Second Progress Report: Finite Element Analysis of the Third Generation Advanced LIGO Mirror Suspension Systems

Joy Westland, LIGO-T1600293

July 29, 2016

1 Progress: Attempted Calculation of Strain Energy

In the past month, work has been made towards checking the accuracy of the program ANSYS. Simpler models are created in order to compare the analytical results to the ANSYS generated results. For example, due to the amount of errors made by the huge difference between a 400 micron wire and a 10 kg mass, a cantilever bar was made in order to apply a constant force to different sections of the bar (errors explained in Problems Encountered). This model was a clamped bar at one end where a force was applied to different increments on the bar. A plot of the maximum and minimum strain energies as a function of distance along the bar was made. There were two individual bars that each had a different cross section: cylindrical and rectangular. In Figure 1 and Figure 2, a rectangular bar and cylindrical bar were split into 20 sections, each section was 5 m long.

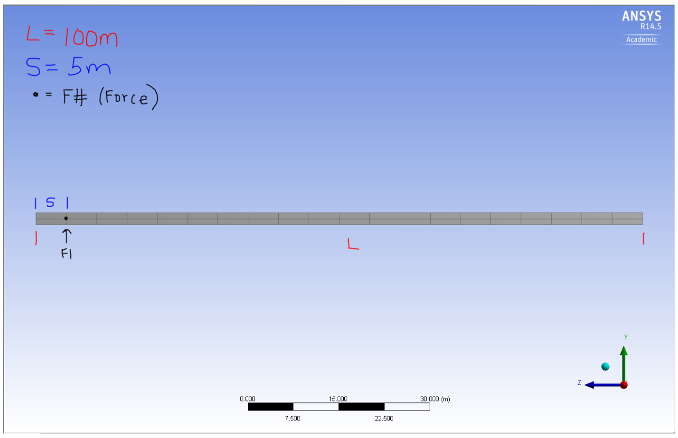


Figure 1: This is a rectangular cantilever bar that has been divided into 20 sections. Each section is 5 m long for a total of 100 m. The cross section is 2 m in diameter.

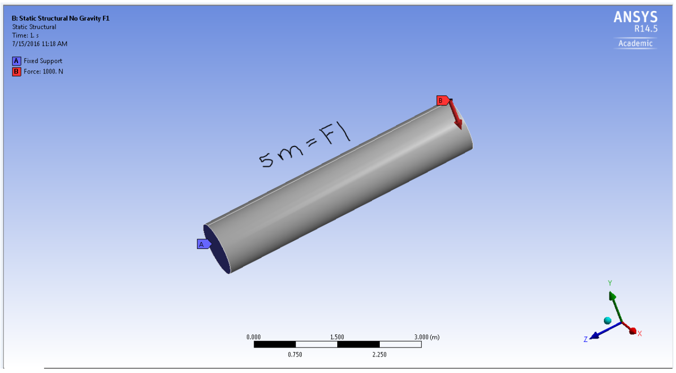


Figure 2: This is a section of the cylindrical cantilever bar where each section was added in each trial. Each section is 5 m long for a total of 100 m. The cross section is 2 m x 2 m square.

The maximum and minimum strain energies of the bar without gravity were calculated. In Figure 3, and Figure 4, the rectangular cross section and cylindrical cross section of the maximum strain energies were graphed.

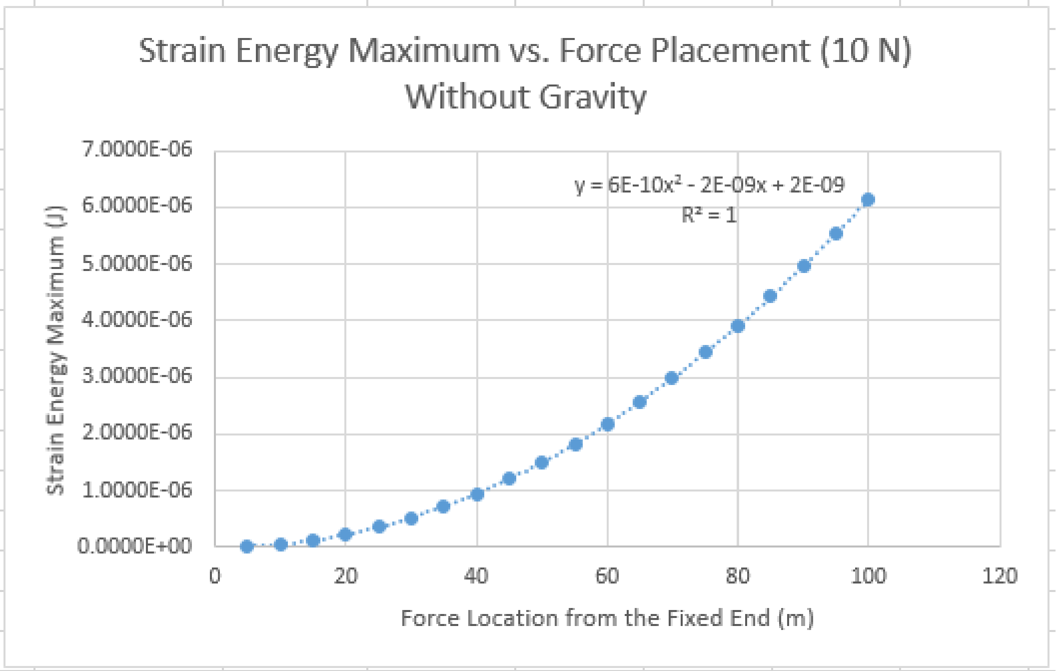


Figure 3: These are the maximum strain energy calculations for a cantilever rectangular bar. This is a rectangular cantilever bar that has been divided into 20 sections. Each section is 5 m long for a total of 100 m. The cross section is a 2 m x 2 m square.

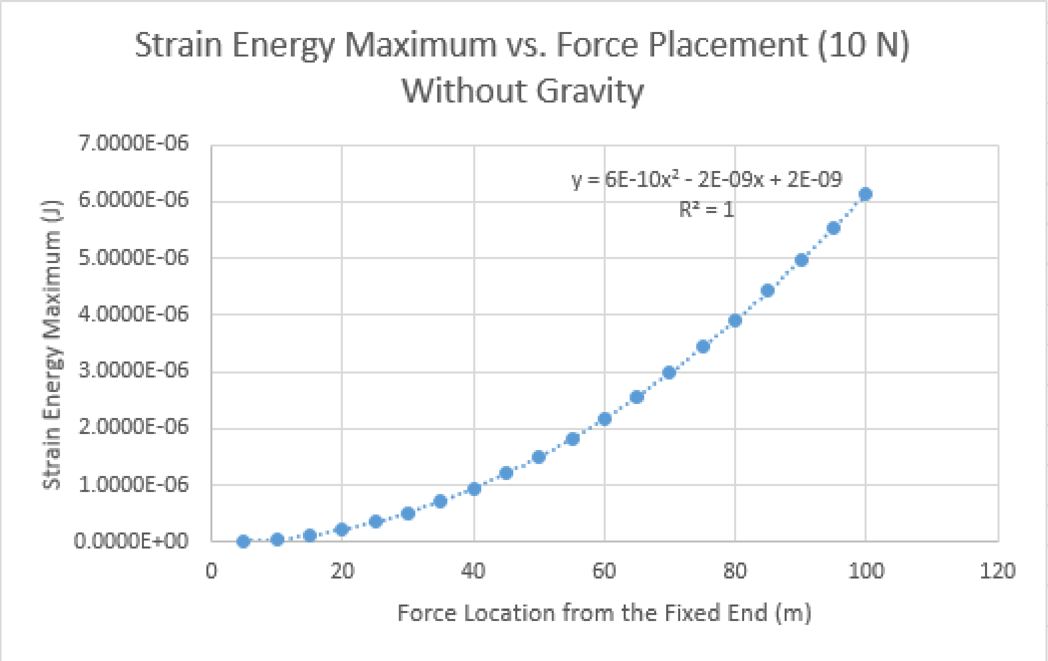


Figure 4: These are the maximum strain energy calculations for a cantilever cylinder bar. This is a cylindrical cantilever bar that has been divided into 20 sections. Each section is 5 m long for a total of 100 m. The cross section is 5 m in diameter.

In both cases, the measurements taken were not quite the right direction that the research should go towards. This is because, the force that will be applied to the face of a test mass will not be a point force but a force with a Gaussian profile. In this case, a new direction was made to work towards smaller steps to test ANSYS. The goal is to apply a transfer function and allow ANSYS to calculate the strain energy associated with that force.

2 Progress: Importing SolidWorks Geometry of the Test Masses, Fibers, and Ears

The first two stages of the mirror suspension system have already been modeled in SolidWorks. This includes both test masses, fibers, connections, and ears. The purpose of using SolidWorks is because SolidWorks allows a user to make a variety of models relatively easily compared to using the ANSYS interface. Due to the complexity of the model, a simpler model was made from only one test mass, the fibers, and the ears in order to increase ANSYS processing efficiency. When using SolidWorks, a user is able to import the drawing into the geometry of ANSYS. From there, a user can change the model directly in ANSYS by adding parts. With a simpler model, ANSYS will be able to run each calculation faster as well as converge easier. Figure 5 shows the lower model of the suspension system.

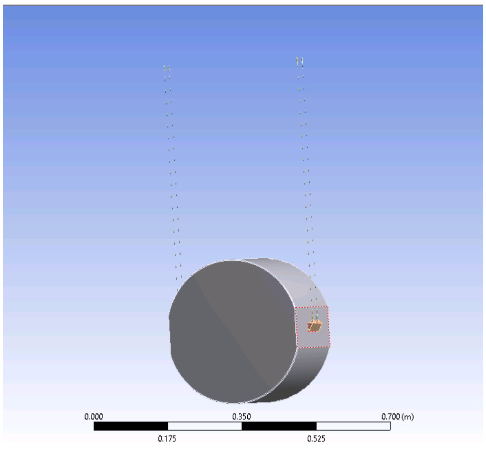


Figure 5: This is a model of the bottom test mass, the fibers, and the ears.

Even though the model is simpler than the full suspension system, when looking at the violin modes and frequencies, a simple pendulum was used. This is because the equation used to calculate the violin mode of a simple pendulum models the wire to be uniform throughout. In the SolidWorks model, the wires are shaped to be the dimensions of the actual fibers used in production where there is tapering at the top and bottom of different lengths and diameters.

3 Progress: Calculating Violin Modes and Frequencies

A simple pendulum model was used to test the accuracy of ANSYS when analytically and computationally calculating the violin modes and frequencies. There were two individual pendulum models, one wire had a diameter of 400 microns and the other wire had a diameter of 800 microns. The suspending mass on each wire was 10 kg. The length of each wire was 0.6 m long, shown in Figure 6.

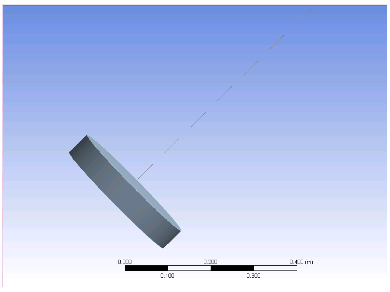
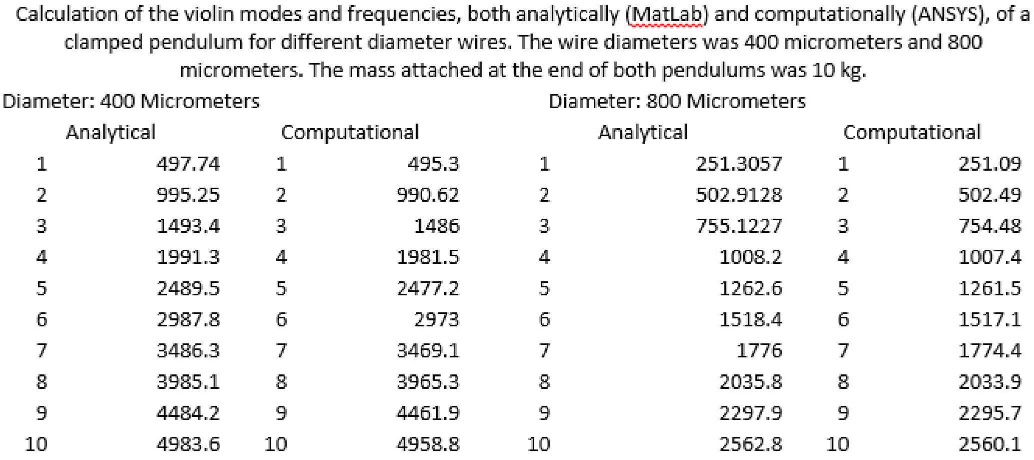


Figure 6: Simple pendulum model where the wire diameter is 400 microns with a length of 0.6 m and an attached mass of 10 kg.

The reason behind starting with a simple pendulum model was to check how ANSYS calculated the violin modes for a uniform wire. The complexity of the full suspension system increases processing power and in order to not waste time, smaller steps have been made to check that ANSYS is taking account of all factors. The actual fibers for the suspension system are tapered at the ends with varying diameters, which causes difficulty in analytically solving the violin modes by hand. By using a uniform fiber and analytically calculating the violin modes, the results can be compared reasonably to ANSYS in order to ensure that ANSYS in accurate. This will increase confidence in the program when using Finite Element Analysis (FEA) for the larger suspension system. The analytical equation for solving the violin frequencies was derived by Willems et al. (2002) which describes: $f_n = \frac{n}{(2l)}\sqrt{\frac{T}{\rho_l}}[1 + \frac{2}{l}\sqrt{\frac{EI}{T}} + \frac{EI}{2T}(\frac{n\pi}{l})^2]$, where n is the nth violin mode, T is the tension in the wire, l is the length of the wire, I is the moment of cross section, E is Young's Modulus of Elasticity, and ρ_l is the linear density.

For each different diameter of the wire, MATLAB was used to analytically solve for each violin frequency, shown in Figure 7.



Calculation of the violin modes and frequencies, both analytically (MatLab) and computationally (ANSYS), of a clamped pendulum for different diameter wires. The wire diameters was 400 micrometers and 800 micrometers. The mass attached at the end of both pendulums was 10 kg.

Diameter: 400 Micrometers				Diameter: 800 Micrometers			
	Analytical		Computational		Analytical		Computational
1	497.74	1	495.3	1	251.3057	1	251.09
2	995.25	2	990.62	2	502.9128	2	502.49
3	1493.4	3	1486	3	755.1227	3	754.48
4	1991.3	4	1981.5	4	1008.2	4	1007.4
5	2489.5	5	2477.2	5	1262.6	5	1261.5
6	2987.8	6	2973	6	1518.4	6	1517.1
7	3486.3	7	3469.1	7	1776	7	1774.4
8	3985.1	8	3965.3	8	2035.8	8	2033.9
9	4484.2	9	4461.9	9	2297.9	9	2295.7
10	4983.6	10	4958.8	10	2562.8	10	2560.1

Figure 7: The analytical and computational comparison between the violin modes and frequencies of a beam in tension.

The results from the analytical solution were within 0.5% or below compared to the computational solution. These results increases the confidence in how ANSYS will accurately calculate the violin modes of a structure. By understanding how ANSYS works with the smaller scaled model, it will allow the acceptance of computational results for more complex models. Once a model gets too complicated, ANSYS is used for FEA modeling because analytically the calculations would be very difficult. The computational results of both the 400 micron and the 800 micron were graphed and can be seen in Figure 8.

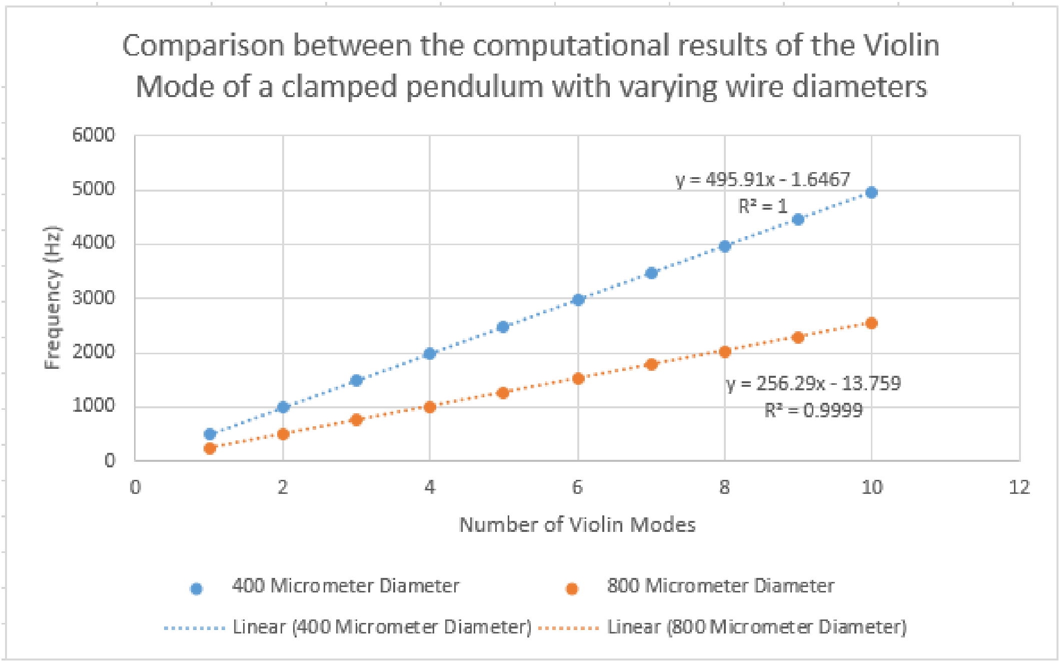


Figure 8: A graph of the computational comparison between the violin modes and frequencies of a simple pendulum model for a 400 micron diameter wire and a 800 micron diameter wire.

As the diameter increased from 400 micron to 800 micron, the frequencies decreased as the violin modes increased.

With this confirmation of the violin modes, then further study will go towards increasing the complexity of the model from a simple pendulum to the bottom stage of the mirror suspension system. The tapered thickness of the actual fibers can be analyzed using ANSYS through FEA. This will then be compared to other experimental ANSYS results in different studies in order to get a sense of using ANSYS towards modeling the full suspension system.

4 Problems Encountered

The problems that were encountered during the simple modeling of pendulums and suspension systems had to do with convergence and boundary conditions when applying a force to the side of the mass. This may be due to the fact when there is a huge difference between one fiber and the mass that is attached. For example, a 400 micron fiber connected to a 10 kg mass did not converge properly and had multiple distortion issues if there was a force that was applied to the side of the face.

Due to the pendulum being complicated, the time it takes for ANSYS to run the FEA analysis increases with each part that is added. If a model were to run and not converge properly, the time used up would be wasted. Also the program does not warn a user until it has run through the process that a setting should have been turned on.

Another problem was figuring out how to apply a Gaussian Force to a surface. There has been progress made to work towards this goal by working with more experienced ANSYS users.

5 Remaining Research Goals

The main goals are to apply a Gaussian Force to one of the faces of the test masses and look at the strain energies associated with the force. Another goal is to apply an oscillating force at different frequencies. More sanity checks will be made based on prior calculations in different papers to ensure that ANSYS is being used properly. The basis of developing the suspension models and ultimately analyze cryogenically frozen mirrors, will depend on the earlier stages of the simpler models. This work will pave the way for future projects in developing the full fledged suspension system.

6 References

Willems, P., Sannibale, V., Weel, J., & Mitrofanov, V. (2002). Investigations of the dynamics and mechanical dissipation of a fused silica suspension. Physics Letters A, 297(1-2), 37-48. doi:10.1016/s0375-9601(02)00380-8