Spring 2021

FEASIBILITY OF ADDITIVE MANUFACTURING FOR VIBRATION ISOLATION SYSTEMS

Matthew Youngblood
may32@zips.uakron.edu

Christian Velo
cdv16@zips.uakron.edu

Gary Facemyer
gwf9@zips.uakron.edu

FEASIBILITY OF ADDITIVE MANUFACTURING FOR VIBRATION ISOLATION SYSTEMS

By

Gary W. Facemyer Jr

Matthew Youngblood

Christian Velo

Final Report for 4600:471 Senior/Honor Design, Spring 2021

Faculty/Honors Advisor: Dr. Quinn

Honors Reader 1: Dr. Ajay Mahajan

Honors Reader 2: Dr. Alper Buldum

03 May 2021

Project No. 50

Abstract

During transport sensitive payloads can become subject to a multitude of vibration or shock environments that can lead to damage. Using only additive manufacturing we seek to design an isolation system that can provide enough vibration and shock damping to properly protect the payload. Using modern FEA software we were able to quickly analyze a variety of solutions and determine the best one. Based on the results we have achieved through analysis, as well as pushing the manufacturing capabilities of 3D printers, we believe it is plausible to use additive manufacturing to create a fully capable vibration isolation system.

Acknowledgements

All three of us have spent the past five years of our lives undertaking what may be one of the most stressful part of our lives. Nonetheless, here we are today completing our final work as undergraduate students at The University of Akron. The completion of this project, and our degrees, would not have been possible without the help and guidance of those who once stood in our shoes. For their contributions to this project and guidance over the last five years the group would like to express our appreciation and gives thanks:

To Dr. Dane Quinn, The University of Akron, and Dr. Adam Brink, Sandia National Laboratories, for not only providing us with a very interesting topic for our capstone project, but also for their guidance, knowledge, and support they have provided to us every step of the way.

To Ansys Inc. whose generous contributions made it possible to take this project to the next level.

To the endless list of professor's and faculty in the College of Engineering who have helped us to stay on our desired paths and find success.

To our family, friends and loved ones whose sacrifices, and support may have sometimes gone unnoticed, but will forever be appreciated and never be forgotten.

Again, we thank you.

Spring 2021 Senior Design Group #50

Table of Contents

1. Introduction

1.1 Background

One of the many duties of Sandia National Labs is maintaining responsibility for all the United States' nuclear weapon systems and components from beginning to end of their lifetime, which usually means dismantlement and disposal [1]. One of the primary tasks associated with maintaining this responsibility is the safe transport of nuclear weapons across the United States which are generally moved utilizing the country's highway system as well as indistinct and unmarked truck and trailers. During transportation, especially on roadways, payloads can be subjected to a multitude of vibrational environments as well as shock environments in the case of a crash. Obviously, anything involving nuclear weapons has large risks associated with that, so the team at Sandia Labs focuses on designing safety components and subsystems to prevent damage or energy being transferred to any nuclear explosive material or other components [2]. From here is where the 'Vibration Isolation' part of our project title is derived from.

It is easy to say that Sandia National Labs is easily a leader in the field of materials science research and development. Currently they are focusing on the research of metamaterials. But what are metamaterials? Metamaterials are any material with properties that have been engineered to have values that are not typically naturally occurring. Due to the complex geometries sometimes associated with these materials they are typically created using additive manufacturing methods. With how young additive manufacturing is, as a field of engineering, the study and usage of metamaterials will be pushing the boundaries of what this style of manufactured part is capable of. From here is where the 'Additive Manufacturing' part of our project title is derived from.

Putting the two things together nets us our project: 'Feasibility of Additive Manufacturing for Vibration Isolation Systems.' During this project we attempted to determine whether it is feasible to only use additive manufacturing, with the caveat of being allowed basic fasteners, tape etc., to design a system that protects a sensitive payload from a multitude of external environments during transportation. In Figure #1, seen below, you can see exactly how we expect our 'input', the external environments, and our 'output', the reduced payload loading, to interact with each other. The 'Vibration Isolation System' block is what we will be focusing much of this design. In

future sections of this report, we will go into greater detail on what the external environments constitute as well as system requirements.

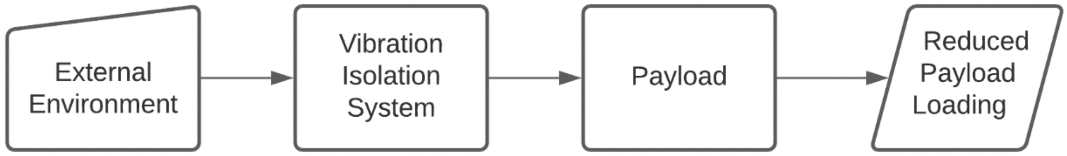


Figure #1: Block Diagram
of IO flow in the system

1.2 Product Definition

In today's modern era of technology there are a multitude of things that incorporate electronics which are generally quite sensitive, in other words delicate, by nature. When critical systems are comprised of these sensitive components and need to be transported it is crucial that they can withstand, and can properly operate in, routine external vibration environments. It is also crucial that in the event of any extreme conditions, such as those caused by crash events, that the critical system avoids catastrophic failure. To propagate these requirements one of the most common approaches is to design a separate isolation system, to handle the external environments, that the critical system can then be placed into with little regard to the external conditions.

Such systems have been largely used, across a multitude of industries, for quite a long time now. Prime examples of these kinds of systems are; suspension systems on vehicles, shipping container isolation systems, earthquake isolation systems for houses as well as isolation pads for machinery in factory environments. Our vibration isolation device will be designed to secure a payload, dampen external vibrations within allowable levels, and ensure the payload survives a crash event.

1.3 External Environments

1.3.1 Narrow-Band Loading Environment

Based on the frequencies of typical road vibrations for the analysis of our device we will mostly be interested in the frequency interval starting at 0 Hz and extending out to 2000 Hz. Based on this frequency interval it would be favorable to have all the natural frequencies be below 2000 Hz to prevent the phenomenon known as resonance from occurring. However, if any resonant peaks do so happen to fall within our frequency interval, then the magnification factor at those peaks should be no greater than 1.50. This design target will help to prevent any harm coming to the payload when the system is vibrated at those natural frequencies.

1.3.2 Random Vibration Environment

A random vibration environment is just as the name suggests, random. Throughout the frequency range of the vibrations their magnitudes are constantly up and down with what appears to be no pattern. Due to the nature of this project, transportation on roadways, we want to create a random vibration spectrum that is well representing of typical road conditions. To facilitate this this, we will consider a Brownian noise spectrum.

Brownian noise spectrums have a power density which decreases as the frequency of the noise is increasing. This aspect is what makes Brown noise the best choice to represent road conditions. Utilizing MATLAB and a Welch's Power Spectral Density Estimate we were easily able to produce a spectrum that is suitable for analysis of our design. This spectrum can be seen below in Figure #2, and our MATLAB code used to produce the spectrum can be seen in Appendix A.

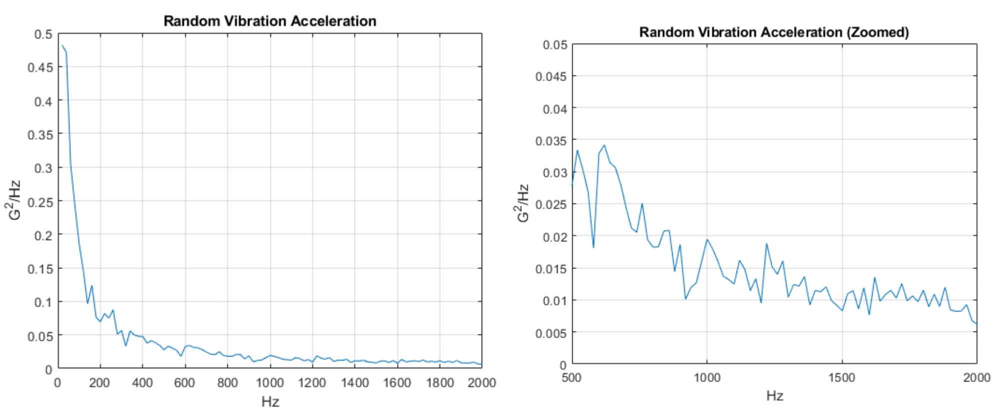


Figure #2: Left, Brownian Vibration Spectrum. Right, zoomed in spectrum to show magnitudes from 500-2000 Hz.

1.3.3 High-Acceleration Shock Loading Environment

As we mentioned previously in the report this vibration isolation system should also be able to ensure the survival of the payload in a crash event. The best way to model crash events is by simulating a 'High-Acceleration Shock Loading.' Essentially what this means is we apply a large amount of acceleration over a very short amount of time, which results in the system being 'shocked.' Typically, these kinds of environments are simulated using a drop table test which provides those high amounts of shock over a short time period while maintaining repeatability [3]. Drop table testing also has the benefit of producing a shock profile that can be easily represented mathematically [3].

For the purpose of shock analysis of our system we will be considering a Haversine shock loading with an interval of t = 0.005s and a magnitude corresponding to a 3m drop for the overall weight of the system and its payload. To model our Haversine shock wave we will first begin by calculating the impact velocity of our system. We first begin with our potential energy in Equation #1 where U is the potential energy, m is the overall mass of the system, g is gravity and h is the height of the drop [3].

$$U = mgh$$

Equation #1: Potential Energy [3]

Then we use the kinetic energy, Equation #2, to perform an energy balance and solve for our impact velocity in Equation #3 [3].

$$K = \frac{1}{2}mv^2$$

Equation #2: Kinetic Energy [3]

$$v = \sqrt{2gh}$$

Equation #3: Impact Velocity [3]

Now that we have our impact velocity, we calculate the peak acceleration using Equation #4, where T is the interval of loading, in our case 0.005s [4].

$$P = \frac{2v}{T}$$

Equation #4: Peak Acceleration [4]

Now we are finally able to apply our Haversine formula, Equation #5, to create our acceleration shock wave where 'T' is the duration of our loading again [4].

$$A = P * sin^2\left(\frac{\pi t}{T}\right)$$

Equation #5: Haversine Acceleration [4]

Applying these formulas and using the overall mass of our system then yields the Haversine shock wave that can be seen in Figure #3. The MATLAB code used to generate this can also be seen in Appendix B.

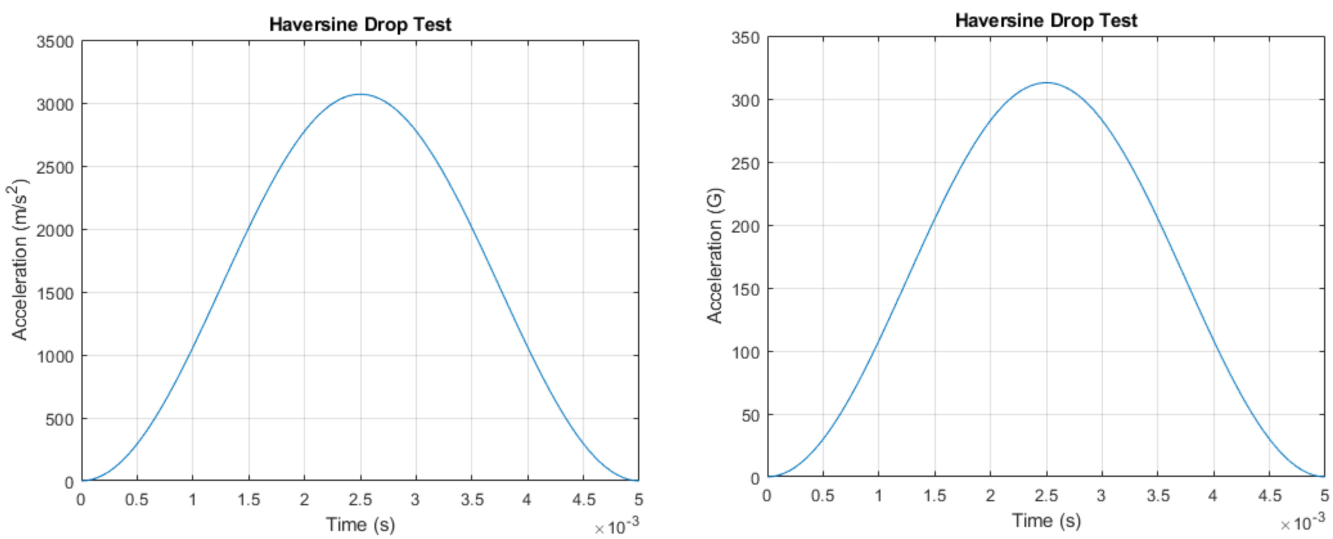


Figure #3: Left, Haversine shock spectrum in units of m/s^2.
Right, Haversine shock spectrum in units of Gs

2. Design

2.1 Design Requirements

2.1.1 Dimensional Requirements

For this project, our scaled payload will consist of a cylinder with a mass m = 3.5 kg, a radius r = 4 cm, as well as a height h = 10 cm. The vibration isolation system must be no larger than a cylinder with radius R = 10 cm and height H = 24 cm. To put these measurements into perspective this is approximately like fitting a standard soup can within a standard paint can.

2.1.2 Design Objectives

There are a few design objectives for this vibration isolation system. The first is that it minimized the acceleration loading on the payload under shock conditions to prevent damage to the payload from occurring. The caveat to this is that there is no requirement that our vibration isolation system survives the shock loading, only the payload. The second objective is to minimize the maximum displacement levels of the payload under the environmental vibration conditions mentioned above. However, it is only required to meet two out three vibration reduction amounts and meeting all three reduction quantities is considered a stretch requirement. In addition to these targets the final design should aim to minimize to overall weight of the system.

2.2 Materials

2.2.1 Material Choices

The usage of additive manufacturing limited the scope of material selection. We initially decided we wanted to use a two-material system, with a more rigid, but lightweight outer shell and an internal more flexible material with better vibration isolation and dampening properties. The selection process for these materials relied heavily upon what materials were compatible with the additive manufacturing technology we had access to, as well as what materials have a high success rate when being used for complex geometries. Through initial research into existing material science and current additive manufacturing practices, we were able to decide upon two materials for our design. We chose the titanium alloy Ti6Al4V (Ti64) for our outer rigid shell, and Nylon PA6 for our internal more flexible and dampening structure.

2.2.2 Homogenized Meta-Materials

Due to the feasibility of 3D printing lattice structures as well as lattice's performance in terms of vibration damping capabilities it will be the center of our focus for this project [5]. Working with lattice structures in an FEA environment can be very tricky however as they cause very dense meshes, if the mesh will even create due to small element sizes, that are almost impossible to generate solutions for. An example of such a meshed lattice structure can be seen in Figure #4. For these reasons 'homogenizing' the material became a necessity for us.

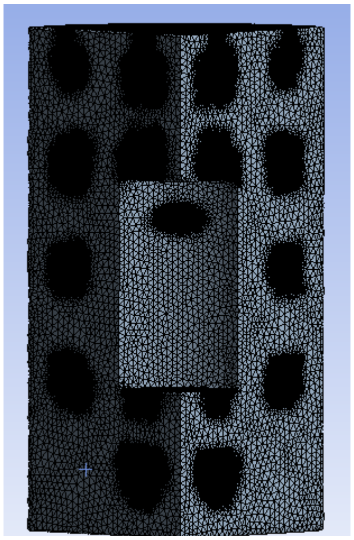


Figure #4: Quarter cut of an initial design with a fully meshed lattice structure. The mesh is so dense it is virtually unsolvable.

Homogenizing a meta-material is the act of taking a structure, such as lattice, that is made from a certain material, performing testing, and then finding the 'material' properties of the lattice. This allows us to perform analysis on the large and complicated lattice structures as if they were made of solid material, drastically reducing the needed time and computer power. This process can be achieved using ANSYS Material Designer.

Within ANSYS Material Designer there are a few lattice parameters that need to be selected such as; Lattice Type/Shape, Cell Size, & Volume Factor. After selecting these parameters and choosing a material ANSYS Material Designer will then spit out a material profile. Due to the ease and simplicity of this process we were able to iteratively test a variation of lattice parameters to draw the conclusion that small changes in parameters were going to have very little effect on the performance of our system.

The next step for us was to validate that the material designer was producing proper and useable results. We would expect small differences in results between a homogenized lattice and an actual lattice due to things such as the mesh differences. To validate this, we took used a 10mm Cubic Cell Lattice w/ 0.25 Volume Factor. We then homogenized this lattice and used both the actual lattice and the homogenized material to create a 40mm cube. After applying the same static loading to the top face of cube there was only a 0.03 mm difference in maximum deflection values and this analysis can be seen in Figure #5. The 0.03mm difference in the two is so small it is almost negligible and should not have a large impact on our analysis.

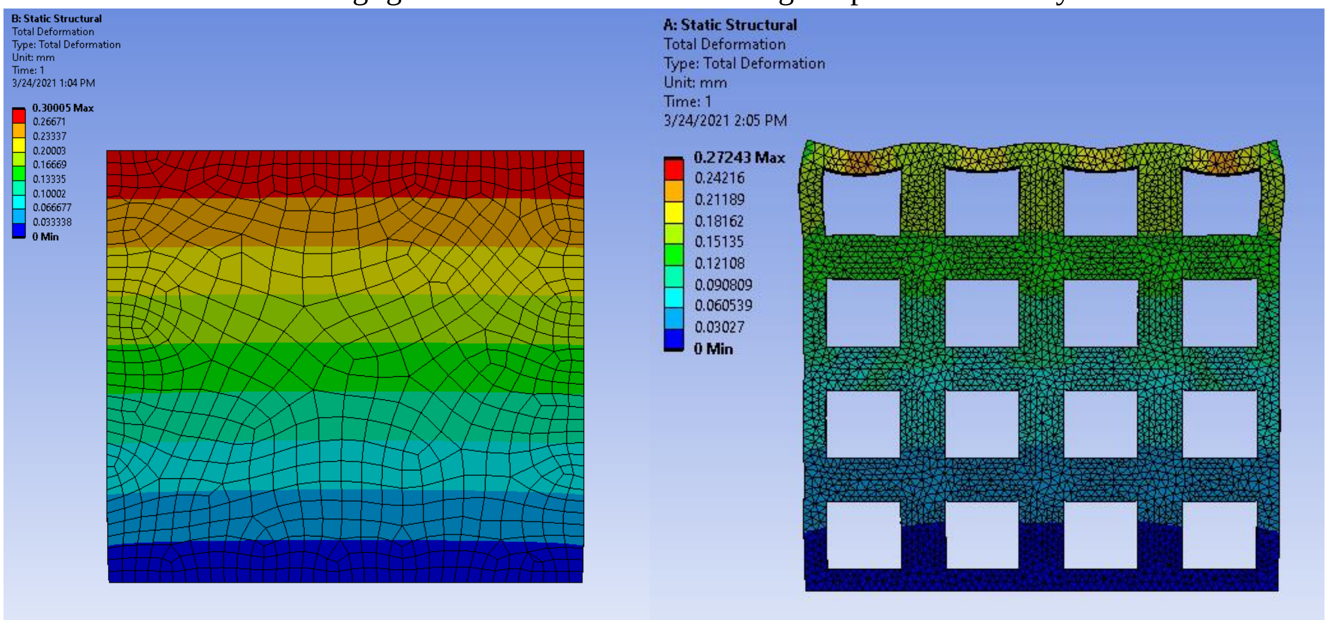


Figure #5: Left, Homogenized Lattice. Right, Lattice Structure

2.3 Design Procedure

Utilizing the two types of materials that we chose to move forward with, a metal and a plastic, it was quite easy to brainstorm an initial design. This design focused on producing two separate parts; an inner nylon core to hold the payload, and an outer titanium shell to add strength and rigidity to the system while still being relatively lightweight. Our thought process here was to use the titanium outer shell to help absorb energy in a shock environment, while the nylon provides vibration damping capabilities. Using the research license granted to us by ANSYS we were able to perform Finite Element Analysis to evaluate the performance of our system and make needed changes quickly and efficiently.

The beauty of combining an additive manufacturing product with FEA analysis is how rapidly testing and prototyping can be accomplished. Using FEA to quickly analyze multiple design versions, as well as homogenized materials, and make any changes to the design to produce satisfying results. This aspect allowed us to easily set-up 'test' environments to ensure the software or our system would perform as expected.

2.4 Final Design

Using our ability to rapidly analyze system performance we went through a few different design variations. Our original design simply consisted of what could be described as a bucket made of titanium lattice that we then inserted our nylon lattice cylinder, with the payload nestled inside, into. This design was useful, because it was simple, in helping to establish our analysis workflow but upon further inspection could be improved upon.

In Figure #6 you can see that some modes of deformation appeared to be far from symmetric, meaning it could affect some of our assumptions based on that symmetry.

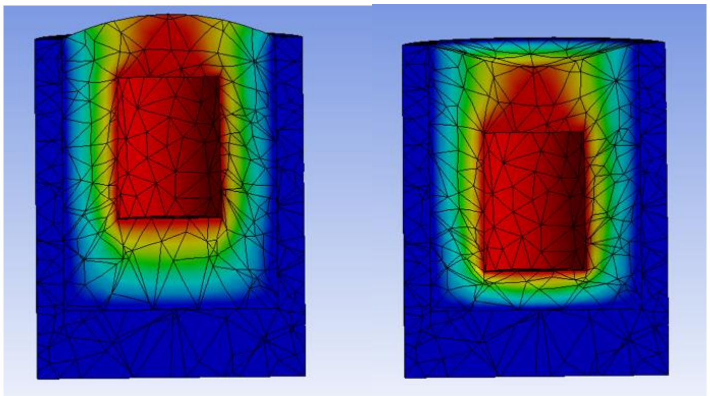


Figure #6: Unsymmetric deformation mode

To combat this, we decided to add a titanium 'cap' to the top so that the nylon inner core is fully encompassed by the outer shell. This had the added benefit of better representing 'bonded' contacts between parts in the assembly. We then better represented these contacts, with the benefit of adding some rigidity to our design, by modeling solid layers of each material at interfaces. Meaning materials alternate solid to lattice to solid from outside in. This is the final model we analyzed and is the modal analyzed all throughout the next section. A half-cut section of this model, a detailed drawing, as well as the lattice unit cells for each material can be found in Appendix E.

2.5 Design Analysis

To perform the analysis of all our design iterations we used ANSYS FEA software, more specifically the ANSYS Mechanical program. We were given full research access to this program thank to the kind sponsorship from ANSYS Inc. Initially after receiving our research license, we took some time to really explore the capabilities of the software and learn proper procedure when utilizing it. After exploring how we could use Topology Optimization, which will be discussed more later, we finally settled on a workbench flow which can be seen in Figure #7 below.

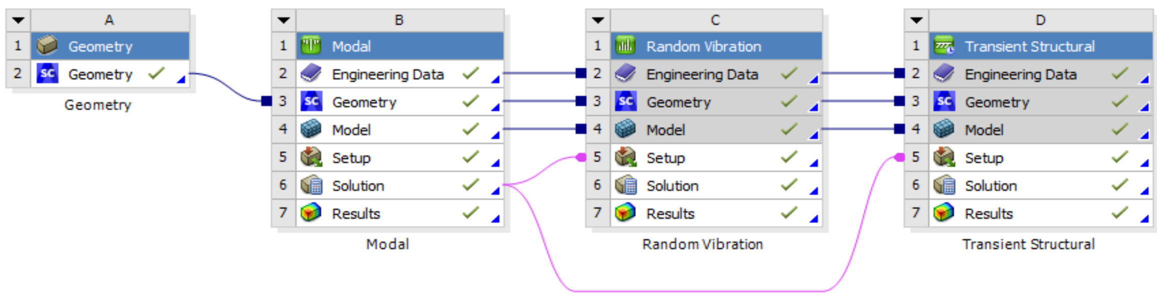


Figure #7: ANSYS workbench analysis workflow

As you can see from this workbench figure, we first began by creating our geometry, modeled in ANSYS Space Claim, which we then performed a Modal analysis on. This Modal analysis was then used as the 'pre-stress' environment to perform the remaining Random Vibration and Transient Structural analyses.

2.5.1 Modal Analysis

A Modal analysis is generally the first step in any form of vibration analysis. The results you would expect to get from a 'Modal' analysis is right in the name, we would expect to receive the modes of deformation for our model. Along with these ANSYS calculates the resonant, or natural, frequencies of the model. After creating the geometry of our inner and outer shell we must prepare the model properly to achieve accurate results. The first step was to apply the payload. To do this the mass of the payload was applied, as a rigid point mass, to the inside cavity. A fixed support was then applied to the bottom face of our model to represent its contact

with the vehicle during transport. The point mass and fixed support can be seen below in Figure #8.

After properly preparing our model for modal analysis the only thing left to do is run it and look at the results. Since we were most interested, and concerned, about the vibrations occurring

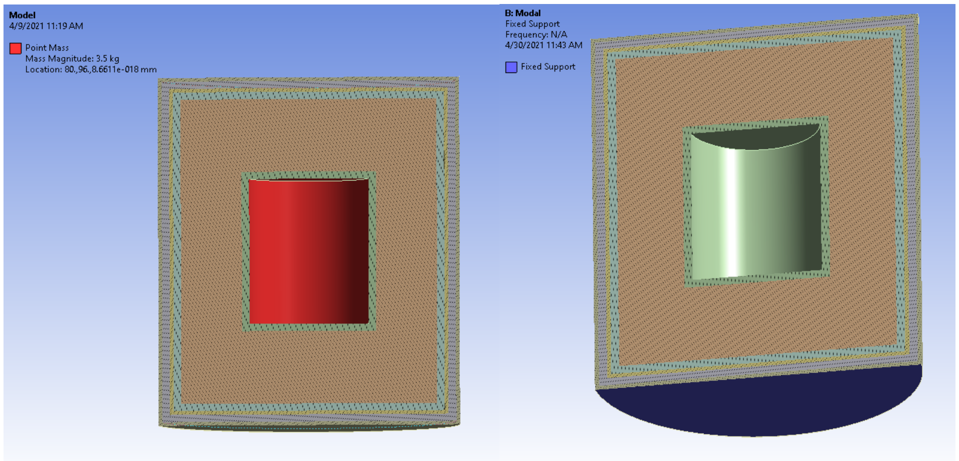


Figure #8: Left, Point Mass, Right, Fixed

below the 2000 Hz mark we calculated modes out to 4000 Hz. This is standard practice for your sampling frequency to be double the range you are interested in. This resulted in us finding nineteen different natural frequencies within this range however, only the first five were below 2000 Hz. With these natural frequencies ANSYS provides us with the mode & shape of deformation at each resonant frequency however, it should be noted that in these types of analysis the magnitude of deformation is meaningless and only the shape is important. The resonant frequencies below 2000 Hz can be seen in Figure #9, while the first mode of deformation can be seen in Figure #10. For the full list of resonant frequencies and the first five modes of deformation please see Appendix D.

	Mode	Frequency [Hz]
1	1.	346.22
2	2.	370.13
3	3.	848.87
4	4.	1203.3
5	5.	1327.4

Figure #9: First five resonant frequencies

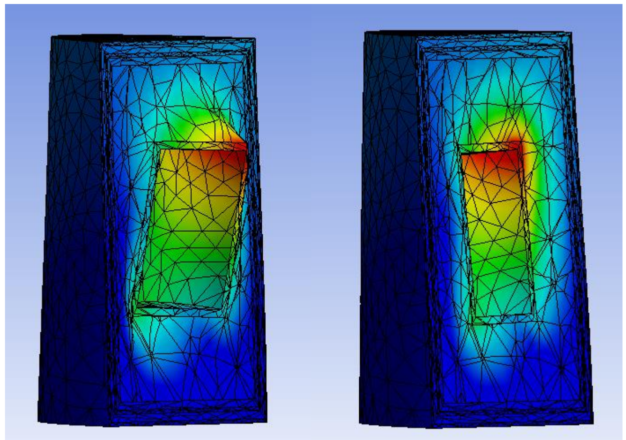


Figure #10: First mode of deformation

2.5.2 Random Vibration Analysis

As was seen earlier in our workbench, and based on the environments we are testing in, the next analysis will be for our random vibration environment. The setup for this analysis is quite straightforward and the only thing we will do is apply our Brownian acceleration spectrum from earlier, Figure #2, in the Y-direction through our fixed support. This will mimic the vibration of a vehicle, as it goes across a typical road surface, and into our isolation system.

To get our results we will simply apply a remote point to our payload cavity so that we can receive Response Power Spectral Density, or a Response PSD, in the forms of deformation and acceleration. Looking at the Deformation Response PSDs in the X & Y-directions, Figure #11 below, we see that our deformations are quite small values which is what we wanted to see based on our design requirements.

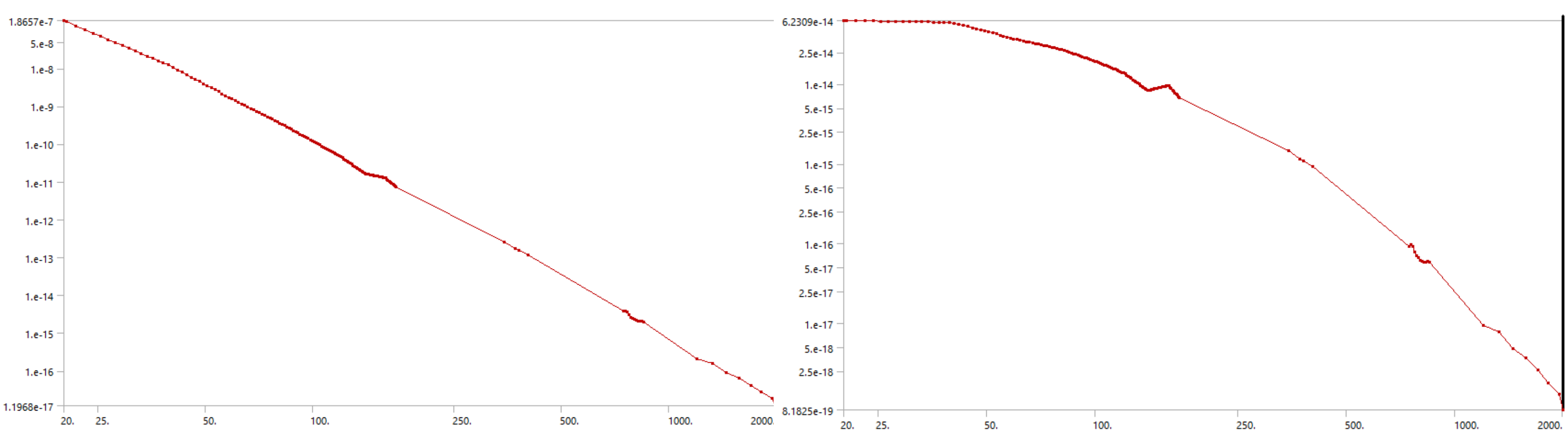


Figure #11: Left, Y Displacement PSD. Right, X Displacement PSD

There is however one more result we must look at from this analysis, that being the Acceleration Response PSDs. Our magnification factors from Brownian acceleration input to Acceleration Response PSD output must be < 1.5. Looking at our Acceleration Response PSDs, Figure #12, we clearly see peaks at the resonant frequencies, which is to be expected and can also be seen in our Displacement Response PSDs.

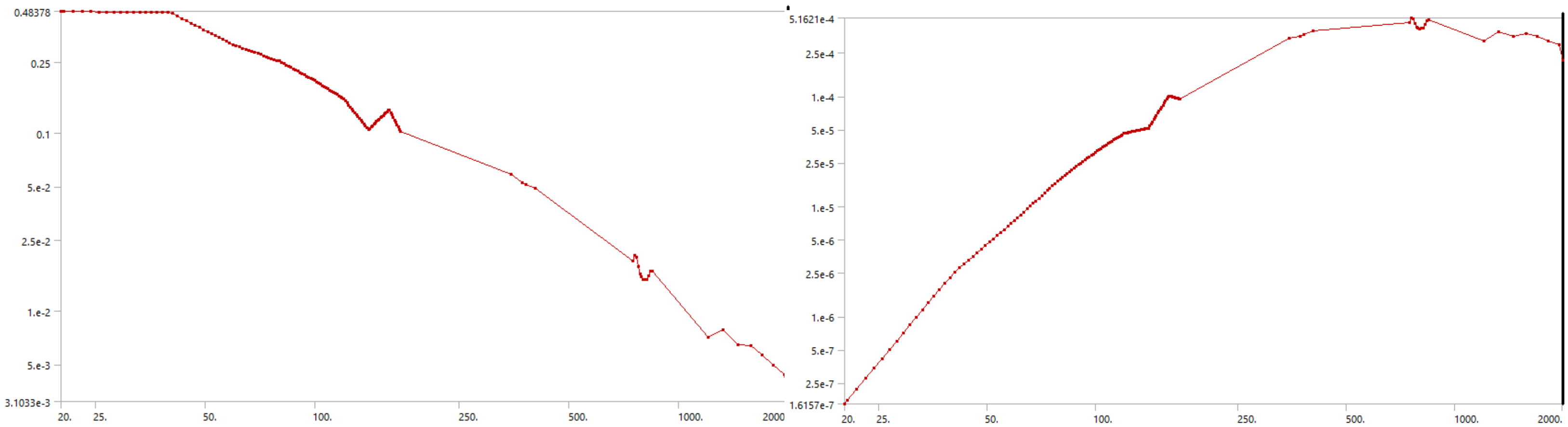


Figure #12: Left, Y Acceleration PSD. Right, X Acceleration PSD

Further analysis of the magnification factor shows that the highest one we see is approx. = 1.03 at the 2nd resonant frequency, 370.13 Hz. This is well within the allowable based on our design requirements, and a table showing the magnification factors for all resonant frequencies < 2000 Hz can be seen below in Table #1.

Resonant Frequency (Hz)	Input (G)	X Output (G)	Y Output (G)	X Mag. Factor	Y Mag. Factor
346.22	0.273776	0.018339029	0.242074369	0.066985558	0.884206369
370.13	0.223314	0.018724583	0.229549559	0.083848608	1.027922023
848.87	0.144063	0.022252416	0.129433381	0.154462634	0.898447215
1203.3	0.097488	0.017932094	0.084391943	0.183941017	0.865662406
1327.4	0.111307	0.019706344	0.088345911	0.177045748	0.793717384

Table #1: Resonant Frequencies < 2000 Hz Magnification Factors

2.5.3 High-Acceleration Shock Analysis

When being physically tested high-acceleration shock analysis is typically performed using a drop table. To mimic this within an FEA analysis you can use an ‘Explicit Dynamics’ simulation that would simulate the movement of the system as it fell, and then what happens once it collides with the drop table. These simulations do however require a ton of computing power and then still could take days upon days to run. Due to the nature of our rapid design & analysis cycle for this project this was not ideal, so we settled for a ‘Transient Structural Analysis.’ This type of

analysis allows us to apply a force, or in this case acceleration, that varies with time and see how the system responds. This analysis does however neglect any impact surface effects.

By applying our Haversine Shock Loading that we calculated earlier to our system and performing the analysis ANSYS provides us with deformation data on our payload pocket, seen below in Figure #13.

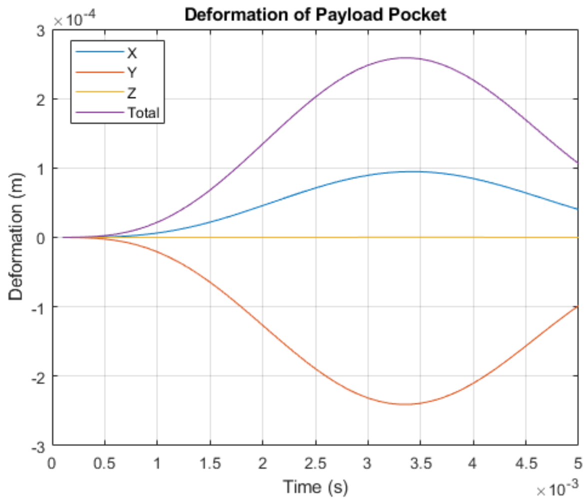


Figure #13: Transient Analysis Payload Deformation

What we are interested in here is the ‘Total’ deformation as it encompasses the movement along all axis and from it, we can derive payload velocity and more importantly payload acceleration. Using a basic MATLAB script, see Appendix C, we can take these data points, perform a polynomial fit, and then take the second derivative of that polynomial to find acceleration. As you can see in Figure #14 the system clearly loads up, the first peak, then rapidly unloads, the first trough. This cycle would continue until the system came to rest. What we are truly interested in seeing is that the maximum acceleration our payload felt was only about 11.6 Gs compared to the over 300G shock applied to the system.

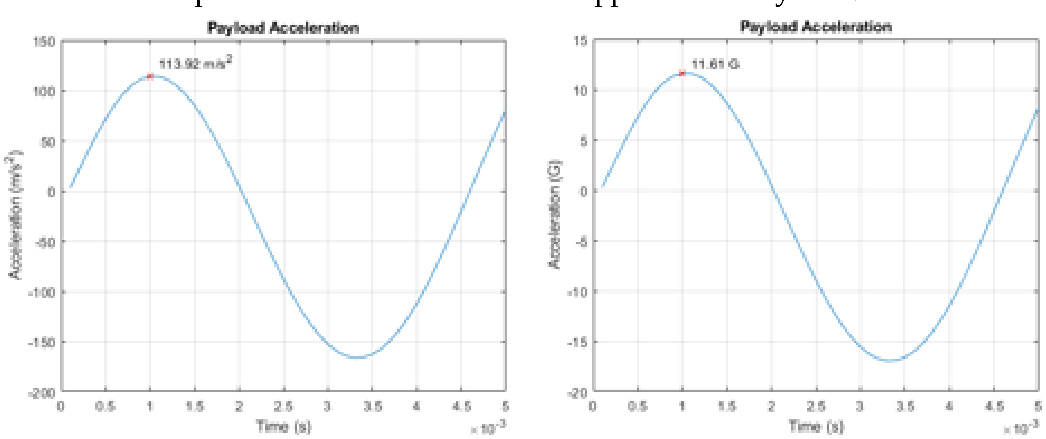


Figure #14: Transient Payload Acceleration shown in units of m/s^2 (Left) & Gs (Right)

2.6 Manufacturability

Unfortunately, due to our time restraints and budget, we were unable to manufacture a complete physical model of our system. Some of the factors that also led to issues is computational power for model slicing during print preparation, and 3D printing technologies available to us at the time. With the correct equipment, our model would be manufacturable with current technologies. Through our access to advanced titanium printing capabilities, we were able to create a scaled down full titanium 8th model of our system to display the overall concept and technology. Computing power during model slicing in the print file preparation portion of the nylon component hindered our ability to successfully print our inner portion of the system. A more advanced slicing software or improved hardware could be a solution to this problem, alongside access to a more appropriate printing method than the Form 2 SLA 3D printer we had access to during our printing steps. A problem faced when printing lattice, especially in a softer material such as nylon is internal supports. Newer technology such as Selective Laser Sintering (SLS) which does not require any internal supports, or the usage of dissolvable supports could help assist with this problem and allow for more complex geometries.

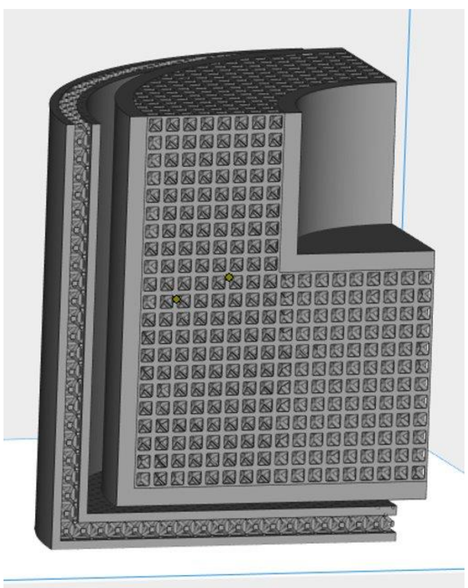
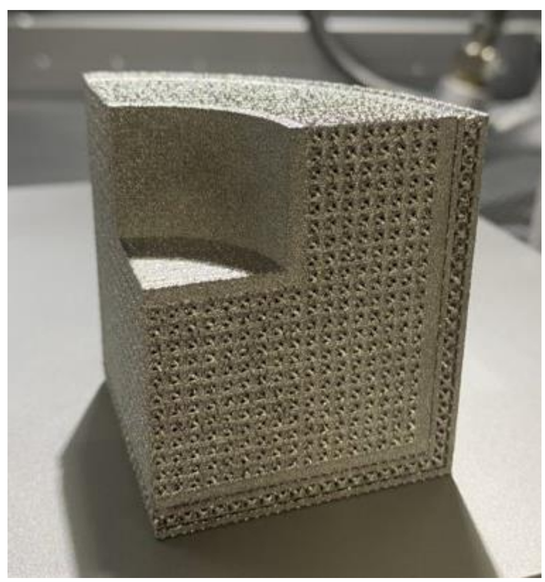



Figure #15: Final 8th Assembly Model (Left) and produced '1/8 cut' model (Right)

3. Costs

The first costs incurred for our project are the labor costs estimates relating to the time spent by each group member. The ideal starting salary for each group member is $80,000 which breaks down to $38.46 per hour for a standard work year. Each of us spent an average of 6 hours per week, which totals 342 hours from the initial start.

$$\$38.46 \text{ x } 342 \text{ hours x } 2.5 = \$32{,}883.30$$

This figure, $32,883.30 is the representative cost for all labor through the design process, prototyping process, and production process. The next costs incurred were the costs for raw material used in the production phase. The price of grade five Ti64 totals $120 / kg, while the price of Nylon PA6 totals $29.95 for one kg of filament. The costs incurred for the produced prototype totals $98.04, while a full-scale model would cost $784.35. If a full-scale model were to be produced, this would bring total project costs to $33,667.65.

Total Mass (Prototype)	
Ti64	0.72945 kg
Nylon	0.35091 kg
Total Mass (Full Model)	
Ti64	5.83562 kg
Nylon	2.80729 kg

Total Cost (Prototype)		
Ti64	$	87.53
Nylon	$	10.51
TOTAL	$	98.04
Total Cost (Full Model)		
Ti64	$	700.27
Nylon	$	84.08
TOTAL	$	784.35

4. Engineering Standards

4.1 Material Testing

During material validation testing for this project, we will be subject to a few ASTM standards. During tensile testing we will be subject to ASTM D638-14 for plastics as well as ASTM E8 / E8M-16ae1 for metals. When perform cyclic testing on plastics we will follow ASTM D7791-17 for uniaxial loadings and ASTM D7774-17 for flexural loadings. Metals will be subject to ASTM E466-15 during cyclic testing. During modal testing of the materials, we will also adhere to ASTM E756-05(2017) on the 'Standard Test Methods for Measuring Vibration-Damping Properties of Materials.'

4.2 Additive Manufacturing Standards

During the production process since we are incorporating a variety of additive manufacturing practices, there are also a multitude of standards to adhere to. The ISO/ASTM 52910:2018(E) standards outlines guidelines and recommendations for designing parts intended for additive manufacturing. When determining the geometric capabilities of the printing systems, a large part of the end stages of this project, we will be subject to ISO/ASTM 52902:2019(E) standards. When working with metal 3D Printers, and creating designs for them, we will take note of the following standards: ISO/ASTM 52911-1:2019(E), F3413-2019, and F3187-16. We will also take note of ISO/ASTM 52921:2013(E) and ISO/ASTM 52900:2015(E) which outline general accepted practices and terminology for additive manufacturing.

5. Discussions

5.1 Assumptions

As with any engineering problem, there are some key assumptions that we make to simplify the problem and make it 'easily' solvable. The largest assumption we made is that due to the symmetry of our model that it would also behave as an isotropic object. As was most likely noticed in the analysis we mostly ignored the Z-direction, which was a decision made based on this assumption.

In additive manufacturing the weakest point of a printed object usually lies in the connection between what is called 'layer lines.' The layer lines are essential the seam where each layer of material meets, and this bond is also generally quite weak under some loadings. Due to computational constraints, as well as how little is known about layer line strength, it is not practical to model these during analysis. Therefore, we must assume that our models do not have any layer lines.

5.2 Damping Model

Initially when setting up a vibration analysis in ANSYS it assumes a very small default value of damping. To properly model the damping in your system you can either give each material its own damping properties, or globally apply damping properties to the entire system. When doing this, especially when using Rayleigh Damping, it is important to use proper values, so system performance is not over calculated. During our research into proper damping values, we erred on the conservative side of values. Through some assumptions about the way Nylon PA6 behaves we found a material damping ratio of 0.07 to be acceptable for our project [6]. Titanium was a much easier material to find a proper damping ratio before since it is a rigid metal and is usually assumed to be much less than 0.01 [7]. We used a damping ratio of 0.005 for titanium. Further research led us to find that the stiffness coefficient for systems was commonly between 0.0008 and 0.064 [8]. This led us to use a conservative value of 0.001. For proper analysis, these coefficients will need to be verified using physical testing.

5.3 Topology Optimization

Near the beginning of this project, we spent time exploring the option of using what is called 'Topology Optimization' to create a design that met all requirements. What topology

optimization does is start with your initial geometry, then using criterion set forth by the user it iteratively changes the geometry until the user's criterion are met. One of the options using this kind of FEM is to generate a lattice structure that met user criterion. However, this process is very computation heavy and generates models with very fine meshes that become almost unsolvable. Appendix F contains a validation example we did to show this method can be useful. In the end we scrapped this method, due to computational power, but took the research and understanding of how it works with us.

5.4 Verification

During the material selection phase of our project, as well as when determining the damping coefficients for these materials, there is a multitude of physical testing that can be performed. If these physical tests match what our FEA software predicts then we know our computer models are accurate. These tests include tensile testing, cyclic testing, and modal testing. These tests would then be performed to the standards outlined in Chapter 4.1 of this report if the current global situation had permitted.

Plans were made at the end of this project for system performance to be tested and verified at Sandia National Labs. These tests included, but were not limited to, drop table testing and shaker table testing. These tests will be able to accurately represent the analysis environments that we have discussed throughout the body of this report. However, due to time constraints, the global pandemic, as well as manufacturing issues these tests were not performed.

6. Conclusion

6.1 Thoughts & Accomplishments

At the beginning of this project, we were three undergraduate mechanical engineering students with only an introductory vibrations course under our belt. Within a week of this paper being finalized we will be three graduated engineers with a much stronger understanding of the multitude of different types of vibrations and analyses that go along with them. In approximately five to six months, we have taken nothing and turned out, what we believe to be, a viable vibration isolation system that has been manufactured solely using 3D printing technology. As with any project we had our problems and were able to work through them. With more time there are other routes we would like to explore, such as filling the lattice structures with a viscoelastic material like Oobleck [10]. In the end we achieved good results and may not be worth our time to take the project from 95% perfect to 99% perfect.

6.2 Future of the Project

While the months and months it may take us to take this project and research to 99% perfection are not worth it, that does not mean it is not worth it for Sandia National Labs. As Sandia continues their research into meta-materials as well as into safe handling of various nuclear systems, they will be able to utilize the research that we have done. It is our hopes that the researchers at Sandia find our project satisfactory and that it is helpful in their future endeavors.

References

[1] Nuclear Weapons. (n.d.). Retrieved November 30, 2020, from https://www.sandia.gov/missions/nuclear_weapons/index.html

[2] Safety & Security. (n.d.). Retrieved November 30, 2020, from https://www.sandia.gov/missions/nuclear_weapons/safety_security.html

[3] Sisemore, C., & Skousen, T. (n.d.). *A Method for Extrapolating Haversine Shock Test Levels*. Lecture presented at 86th Shock and Vibration Symposium in Sandia National Laboratories, Albuquerque, New Mexico.

[4] Varat, M. S., & Husher, S. E. (n.d.). *Crash Pulse Modeling for Vehicle Safety Research* (Vol. 18, Enhanced Safety of Vehicles (ESV), Tech. No. 501). NHTSA.

[5] An, X., Lai, C., He, W., & Fan, H. (2019). Three-dimensional meta-truss lattice composite structures with vibration isolation performance. *Extreme Mechanics Letters, 33*. doi:https://doi.org/10.1016/j.eml.2019.100577

[6] Anand, T., & Senthilvelan, T. (2018). INVESTIGATION OF DAMPING PROPERTIES OF PA6 HYBRID NANO COMPOSITES WITH MWCNT AND COPPER NANO PARTICLES. *International Journal of Advanced Research in Engineering and Technology (IJARET), 9*(3), 194-199. doi:10.1007/978-981-15-4745-4_24

[7] Viscous damping ratios for different systems and materials. (n.d.). Retrieved April 06, 2021, from http://help.solidworks.com/2016/english/solidworks/cworks/r_viscous_damping_ratios.htm

[8] Song, Z., & Su, C. (2017). Computation of rayleigh damping coefficients for the seismic analysis of a hydro-powerhouse. *Shock and Vibration, 2017*, 1-11. doi:10.1155/2017/2046345

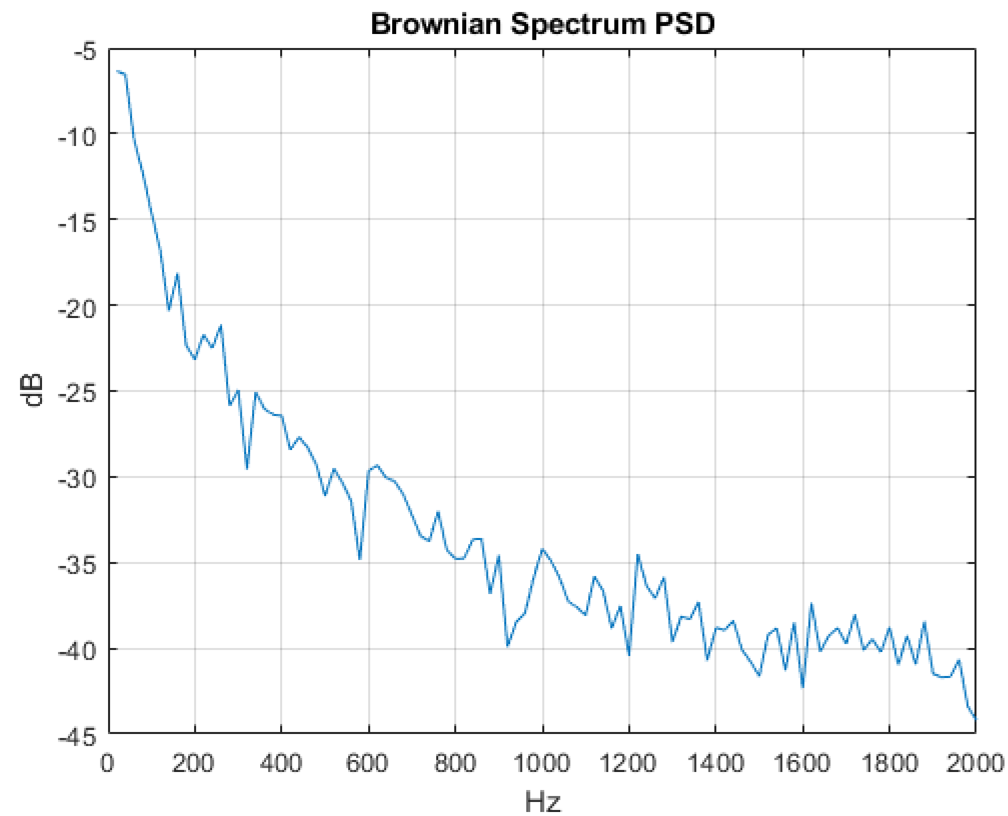
[9] Arch bridge. (n.d.). Retrieved January 21, 2021, from https://www.britannica.com/technology/arch-bridge

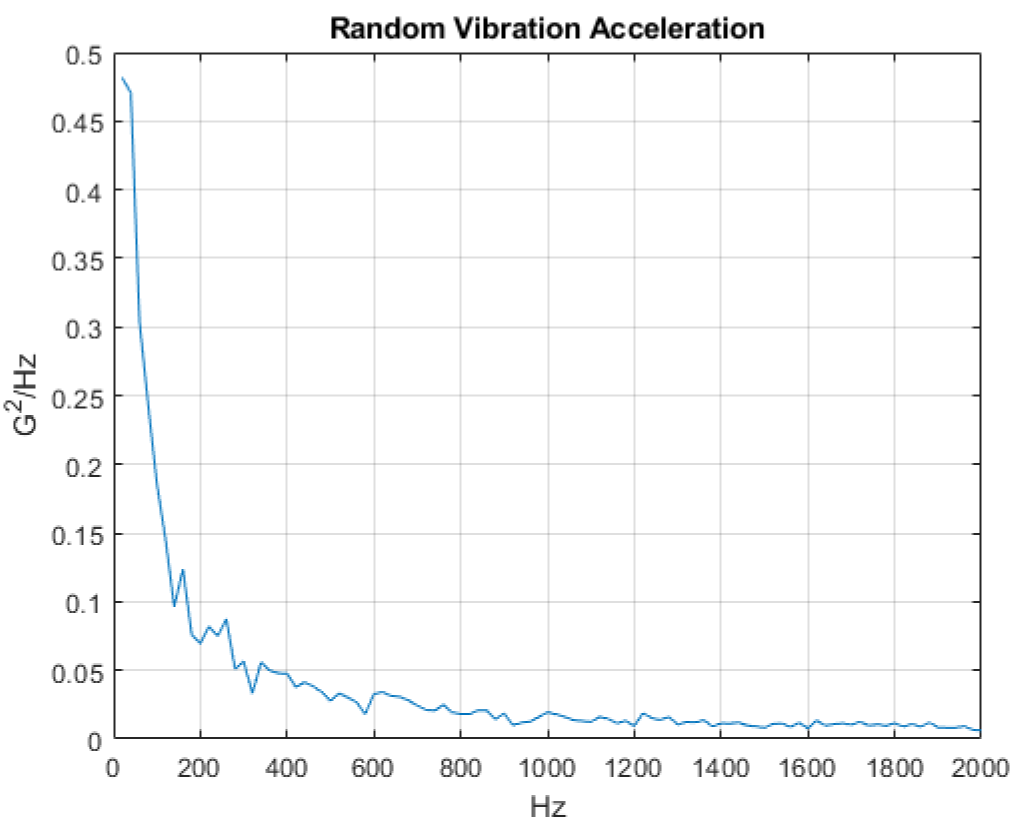
[10] Wang, R., Shang, J., Li, X., Luo, Z., & Wu, W. (2018). Vibration and damping characteristics of 3D printed Kagome lattice with viscoelastic material filling. *Scientific Reports, 8*. doi:10.1038/s41598-018-27963-4

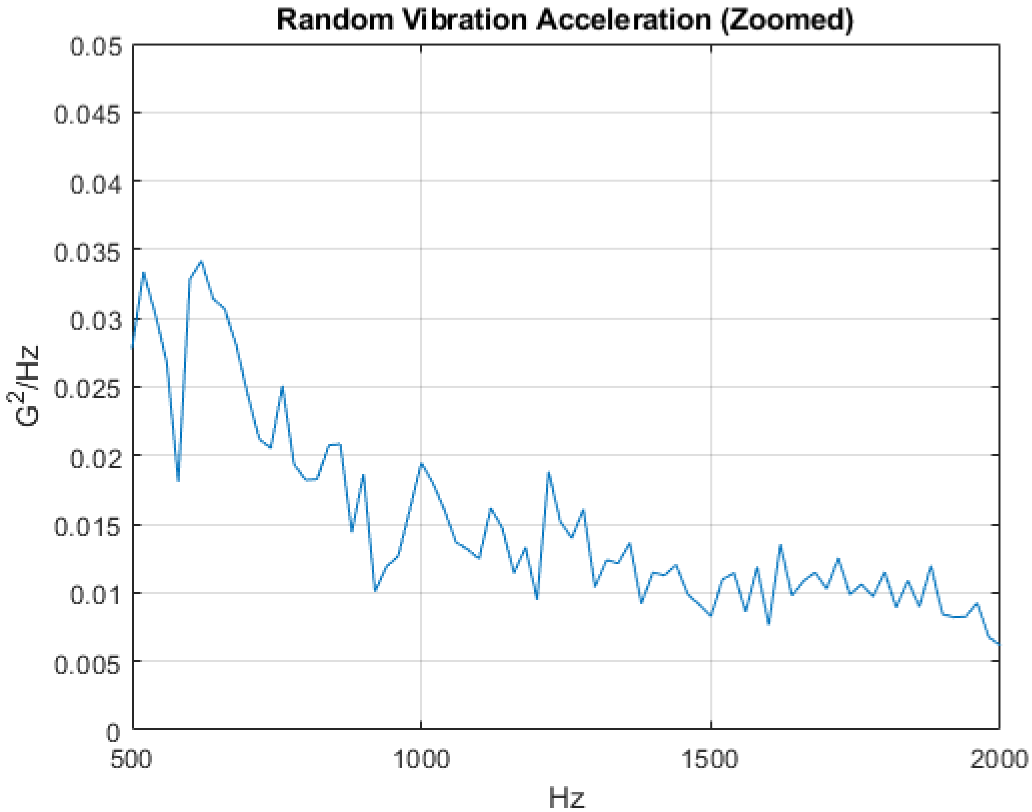
Appendix A: Brownian Noise

Generates a Brownian Random Vibration Acceleration Spectrum Outputs all values to an Excel File Made for use in ANSYS Random Vibration Analysis

```
clear all
clc

Fs = 4000; %Sampling Frequency

rng default
BrownNoise = dsp.ColoredNoise('Color','brown','SamplesPerFrame',5000);
x = BrownNoise();

[Pxx,F] = pwelch(x,[],[],200,Fs,'onesided','psd');

figure(1)
plot((F(2:end)),10*log10(Pxx(2:end)))
title('Brownian Spectrum PSD')
xlabel('Hz')
ylabel('dB')
grid on

figure(2)
plot((F(2:end)),sqrt(Pxx(2:end)))
title('Random Vibration Acceleration')
xlabel('Hz')
ylabel('G^2/Hz')
grid on

figure(3)
plot((F(2:end)),sqrt(Pxx(2:end)))
title('Random Vibration Acceleration (Zoomed)')
xlabel('Hz')
ylabel('G^2/Hz')
ylim([0 0.05])
xlim([500 2000])
grid on

filename = 'BrownianSpectrum.xlsx';
recycle on
delete(filename);
writematrix(F(2:end),filename,'Sheet',1,'Range','A1')
writematrix(sqrt(Pxx(2:end)),filename,'Sheet',1,'Range','B1')
```


Brownian Spectrum PSD
dB
Hz
-5
-10
-15
-20
-25
-30
-35
-40
-45
0
200
400
600
800
1000
1200
1400
1600
1800
2000


Random Vibration Acceleration
G^2/Hz
Hz
0.5
0.45
0.4
0.35
0.3
0.25
0.2
0.15
0.1
0.05
0
0
200
400
600
800
1000
1200
1400
1600
1800
2000

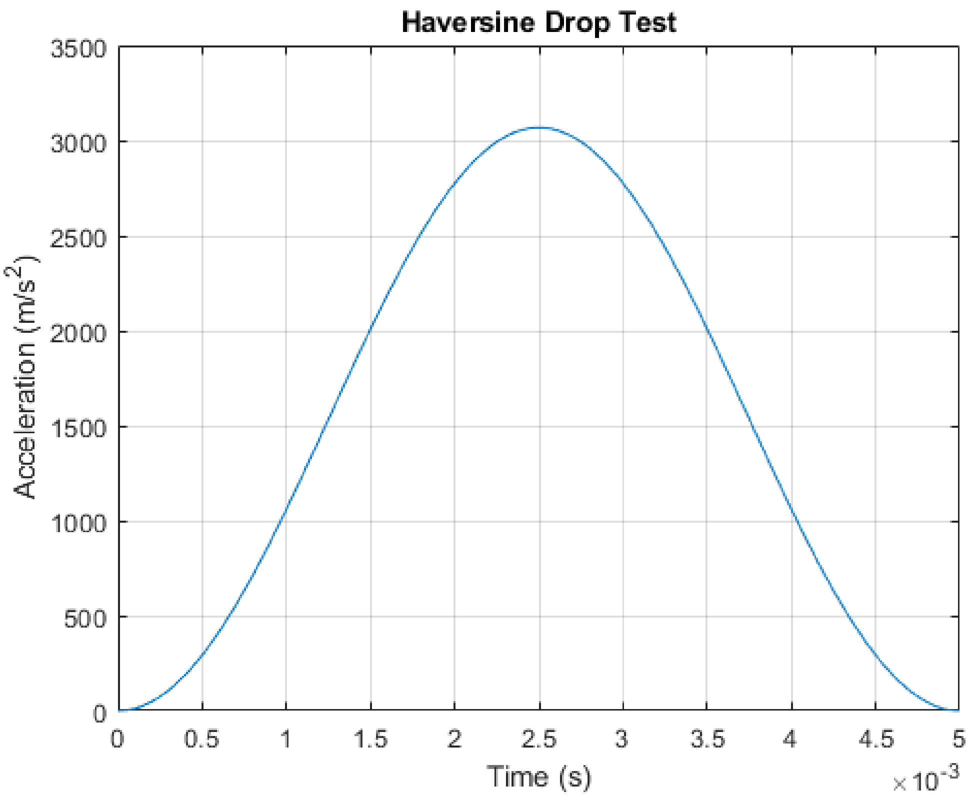


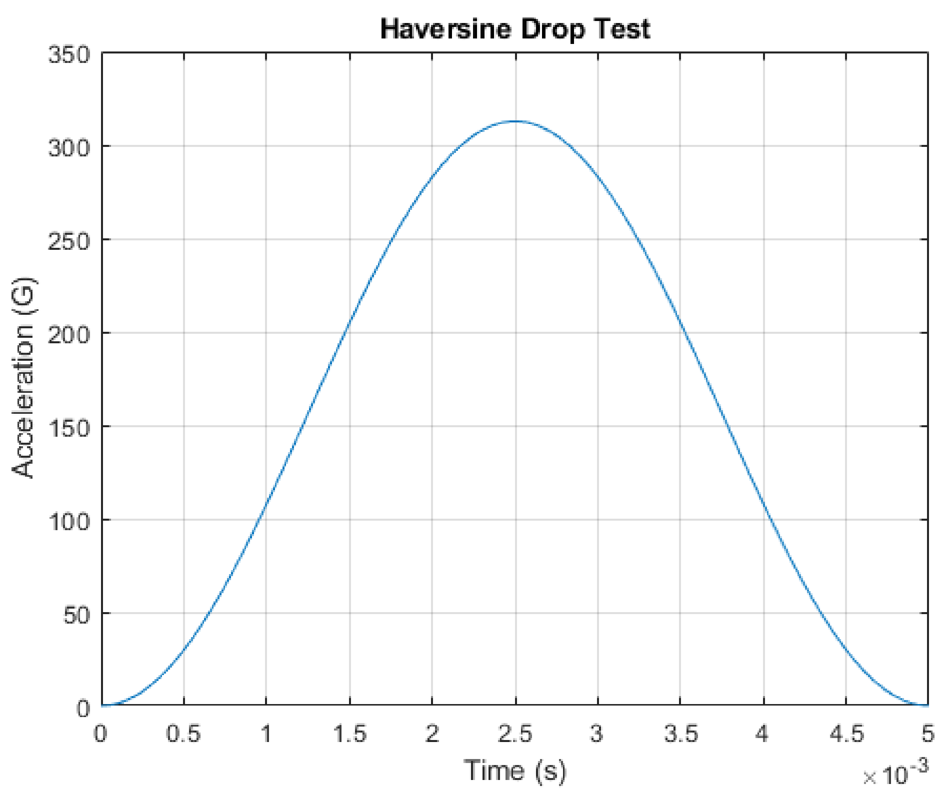
Published with MATLAB® R2020b

Appendix B: Haversine Drop Testing

Generates a Haversine Acceleration Shock Wave Outputs all values to an Excel File Made for use in ANSYS Transient Structural Analysis

```
clear all
clc

%Mass of what is being tested
mass = 12; %kg

h = 3; %drop height, m
g = 9.81; %garvity, m/s^2
T = 0.005; %pulse time duration
t = linspace(0,T,100);

Impact = sqrt(2*g*h); %Impact Velocity, m/s
P = (2*Impact)/T;
A = P.*(sin((pi.*t)/T).^2);

figure(1)
plot(t,A)
xlabel('Time (s)')
ylabel('Acceleration (m/s^2)')
title('Haversine Drop Test')
grid on

figure(2)
plot(t,A./9.81)
xlabel('Time (s)')
ylabel('Acceleration (G)')
title('Haversine Drop Test')
grid on

filename = 'HaversineLoading.xlsx';
recycle on
delete(filename);
writematrix(t(1:end)',filename,'Sheet',1,'Range','A1')
writematrix(A(1:end)',filename,'Sheet',1,'Range','B1')
```





Published with MATLAB® R2020b

Appendix C: Transient Analysis

Performs polynomial fitting of deformation output from ANSYS. Then takes derivatives to find max acceleration Payload faces during Shock Loading.

Command Cleanup

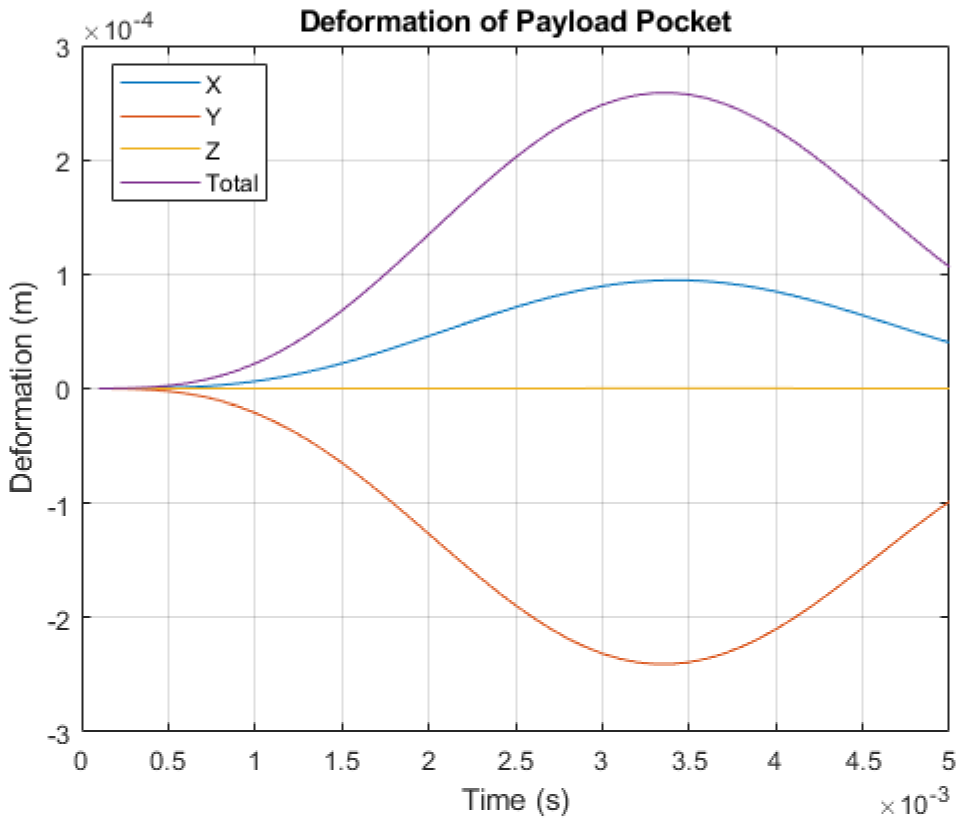
```
clear all
clc
set(0,'DefaultLegendAutoUpdate','off')
```

Importation of Data

```
ExcelFile = 'TransientData.xlsx';
t = xlsread(ExcelFile,1,'A1:A50');
x = xlsread(ExcelFile,1,'B1:B50');
y = xlsread(ExcelFile,1,'C1:C50');
z = xlsread(ExcelFile,1,'D1:D50');
total = xlsread(ExcelFile,1,'E1:E50');
```

Curve Fitting

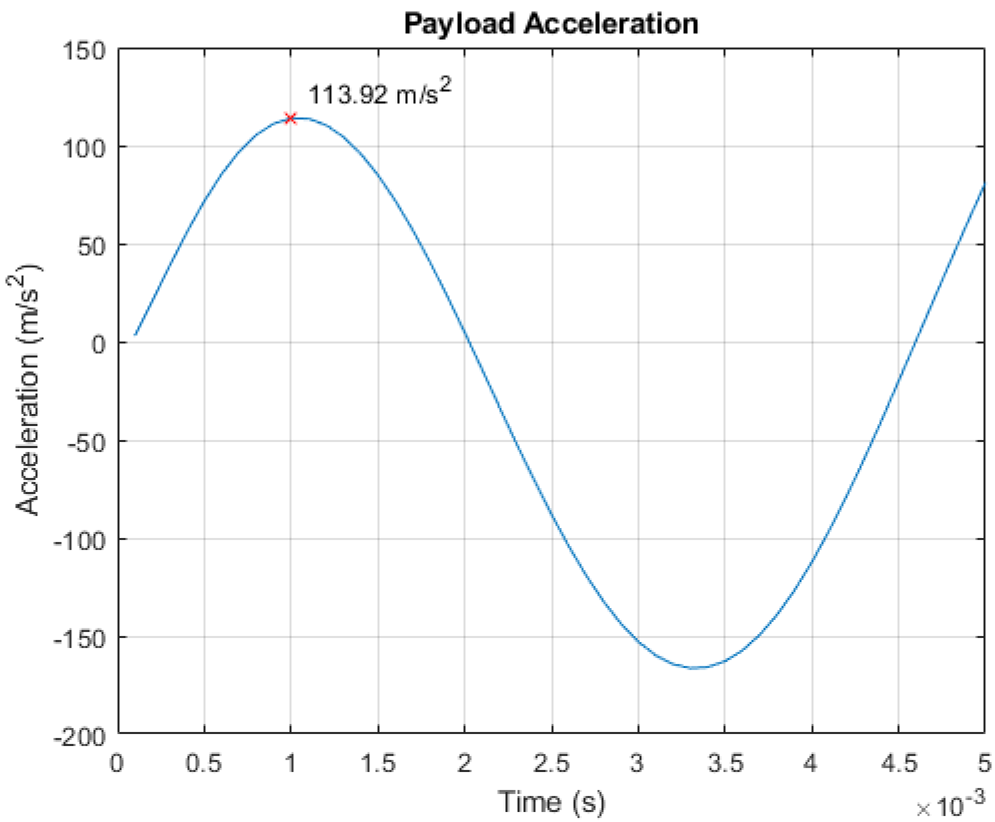
```
xfit = polyfit(t,x,10);
yfit = polyfit(t,y,10);
zfit = polyfit(t,z,10);
totalfit = polyfit(t,total,10);

xval = polyval(xfit,t);
yval = polyval(yfit,t);
zval = polyval(zfit,t);
totalval = polyval(totalfit,t);

figure(1)
plot(t,xval,t,yval,t,zval,t,totalval)
legend('X','Y','Z','Total','Location','Best')
ylabel('Deformation (m)')
xlabel('Time (s)')
title('Deformation of Payload Pocket')
grid on
```

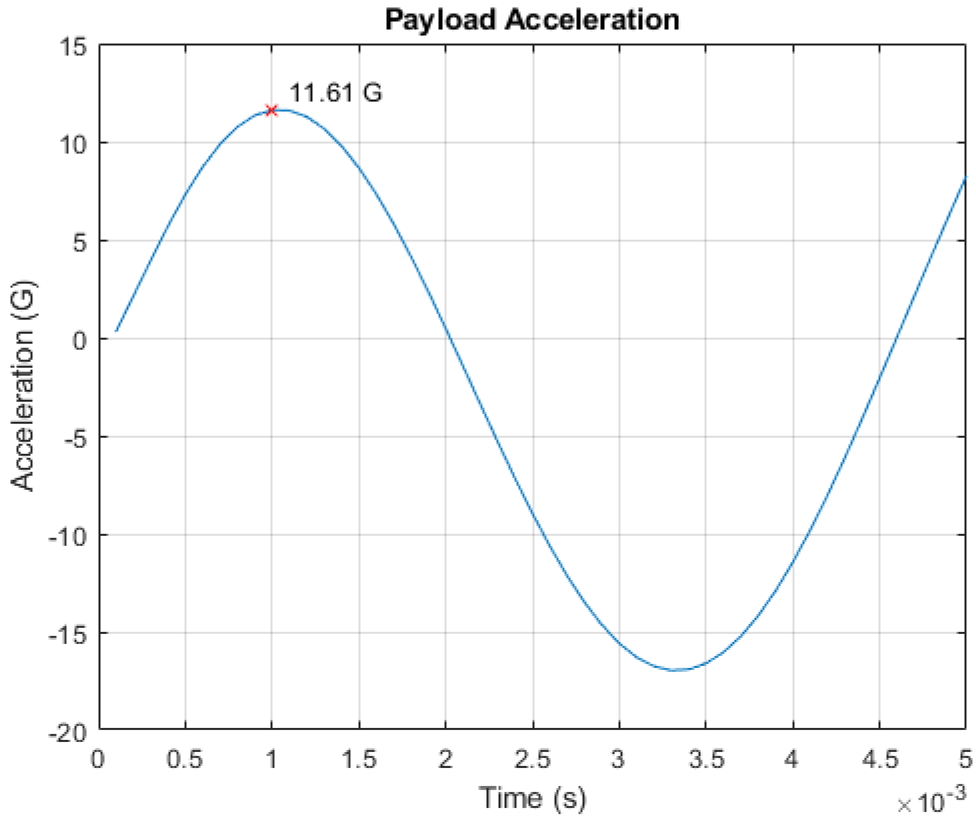


Total Acceleration

```
TotalVelocity = polyder(totalfit);
TotalAccel = polyder(TotalVelocity);

AccelVal = polyval(TotalAccel,t);
AccelG = AccelVal/9.81;
[~,imax] = max(AccelVal);

textms = sprintf('%4.2f m/s^2',AccelVal(imax));
figure(2)
plot(t,AccelVal)
hold on
plot(t(imax),AccelVal(imax),'xr')
text(t(imax+1),AccelVal(imax)+15,textms)
title('Payload Acceleration')
xlabel('Time (s)')
ylabel('Acceleration (m/s^2)')
hold off
grid on

textG = sprintf('%4.2f G',AccelG(imax));
```

```
figure(3)
plot(t,AccelG)
hold on
plot(t(imax),AccelG(imax),'xr')
text(t(imax+1),AccelG(imax)+1,textG)
title('Payload Acceleration')
xlabel('Time (s)')
ylabel('Acceleration (G)')
hold off
grid on
```



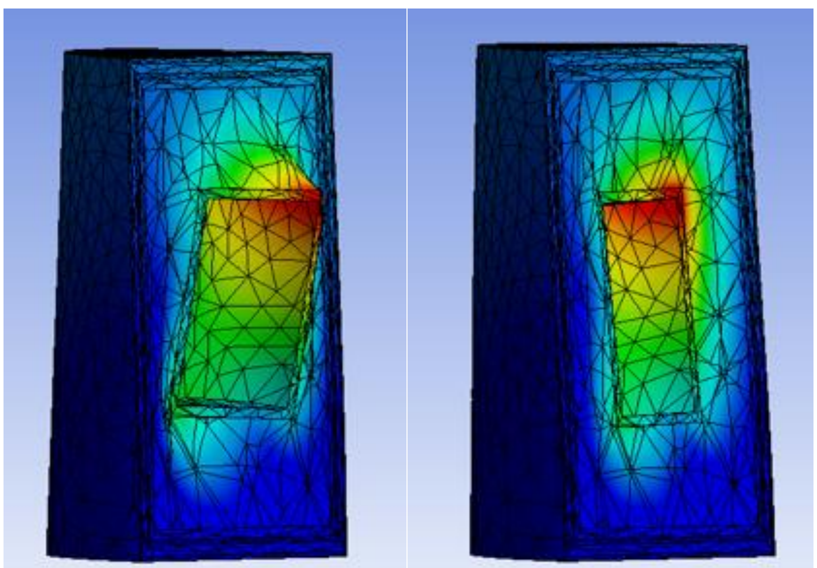


Published with MATLAB® R2020b

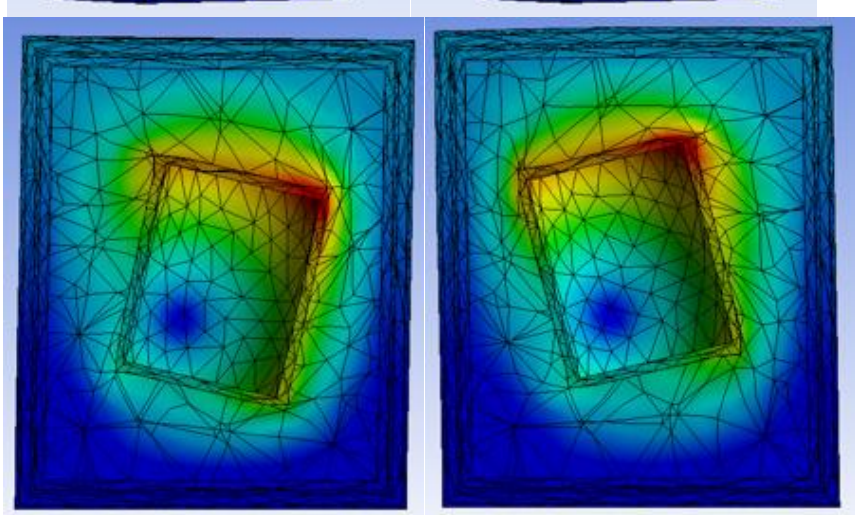
Appendix D: Modal Results

	Mode	Frequency [Hz]
1	1.	346.22
2	2.	370.13
3	3.	848.87
4	4.	1203.3
5	5.	1327.4
6	6.	2244.7
7	7.	2307.4
8	8.	2447.2
9	9.	2507.7
10	10.	2582.8
11	11.	2661.2
12	12.	2898.1
13	13.	2912.2
14	14.	2943.7
15	15.	3742.9
16	16.	3837.2
17	17.	3872.8
18	18.	3923.6
19	19.	3997.4

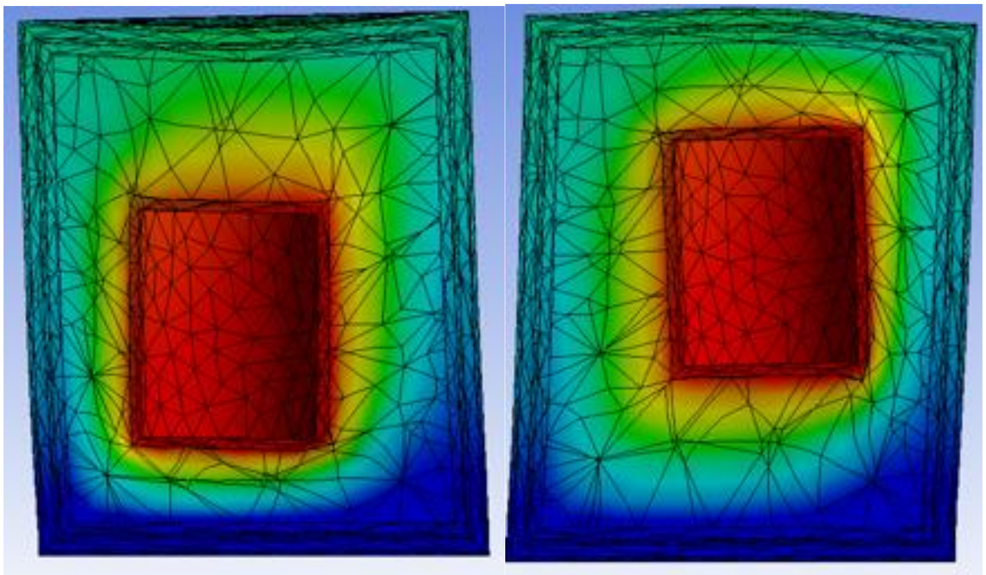
Resonant Frequencies calculated to sampling frequency of 4000 Hz



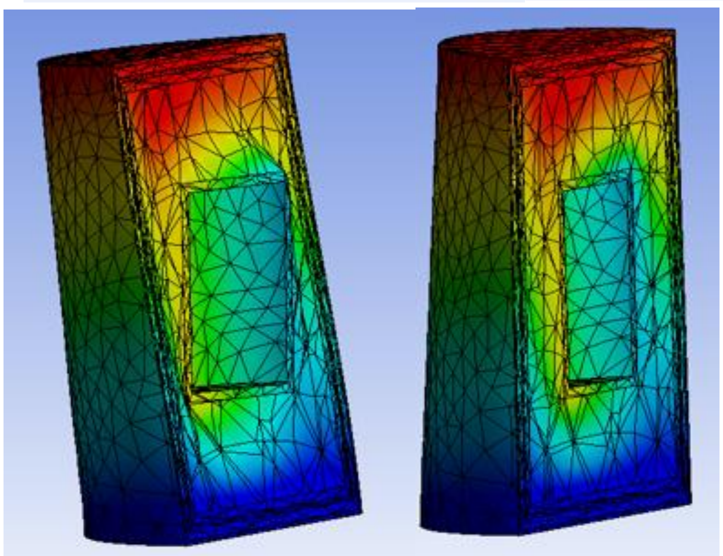
First Mode of Deformation



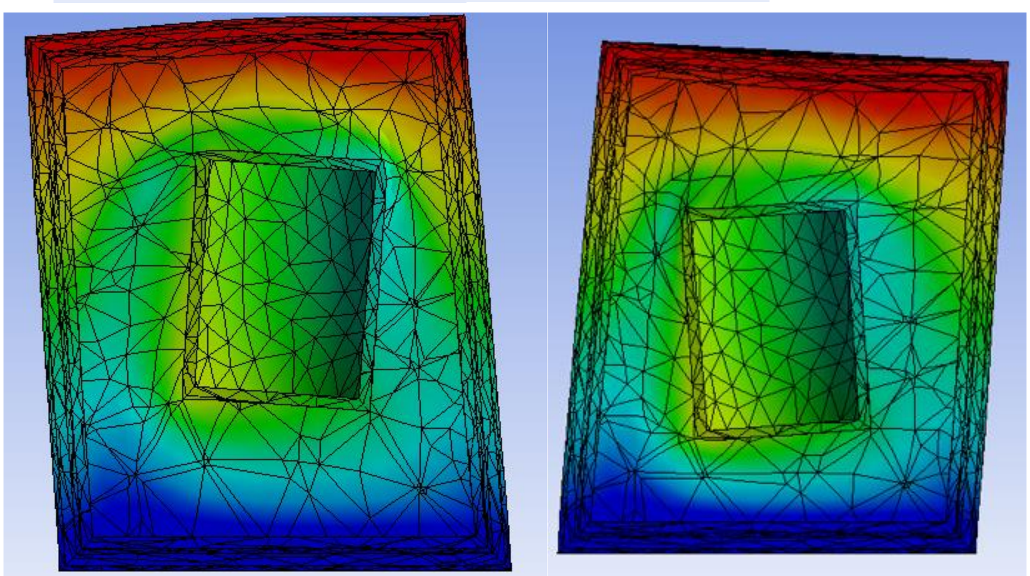
Second Mode of Deformation



Third Mode of Deformation

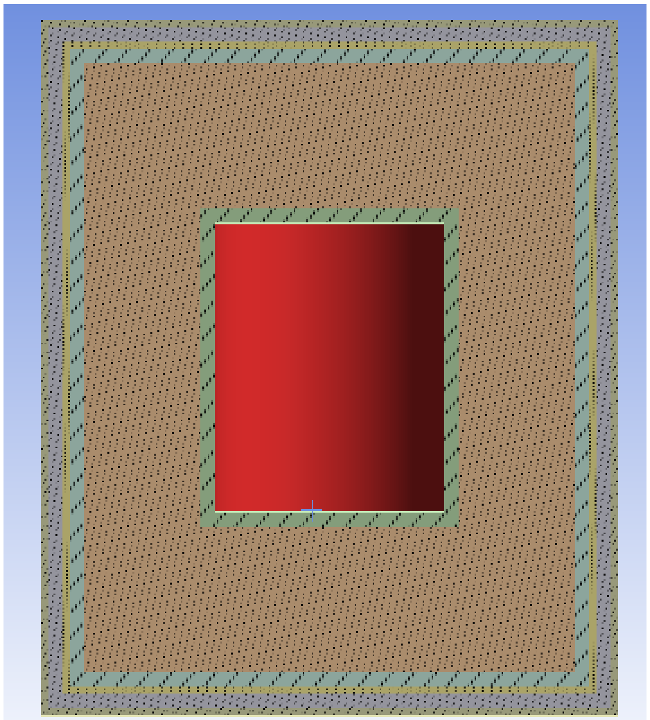


Fourth Mode of Deformation

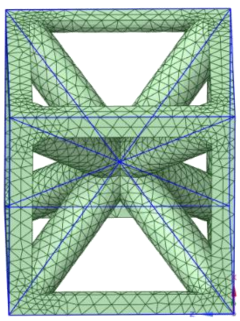


Fifth Mode of Deformation

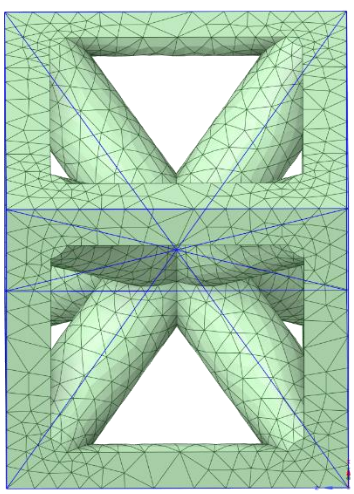
Appendix E: Final Model



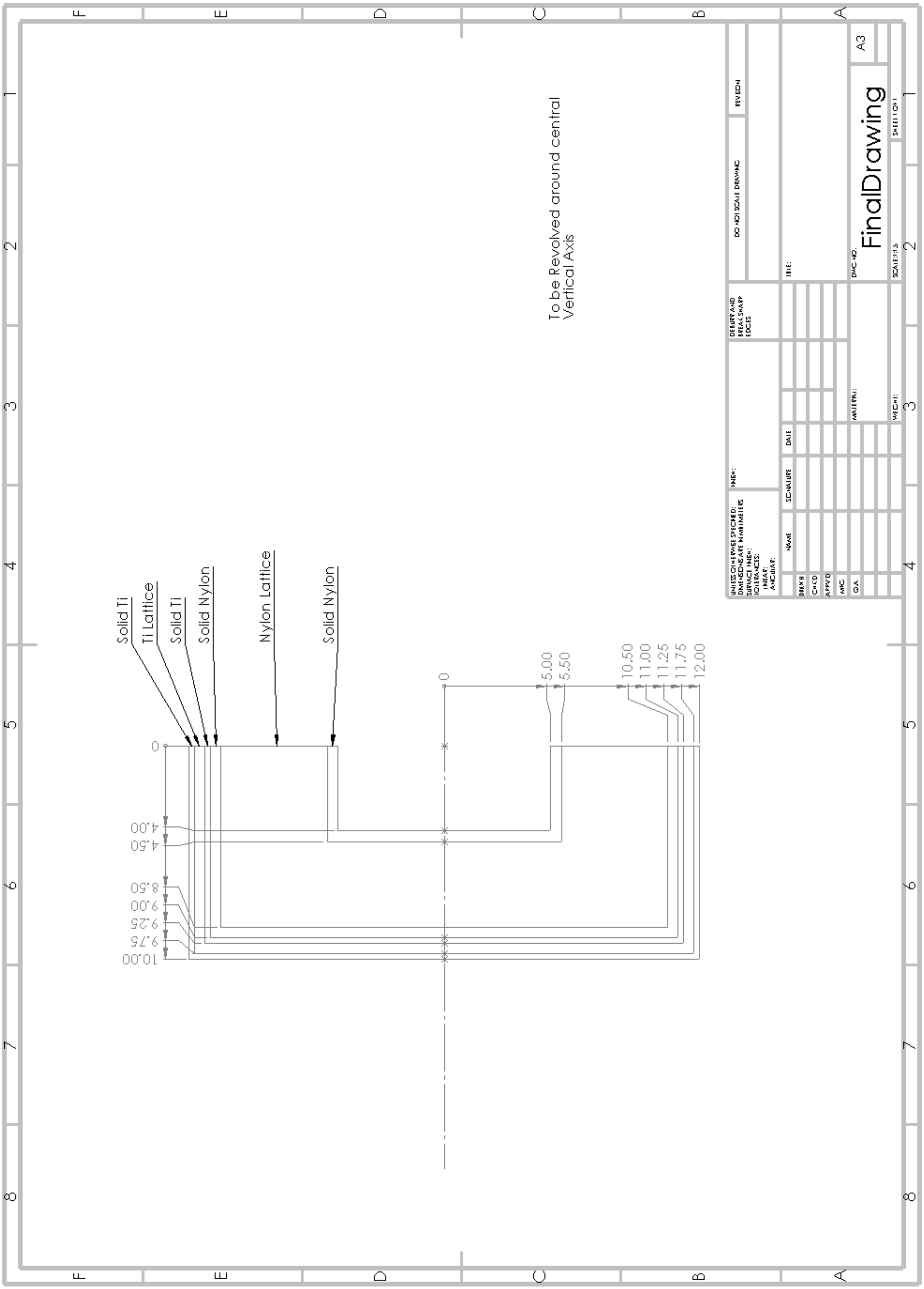
Half-Cut Section Plane view of Finalized Model



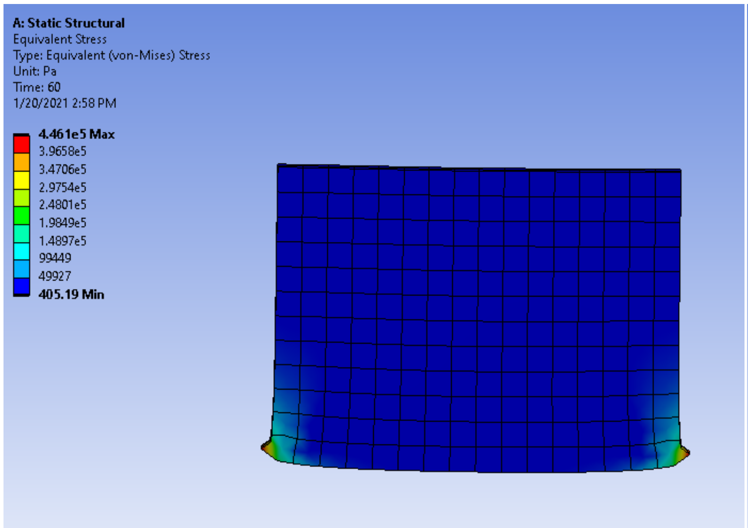
Ti Lattice Unit Cell: Cubic w/Center Support, 2.5mm Cell Size, 0.25 Volume Fraction

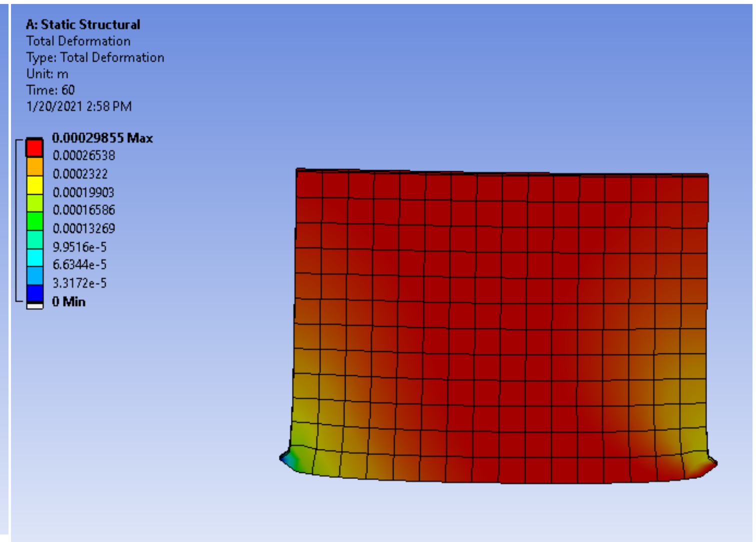


Nylon (PA6) Lattice Unit Cell: Cubic w/Center Support, 5mm Cell Size, 0.375 Volume Fraction

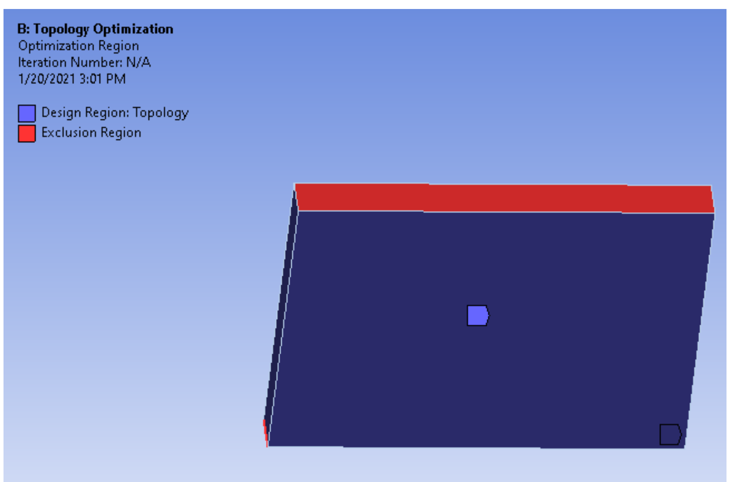

To be Revolved around central
Vertical Axis
Solid Ti
Ti Lattice
Solid Ti
Solid Nylon
Nylon Lattice
Solid Nylon
0
4.00
4.50
8.50
9.00
9.25
9.75
10.00
0
5.00
5.50
10.50
11.00
11.25
11.75
12.00
FinalDrawing
A3

Appendix F: Topology Optimization Validation Case





We start with a statically loaded simple support beam.



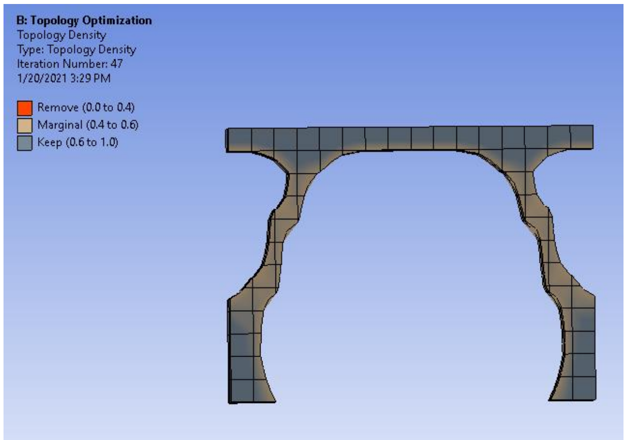
Details of "Response Constraint"

Scope	
Scoping Method	Optimization Region
Optimization Region Selection	Optimization Region
Definition	
Type	Response Constraint
Response	Global von-Mises Stress
Maximum	2.5e+005 Pa
Environment Selection	All Static Structural
Suppressed	No

Details of "Response Constraint 2"

Scope	
Scoping Method	Optimization Region
Optimization Region Selection	Optimization Region
Definition	
Type	Response Constraint
Response	Mass
Define By	Range
Percent to Retain (Min)	10 %
Percent to Retain (Max)	20 %
Suppressed	No

Then we setup the topology optimization and all required constraints.



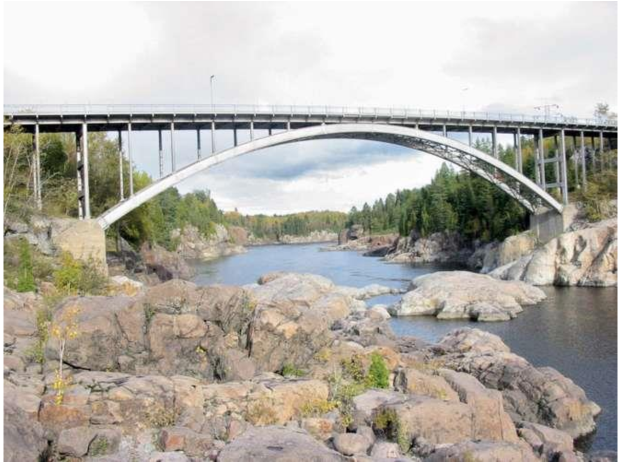
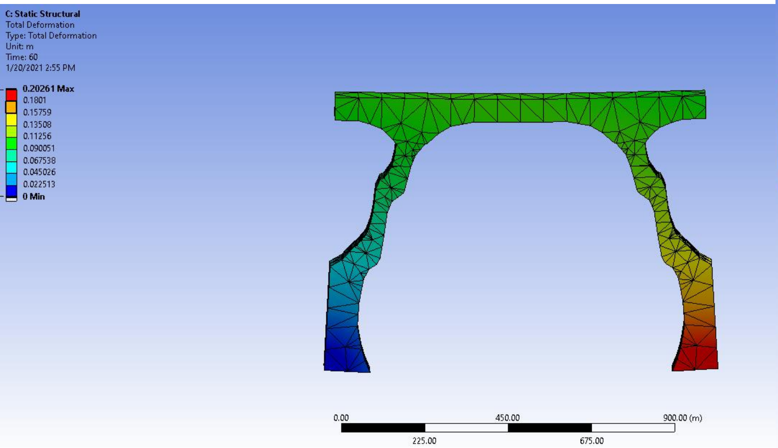
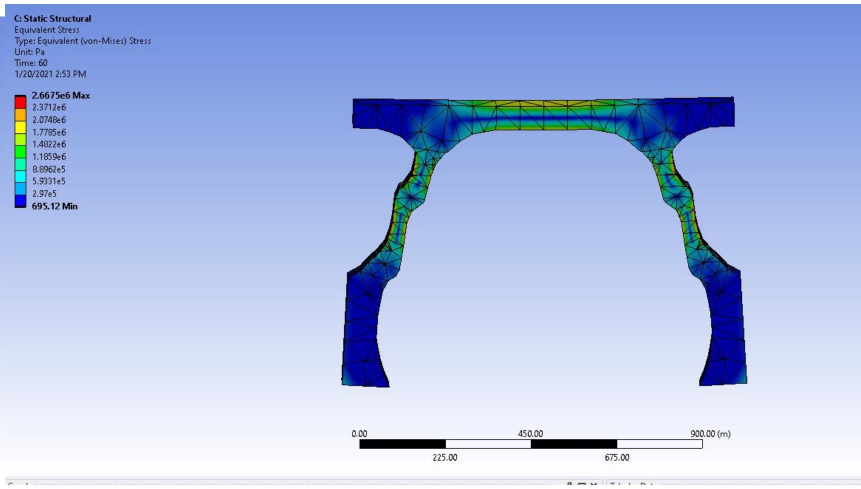


Photo Courtesy of [9]

And the geometry that is spit out mimic's real-world designs being used today.





Performing the same static analysis on the new geometry shows that the stress is close to the maximum we set in our criterion. These discrepancies in values are most likely caused by differences in meshes.