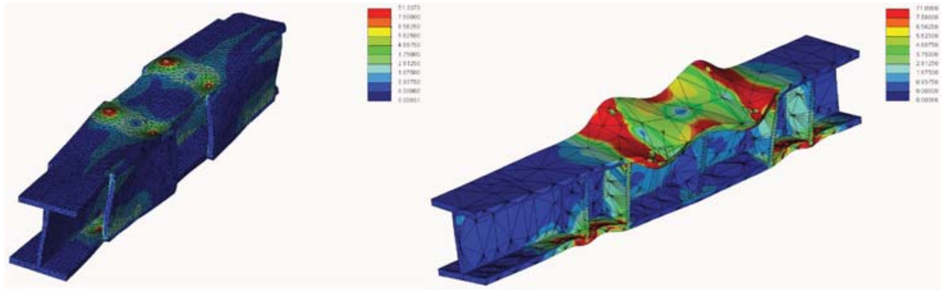

Figure 23. FEA results of initial designs for the Wide Flange Strengthening Plates. The Von Mises Stresses, ksi, fringe plot is scaled such that the color red denotes an area of the material violating the required factor of safety.

3. Additional Renders

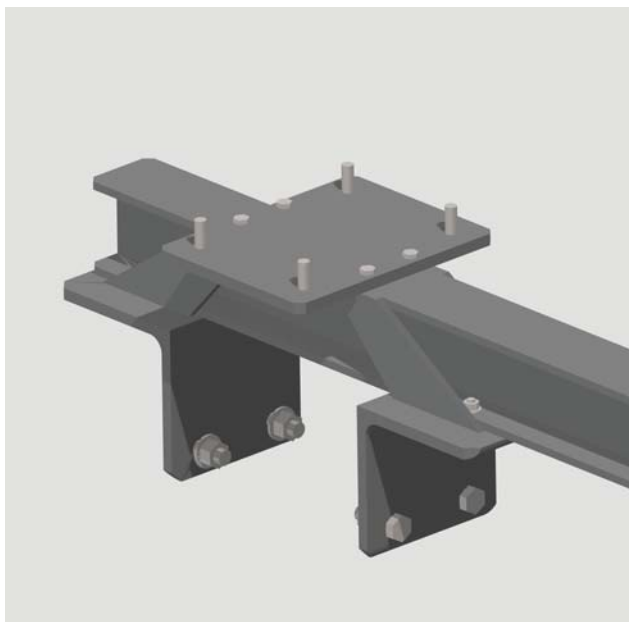

Figure 24. A Creo Rendering of the hoist plate, wide flange, and strengthening plates

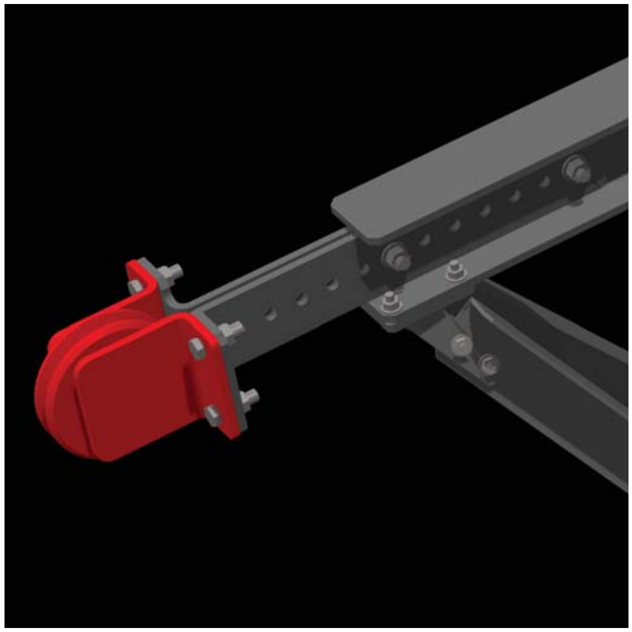

Figure 25. A Creo rendering of the pulley, pulley adapter, and wide flange-to-angle beam adapter

Acknowledgments

First and foremost, I would like to thank Michael H. Haddock, Joseph Porta, and NE-M2 for providing me with the opportunity to intern as a mechanical design engineer at NASA KSC. I am particularly grateful for the mentor-ship and guidance of Michael Haddock throughout the design of the Orion CM/SM Structural Weight and CG Simulator. I also wish to thank Patrick Maloney for providing me with and advising me on the second design project, the VMS Hoist Structure for OSMU Testing.

Many other NASA engineers also deserve to be acknowledged for their assistance with the two design projects. Specifically, I would like to thank Steven Larsen, Kurt Smith, Martin Grashik, Kevin Ricksecker, William Manley, and Nayza Caban of NE-M2 for offering best-practice design advice and assistance with various Creo tools. I would also like to thank David Chesnutt and José Mayí-Rivas of NE-M1 for providing advice on interpreting FEA results, and Chief Engineer Dr. Eric Thaxton for his help with structure optimization.

Lastly, I am very grateful for the cooperation of NASA Langely, specifically Robert Parker, for his assistance in acquiring CAD models and reports pertaining to the CM2 Pathfinder mock-up, which were required to successfully design the simulator.

References

[1]Maliska, H. A., "Flight Test Article PA-1 Geometry & Mass Properties Requirements Document," NASA FTO-AFT-FTA-003-PA1, 2009.

[2]Manley, W. C., "CM2 Short Stack Mockup Handling," NASA, 2012.

[3]Atherley, R.,"PA-1 CM Pathfinder Lifting, Handling and Transportation Structural Analysis Report," NASA FTA-0222, 2009.

[4]Dow, B. C., "LaRC Pad Abort 1 Primary Longeron Joint Analysis Report," NASA FTA-0209, 2009.

[5]Mayí-Rivas, J. J., "CM-SM Stand Mockup Design," NASA, 2012.