Design Analyses

- **Chamber Implosion Risk: ANALYSIS**
    - **(i)** We ran FEA on the chamber to validate that the material and geometry can withstand a vacuum in order to reduce the risk of it imploding
    - **(ii)** We assumed that the pressure outside the chamber was atmospheric pressure, and inside was a perfect vacuum
    - **(iii)**
        - The simulation was performed by joining the chamber and the lid to create a sealed vessel.
        - The chamber lid was fixed as this is consistent with real operating conditions.
        - An external pressure of 14.7 PSI (1 atm) was applied to the outer surface of the chamber, and zero pressure was applied internally to simulate the pressure differential experienced from pulling a vacuum. We also decided to analyze if the internal pressure was at 150 PSI, which could be a scenario if the target bursts.
    - **(iv) Results**

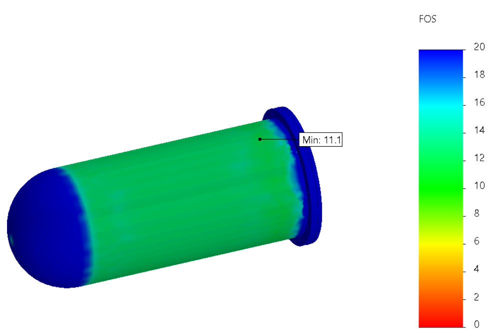

Figure 18: Static Analysis (Vacuum)

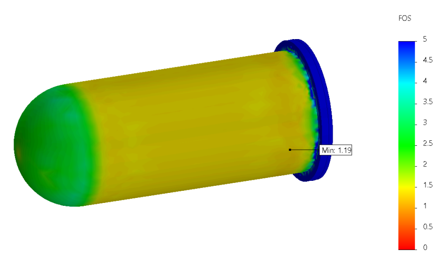

Figure 19: Static Analysis (150 PSI)

        - As seen in **Figures 18 & 19,** the factor of safety for vacuum and 150 PSI pressure are shown.
            - **Minimum Factor of Safety (FOS) of 11.1 for a vacuum**
            - **Minimum Factor of Safety of 1.19 at 150 PSI**

    - **(v) Result Interpretation**
        - A factor of safety of 11.1 indicates the design is more than robust enough to handle a vacuum.
        - In the event of a component failure resulting in chamber pressurization, a factor of safety of 1.19 indicates the chamber could safely withstand this

pressure of 150 PSI, at which point pressure relief valves would cause a system wide vent

    - These results can be used to justify vessel thickness and geometry, but could also be a justification for adjustments to be made for additional safety in the case of failure, as well as design optimization for a more compact and efficient design

  - **(vi) Analysis Limitations**
    - Assumptions were made for the joining of the two components to simulate a sealed vessel, as they would be fastened together in real operational conditions and not bonded as was done in the simulation
      - This likely has an impact on the simulations accuracy for vacuum and pressurized conditions

- **Equipment Weight Risk: ANALYSIS**
  - **(i)** We did an FEA analysis that looked at the stresses on the frame caused by the heaviest pieces of equipment.
  - **(ii)** We assumed that the leak detector, vacuum pump, and piping had the largest effect on the stresses of the frame, as they are the heaviest equipment
  - **(iii)**
    - Static Analysis
      - The bottom surface of the frame was fixed, as this mimics the bolting of the frame to the ground
      - A 250 newton load was placed on the second layer of the right side, 400 newtons on the first layer of the right side, and 100 newtons on the left side to simulate the anticipated loads of the leak detector, vacuum, and piping respectively
    - Vibrational Analysis

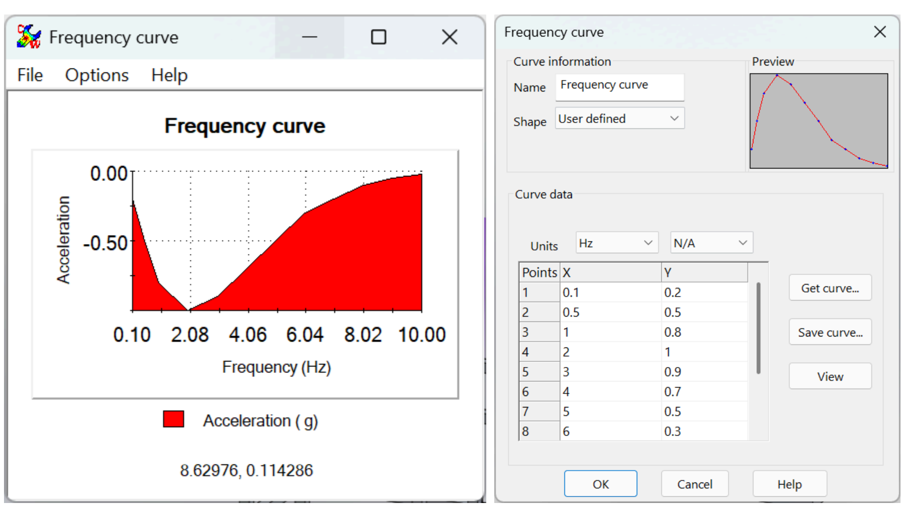

Figure 20: Frame Vibrational Analysis Setup (Base Excitation Frequency Curve)

- The bottom surface of the frame was fixed, as this mimics the bolting of the frame to the ground
- A uniform base excitation of 1g was placed on the frame with a frequency variation that was taken from the PSD (power spectral density) curve seen in **Figure 20**.

○ **(iv) Results**

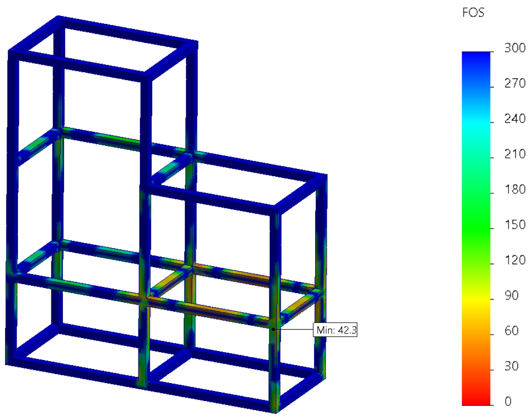

Figure 21: Frame Static Analysis Results

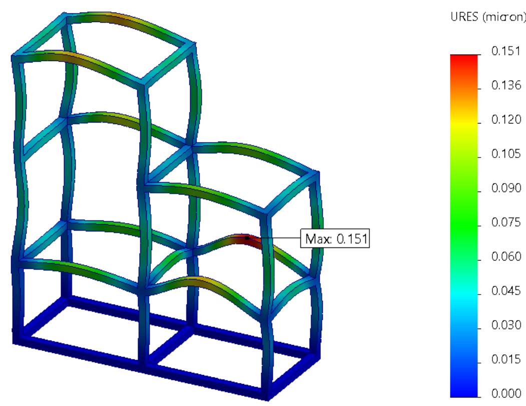

Figure 22: Frame Vibrational Analysis Results

- ■ **Static Analysis**
  - **Minimum Factor of Safety of 42.3**
- ■ **Vibrational Analysis**
  - **Maximum Deformation of 0.151 microns**

- ■ As seen in **Figure 21** , the factor of safety is sufficiently high on the frame indicating that it won't deform due to the leak detector, vacuum pump, or piping weight
- ■ The maximum deformation is minimal, as seen in **Figure 22**, indicating the robust design is more than sufficient enough for anticipated vibrational loads

○ **(v) Result interpretation**

- ■ **Static Analysis**
  - A factor of safety of 42.3 indicated the 80/20 Aluminum frame is more than robust enough for current loads from components
  - With such a high FOS, there is the possibility for the addition of more equipment and components
- ■ **Vibrational Analysis**

        - A maximum deformation of 0.151 microns in the event of vibrational loads within the expected frequency range indicates the frame is rigid enough for seismic activity.
        - Such small deformation also shows that higher frequency vibrations would not result in catastrophic structural failure.
  - **(vi) Analysis Limitation**
    - The model was simplified by creating a single rigid structure made of 6063 aluminum, as opposed to multiple aluminum extrusions joined with fasteners and brackets.
        - This was done due to computational limitations as well as lack of time and experience with more complex models.
        - Therefore the simulations do not fully explore the effects of the static and vibrational loads on the joints, fasteners, and other joining components, which is necessary to take into consideration when considering the operation in a production environment
        - This had a larger impact on the vibrational analysis as the vibrational loads are more impactful for the smaller joining components
        - Given more time, a higher fidelity model with more components could be developed to more accurately simulate the effects on vibrations and static loads.