Vibration Isolation Design Guidelines

The following document is a guideline for FEA, MATLAB, and Simscape use for projects in the field of vibration isolation.

1. Introduction

The purpose of this document is to aid in the design process of Maxar's Vibration Isolation System by figuring out design targets from sponsor requirements. It will go over various analysis techniques and tools that we have developed that we believe are helpful. It will also highlight some issues that we have encountered.

2. Design

2.1 Requirements

Based on the project requirements, the most important ones to consider that will primarily drive the design would be the frequency and damping. Secondary requirements that will also need to be considered is quasi-static loading and the damping material properties. Other requirements must be addressed, but they are not as critical, thus can be considered later.

2.1.1 Frequency

- Means the first mode of the overall system (flexure plate/payload mass included)
- The first mode should be the vibrational mode of interest (Z-vibration, not XY).
    - Factors to consider: Stiffness and mass
        - Higher stiffness = higher natural frequency
        - Higher mass = lower natural frequency

2.1.2 Damping

- Interpreted as transmissibility (acceleration of interest/input acceleration)
    - Lower transmissibility = more damping
        - Orbit condition will need more damping than launch even though the base acceleration for launch is higher
    - What affects transmissibility
        - Damping ratio/damping coefficient
- Things to consider to produce the highest damping
    - Strain energy

2.1.3 Quasi-Static Loading

- Main structural requirement
- Simplified assumption of all dynamic and static loads combined as an acceleration applied at the center of mass
    - Can be applied at any axis (design for worse case axis)

2.1.4 Viscoelastic Material

- Main damping material used
- Viscoelastic - Energy is partially recovered during loading
- Material Loss Factor - Ratio of energy dissipated per cycle to maximum strain energy stored
- Modulus - The stiffness of the material
    - Shear (G) or Elastic (E) modulus
- Material loss factor and modulus are both affected by greatly affected by temperature and frequency
    - Design for worst case conditions within requirement range.

3. Analysis

3.1 Excel, MATLAB & Simulink

Excel is a powerful and simple tool to begin preliminary calculations. Additionally, the greatest strength to Excel is the ability to iterate and solve for optimized values quickly. MATLAB scripts and Simulink block diagrams allow engineers to run in-depth calculations and graphically process the results, similarly to an Excel spreadsheet. The main advantage of Simulink are the numerous tool boxes at one's disposal. For this project, we used the digital signal processing toolbox to generate a customizable sine sweep (chirp) input and the SimScape toolbox to model the pedestal and accompanying components as a system of masses, springs, and dampers within the Simulink environment.

3.1.1 Excel Spreadsheets

Transmissibility and Damping Ratio Calculator

The first spreadsheet uses user input values for the damping coefficient of the pedestal, equivalent stiffness of the pedestal (calculated from simple hand calculations), and the number of "linkages" present in the design. This allows the spreadsheet to calculate the damping ratio of the system, consequently calculating the overall transmissibility of the system over a frequency range. The file contains three different sheets, one for launch conditions, one for orbit conditions, and finally a sheet that calculates the transmissibility of the 2018-2019 senior project pedestal which was verified on Maxar's shake table.

Input Acceleration	8	g
Linkage Damping Coeffiecient	7.025	kg/s [retrieved from MATLAB script]
	0.481362	slug/s
Pedestal Mass	1	lb
	0.453721	kg
	0.031081	slug
Pedestal Stiffness	2.60E+05	N/m
	1.48E+03	lbf/in
Pedestal Natural Frequency	756.9941	rad/s
	120.4793	Hz

Number of links	6	
Pedestal Damping Coefficient	42.15	kg/s
	2.888	slug/s
System Damping Coefficient	126.45	kg/s
	8.665	slug/s
System Stiffness	8.94E+05	N/m [launch]
	5.10E+03	lbf/in
System Natural Frequency	456.87	rad/s
	72.71	Hz
System Mass	4.282	kg
	0.293	slug
Damping Ratio	0.032	

Figure 1: Table to write in acceleration input, damping coefficient and system stiffness. (Note: values not highlighted in yellow are calculated by the spreadsheet)

Based on the following values seen in Figure 1., the spreadsheet is able to calculate pedestal natural frequency, system natural frequency (three pedestals and the flexure plate), system mass, system displacement, and most importantly, the damping ratio. Furthermore, the system displacement and transmissibility are plotted against forcing frequency (seen in Figure 2 and Figure 3).

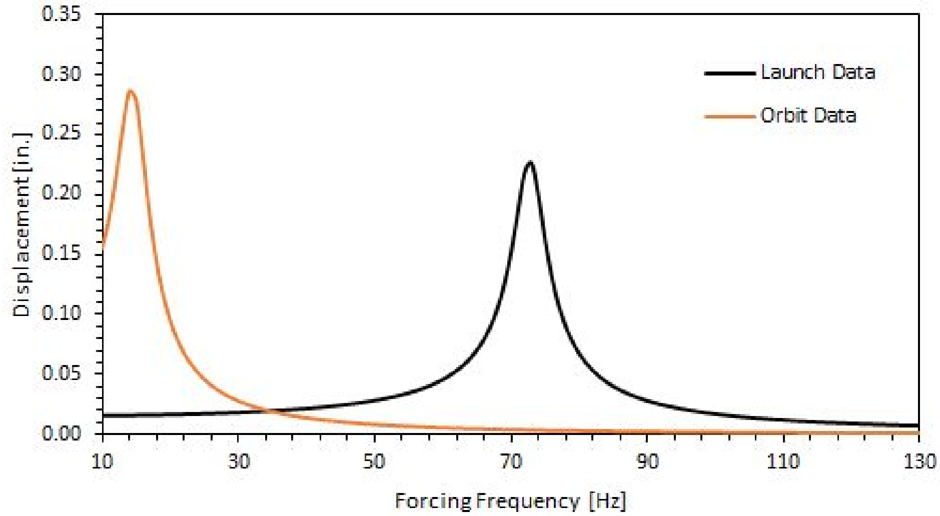

Figure 2: Comparing the Overall Displacement of the System During Launch vs. Orbit.

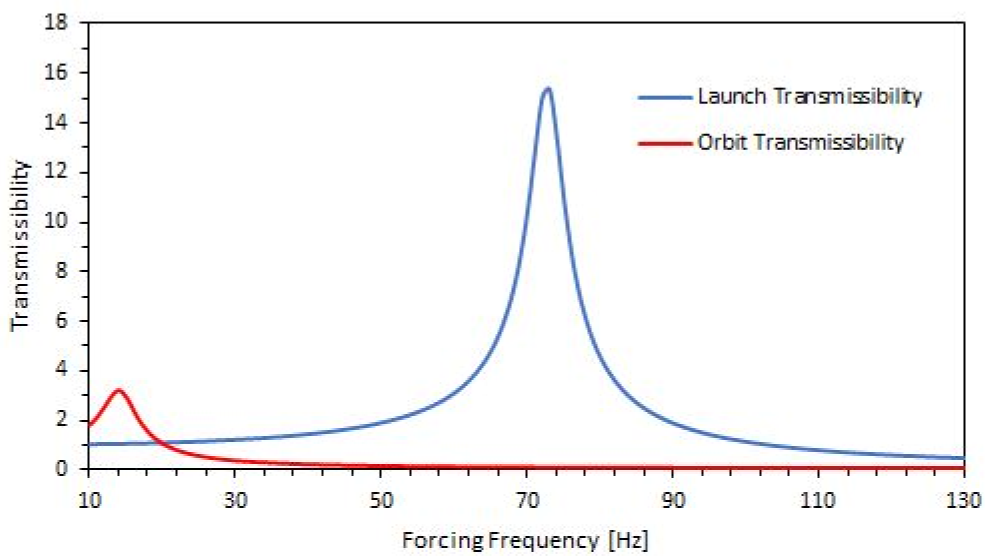

Figure 3: Comparing the Transmissibility of the System During Launch vs. Orbit.

When comparing the data from our system compared to the 2018-2019 senior project, we noticed that we had similar transmissibility values, verifying that our calculations are reasonable (although nowhere close to the target transmissibilities for both launch and orbit). After running initial calculations, our team recommends using the MATLAB script (mentioned in the next section) to provide the Excel sheet values for pedestal stiffness and damping ratio to get more accurate results.

VEM Strength-Predicting Calculator

The 2018-2019 senior project had previously performed testing which validated the damping effects of various VEMs. We used this data to select a prototype VEM, Flex Seal, to test at Cal Poly with potting capabilities for shape and application versatility with our modified design. However, we needed to validate whether or not the VEM would fail with the predicted shear stress and required deflection to hit the damping requirements of our system.

We conducted our own tests per ASTM D1002 (Adhesive Lap Joint Shear Testing) to find the joint capacity of the VEM sandwich beam. Separately, we also conducted a composite beam sandwich panel test per ASTM E756-05 (Measuring Vibration-Damping Properties of Materials) to find the shear modulus of our prototype material. Those values are inputted into Volkersen's equations (for shear stress in VEM) and provide a maximum bondline shear strength for that particular VEM as shown in Figure 4.

	Setup				Intermediate Calculations			
Joint Capacity	Load	P	2300	N	ω	0.095820649	Shear/axial stiffness ratio	
	Length of overlap	L	0.1016	m	ωL	0.009735378		
					if wL<<1 = uniform			
		Adherends			if wL>>1 = non-uniform			
		t (m)	b (m)	E (N/m^2)				
	Top (1)	0.002	0.025	6.90E+10				
	Bottom (2)	0.002	0.025	6.90E+10				
		Adhesive			C1	891258.8217	pa	
Flex Seal modulus validated per		t (m)	b (m)	G (N/m^2)	C2	-4338.336492	pa	
ASTM E756-05		0.060	0.025	1.93E+06		891258.8217	pa	Maximum Bondline Shear
					τ	129.261613	psi	

Figure 4. Calculation of maximum shear strength using specified test results. Inputs include dimensions of beams and VEM layer, as well as the adherend elastic modulus (aluminum) and VEM shear modulus.

It is important to note that the bondline shear stress does not have a uniform distribution, however, the shear/axial stiffness ratio (ωL) provides a gauge as to whether or not this value can be evaluated as uniform. In our case, this value is much less than one, so we can assume a uniform bondline strength distribution.

Next, the calculator takes the geometry of our system's linkages, the shear moduli of both prototype and actual VEMs, and the maximum bondline shear strength of the Flex Seal.

DIMENSIONS				Intermediate		
OD	0.58	in		R	0.165	in^4
ID	0.25	in		J	0.010726	in^4
θ	45	deg				
L	2.125	in				
PROPERTIES						
G	280	psi	(Tested VEM)			
G	1000	psi	(Actual system VEM)			
Flex Seal						
STRAIN	0.461649	in/in	(maximum allowable strain applied to disk, Flex Seal)			
Applithane						
STRAIN	0.129262	in/in	(maximum allowable strain of actual VEM)			

Figure 5. Approximate the maximum strain that the system VEM can withstand without failure.

The output is a strain value for the maximum strain that the real VEM can withstand. This strain value can help to calculate the maximum allowable link movement and maximum load applied to the assembly.

3.1.2 MATLAB & Simulink

Due to the interconnected nature of the MATLAB scripts and the Simulink models, it would be inappropriate to talk about the two as two different sections. Instead, the following sections will talk about each MATLAB script and their respective Simulink block diagrams.

Strain Energy MATLAB Script (2018-2019 senior project)

When our team first started the project, we attempted to use the MATLAB script from the previous year's senior project. We found that with every design change, the script became less and less relevant, eventually ignoring the script altogether. This led to a newly revised MATLAB script for our new design.

System Characterization MATLAB Script (2019-2020 senior project)

To gain better results from the Excel spreadsheet that correlated to our new design, we created a MATLAB script that calculates the following values: system stiffness (for launch and orbit), pedestal stiffness (for launch and orbit), VEM stiffness, and linkage damping coefficient. First, the user defines a target stiffness, material properties of the pedestal, and material properties of the VEM. Using the following values, the script calculates the stiffness of each "torsional" joint by calculating torsional stiffness to linear stiffness and provides a target stiffness value with its accompanying damping coefficient value necessary to hit the target natural frequency (both for launch and orbit conditions). An excerpt of the torsional member stiffness calculations are shown below:

```
% VEM Torsion Member Stiffness in each joint in N/m
L_link = 2.125;                                 % Moment arm of linkage in in.
Torque = L_link*lb_force*cosd(45)/2;     % Torque applied at the VEM joint in lb-in
k_tor = 142.75;               % Stiffness of each VEM joint in the linkages in lb/in (25000 N/m target)
x_vert = lb_force/k_tor;                         % Vertical displacement in in., small angle approx.
theta = x_vert/tand(45);                        % Theta in small angle approx. using linkage angle
J = Torque*L_VEM/1000/theta;      % Polar Moment of Inertia in in^4, 1000 is VEM Shear modulus
d_in = 0.25;                                      % Inner diameter of VEM joint in in.
d_out = (J*32/pi()+d_in^4)^(1/4);           % Outer diameter of VEM joint in in.
```

Note that the above calculations assume that the VEM behaves in a linear manner, as seen in the linear to torsional stiffness conversion. Unfortunately since visco-elastic material is a nonlinear material, this is not the most accurate way to calculate the stiffness of the VEM. Further iterations of this code should account for nonlinearity of the material.

Orbit Configuration SimScape Model (2019-2020 Senior Project)

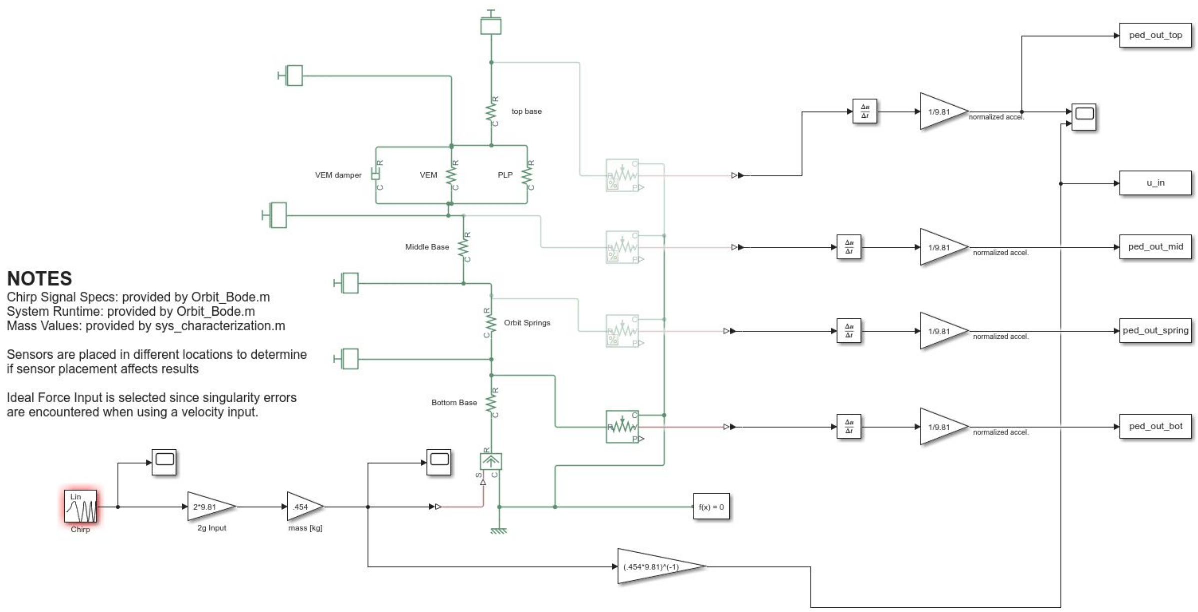

Figure 6: Orbit Configuration Pedestal Simscape Block Diagram

Our team noticed that if you try to link the SimScape model to this system characterization script, you will encounter singularity errors. Simulink will recommend decreasing the step size, however, we have gone as far as decreasing step size to 1E-7 seconds without any success. Our team recommends that any future teams do not link the Simscape model to the system characterization MATLAB script.

The first block to take notice to is the chirp signal. Simulink has two different chirp (sine sweep) blocks to work with, our team has found that the chirp signal from the digital signal processing toolbox (DSP toolbox) allows more degrees of customization. First, one must decide whether the chirp signal increases at a linear versus logarithmic rate. (The pictured pedestal model utilizes a linear sine sweep since it is easier to model in the later mentioned bode plot MATLAB script.) The following inputs for the sine sweep are specified by the user in the bode plot MATLAB script: initial frequency, final frequency and sampling time (sampling rate frequency).

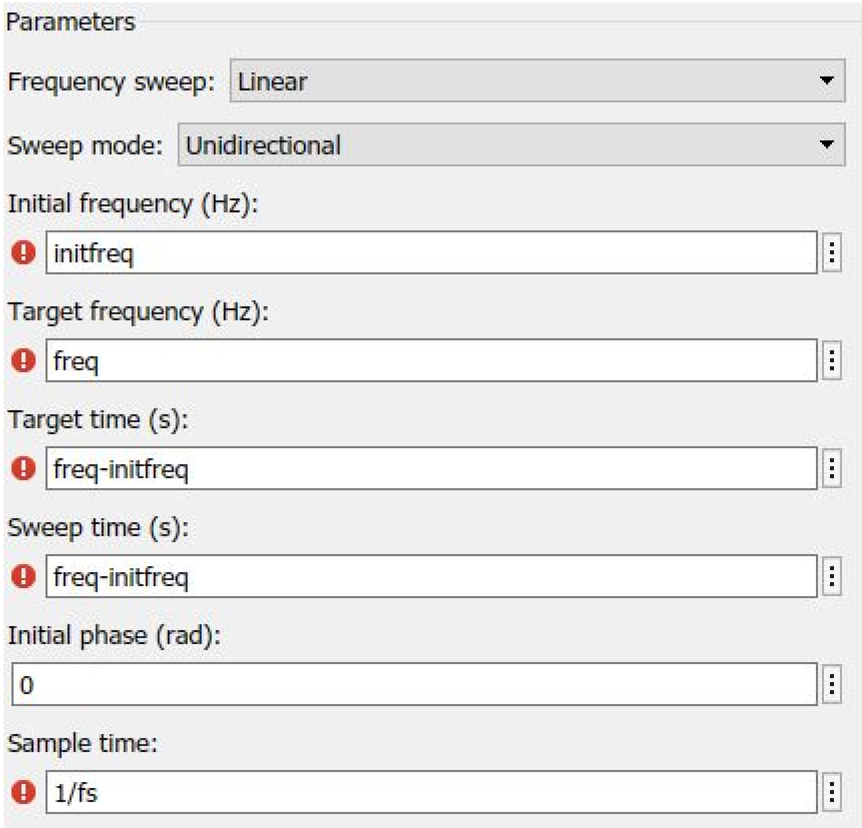

Figure 7: Chirp Signal Block Parameters

As a rule of thumb, the sampling rate frequency should be at least three times faster than the fastest expected frequency (including noise). After setting up the sine sweep block, the signal must be multiplied by gravity and a multiplication factor for the various loading cases. This allows the sine sweep to be utilized as an ideal force input. In other models, our team used an ideal velocity source instead of the ideal force source, however, our team encountered singularity errors. As for all SimScape models a "solver configuration" block must be included anywhere in the model and a reference ground must be included for the force source as well as all of the sensors.

Notice that each "stage" of the pedestal is assigned its own respective mass. Initially, we modelled the pedestal using a lumped mass, however, we were getting inconclusive results and strange plot shapes. When diagnosing the issue, we noticed that many of the masses were similar in magnitude, therefore, it was inappropriate to model the pedestal using a singular lumped mass. Additionally, by modelling the pedestal with multiple masses the output graph is no longer a singular peak. This makes sense since each "stage" has its own natural frequency and therefore smaller peaks should be seen at each resonant frequency.

Each stage is placed in series, with sensors placed accordingly. Finally,derive the velocity output of the displacement sensors once to ensure the lowest amount of corrupted data going to the MATLAB workspace.

Bode Plot MATLAB Script (Original Author: Kevin Ziemann)

The bode plot MATLAB script takes the output values from the Simscape model and converts the time domain response to a frequency domain. This allows users to view the resonant frequency and amplitude against a varying frequency on the horizontal axis instead of trying to infer the frequency from a time plot. The MATLAB script uses fast fourier transform (fft) to change the domain of the system from time to frequency. Finally, there is an additional function that takes the frequency response data and plots the phase and amplitude on a bode plot.

3.2 Finite Element Analysis (FEA)

FEA is an incredibly useful tool, but it should not be used to drive the design of the system. Its primary purpose is to verify your hand calculations and basic stiffness models as well as tuning. ABAQUS or ANSYS or an equivalent NASTRAN solver should be used as the primary FEA software. It is highly recommended that someone in the group learn one of these than use built-in FEA packages such as SolidWorks and Fusion.

3.2.1 Analysis Focus

The most important design consideration, damping, should NOT be done in FEA. It is extremely hard to replicate the properties of viscoelastic material as it exhibits nonlinear behavior. Nonlinear analysis also drastically increases simulation run time as it has to solve for multiple steps than just one. Listed below are the two main types of analysis that will benefit the design the most.

Modal Analysis

- Modal frequencies of the system
- Strain energy density

Structural Analysis

- Quasi-static loading
- Dynamic loading

3.2.2 Modal Analysis

Modal analysis is used to verify your frequency range of the system. It should be used to compare the analysis from the 1-D Simscape model. The most important factors to consider when performing a modal analysis would be the mass and stiffness properties of the materials used. Stiffness is not just determined by material properties, geometry and cross-section is also taken into consideration. Correctly modeled boundary conditions are needed to ensure the model behaves similar to what a real life prototype would.

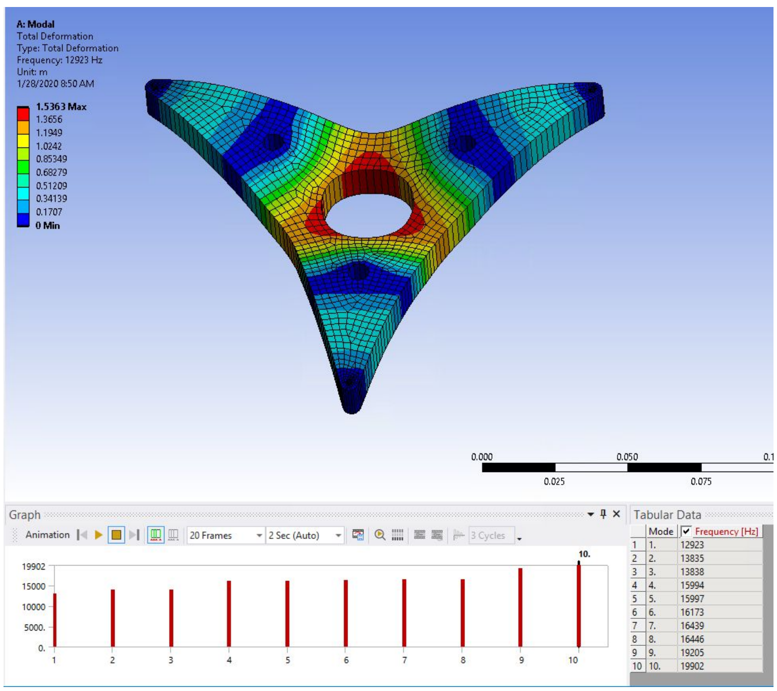

Figure 8. Modal analysis of a baseplate in ANSYS

When attempting to solve the simulation, it is recommended to solve for multiple modes instead of just the first mode. Sometimes the design might have a first frequency mode that does not align with your mode of interest. For example, an object might have a first mode at 15 Hz but in torsion rather than vertical displacement (which might be more important for damping). In this case, you would want to keep looking at the other modes until you see the mode you are interested in. The image shown below is a good example. The 2nd mode is the one that is most critical for our design, but there is a first mode of 6.8Hz that is less important. The overall frequency of this particular design would be 27.6Hz instead of 6.8Hz.

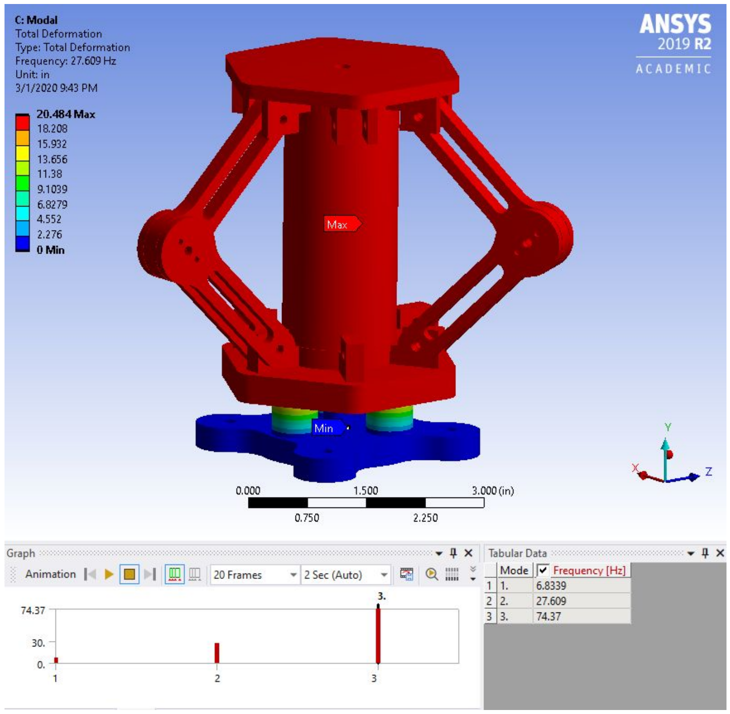

Figure 9. The vertical mode at 27.6Hz is more important than the 6.8Hz first mode.

Strain energy density can also be extracted from a modal analysis; however, we had some issues with getting it to work in ANSYS. It is more important to find strain energy through modal than a structural analysis, as the strain energy at certain modes are more important to analyze damping behavior than just performing a simple directional load to analyze areas of highest strain energy.

3.2.3 Structural Analysis

Structural analysis is the best way to determine if your system meets the quasi-static requirement. It can also be used to analyze specific structural members under their worst case loading to check for failure. Sometimes quasi-static loading might not be the worst case scenario to design for. Check the transmissibility and acceleration input to see if the dynamic loading is higher than the quasi-static. In our case, with an input acceleration of 8g with a maximum transmissibility of 4, our highest acceleration would be 32g instead of the 20g quasi-static requirement.

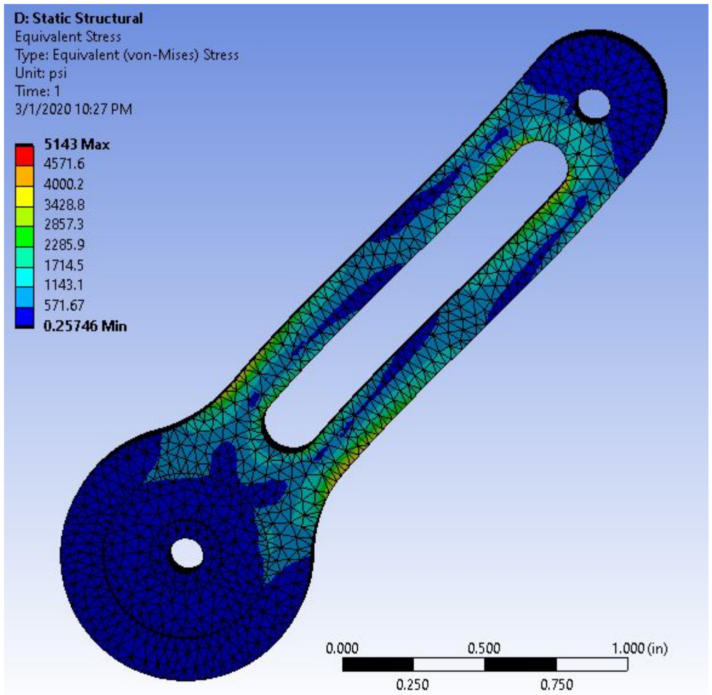

Figure 10. Structural analysis of a linkage by applying a bearing load of 32g