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

Joy Westland, LIGO-T1600293

August 3, 2016

1 Introduction to the Laser Interferometer Gravitational-Wave Observatory Advancement

In 1916, Albert Einstein predicted the existence of gravitational waves from the field equations of general relativity. Gravitational waves travel at the speed of light as ripples in the curvature of spacetime (Aston et al., 2012). Gravitational waves are a strain in space-time caused by accelerating masses. This can be seen as 'waves' of distorted space being radiated by the source (What are Gravitational Waves?, 2016). The U.S. Laser Interferometer Gravitational-Wave Observatory, LIGO, has two facilities located in Hanford, Washington and Livingston, Louisiana. The process of detecting gravitational waves has been the main goal for the LIGO facilities. Enhanced Michelson interferometers are used to detect gravitational wave amplitudes. The detectors are dual recycled Michelson interferometers with Fabry-Perot cavities in the arms (Arrain and Mueller, 2008). After upgrading the equipment, advanced LIGO was designed to increase the sensitivity of the LIGO detectors and decrease the amount of thermal noise. From the initial LIGO (iLIGO) to the advanced LIGO (aLIGO), the jump in sensitivity allowed scientists to make the first direct detection of gravitational waves associated with a binary black hole merger on September 14, 2015 (GW150914). Both aLIGO facilities simultaneously observed a gravitational wave signal from this event (Abbot, 2016). This direct detection led to another discovery of binary black holes on December 26, 2015 (GW151226).

With the advancing of technology, the main goal for aLIGO is to become more sensitive such that the detections of gravitational waves can extend to other sources such as neutron-star binary mergers. A few differences between iLIGO and aLIGO are that the input laser power changed from 10 W to 180 W, the mirror mass changed from 10 kg to 40 kg, the power-recycled Fabry-Perot arm cavity Michelson became a dual-recycled Fabry-Perot arm cavity Michelson, the seismic isolation performance improved from 50 Hz to 12 Hz, and the mirror suspensions increased from a single pendulum to a quadruple pendulum (Weinstein, 2012). The suspension design caused noise sources that would limit initial LIGO sensitivity and thus needed to be upgraded. Seismic noises are from the natural occurrences such as earthquakes and tidal waves to man-made sources such as traffic. Thermal noise results from the thermal energy of atoms and molecules in the mirrors and their suspensions which are at a finite temperature. According to the fluctuation dissipation theorem, off-resonance noise can be reduced by using low dissipation materials in our suspension system which stores most of this thermal energy close to the resonant modes. There are a number of noise sources that affect the sensitivity of the interferometer gravitational wave detector (Kumar, 2013).

The more common sources of noises are seismic, gravitational gradient, thermal, and quantum noise.

A goal of physicists is to increase the amount of advanced gravitational wave networks because it improves the ability to locate sources in the sky while also increasing the detection sensitivity. By multiplying the amount of interferometers globally, it will allow the system to enhance the network sky coverage and maximum time coverage. With the increase of interferometers, the detection confidence will increase because there will be an increase of signal detections for a single source. In order to increase the sensitivity, the current research is being done to decrease the different noises.

2 Motivation for Decreasing Thermal Noise in Advanced LIGO Detectors

The main concern of enhancing aLIGO is the process of reducing the different noises associated with the apparatus. In this project, the main source of noise is thermal noise. The current material that is being used for the test masses and fibers is fused silica. Fused silica helps reduce the amount of off resonance Brownian motion from the atoms and molecules. In order to reduce the thermal noise, one potential process is using cryogenic techniques (Heptonstall, 2004). The focus of this project is to investigate fused silica using Finite Element Analysis (FEA) to analyze their properties as cryogenically-cooled suspension fibers. The main goal of this project is to work towards building a full model suspension system to allow for direct calculation of the mechanical admittance. By using FEA, the project will analyze built models for the gravitational wave detector mirror suspensions, specifically for third generation detectors using a fused-silica hybrid type suspension for the interferometry mirrors.

3 The Approach Towards Decreasing Thermal Noise

The approach for this project involves beginning with basic models that will allow me to make consistency checks with analytical models which can be tested with experimental measurements. Experimental measurements would be based on real world tests to make sure that the models respond accurately. The main goal is to construct the sections of the model piece by piece in order to completely model the final stage of a cryogenically cooled mirror suspension made from a hybrid fused silica/silicon material; the fused silica fibers will be attached to a silicon mass.

4 The Methods Used to Model the Suspension Systems

The methods that are being used is using a program known as ANSYS. ANSYS allows a user to use FEA for multiple models using different analyses. The analyses that have been used to become familiar with the program are Modal analysis and Static Structural analysis. The goal is to continue to become familiar with the program in order to begin modeling the actual suspension system. By comparing analytically and computationally, there will be multiple checks on the ANSYS program to ensure the program accounts for the "real world" forces and natural phenomenon correctly. Not only is ANSYS being checked, but also the modeling techniques. This can be seen by the importance of ensuring the models have the correct meshing and mesh density and are built in such a way as to accurately reflect the real physical system. This is important because by accurately describing the mirror suspensions computationally, then experiments will be made with small increments without making a physical suspension system.

5 Progress Towards Modeling the Suspension Systems

The progress that has been made is becoming familiar with the ANSYS program. The beginning processes that have been made is to experiment with the different meshing systems that are available. For example, there are different types of meshing basics that allow the program to analyze different parts of a model more efficiently or effectively. By maximizing performance and decreasing the amount of nodes and elements used, decreases the processing power and loading time for each solution. The main goal in meshing is to capture as much information as possible without using excess amount of nodes and elements. One way to change the amount of nodes and elements a model has is by changing the relevance. The relevance is used to define the sizing and density of the nodes and elements. By increasing the relevance, the amount of nodes and elements increase overall, as shown in Table 1.

Relevance	Relevance Center: Course	Relevance Center: Medium	Relevance Center: Fine
-100	Nodes: 216 Elements: 125	Nodes: 1000 Elements: 729	Nodes: 343 Elements: 216
0	Nodes: 2197 Elements: 1728	Nodes: 13824 Elements: 12167	Nodes: 68921 Elements: 64000
100	Nodes: 8000 Elements: 6859	Nodes: 54872 Elements: 50653	Nodes: 205379 Elements: 195112

Table 1: Relevance Center of a 20 mm x 20 mm cube. Here are the various pre determined extreme choices a user can make when deciding on a meshing size.

The main process was to determine whether analytical results matched up to the computational results of the ANSYS program, a model of a cantilever bar was used. The equation that was used to analytically solve the different frequencies was: $\omega_n = (k_n L)^2 \sqrt{EI/mL^4}$. Where ω was measured in radians/second, n is the number of modes, E is Young's Modulus, I is the moment of cross section, L is for the length of the bar, and m is the mass per unit length. For I as a rectangular cross section is $I = ba^3/12$ (where a and b are the sides) and for a circular cross section $I = \pi/64 * d^4$ (where d is the diameter). Lastly the different modes of $k_n L : (k_1 L) = 1.875$, $(k_2 L) = 4.694$, $(k_3 L) = 7.855$, $(k_4 L) = 10.996$, $(k_5 L) = 14.137$ and for modes above 5 then $(k_n L) = (2n - 1)(\pi/2)$.

In Figure 1, the frequency was calculated in Hz, and the analytical results closely matched the computational results. However, as the frequency increased in the amount of modes, the analytical results did not match as closely to the computational results.

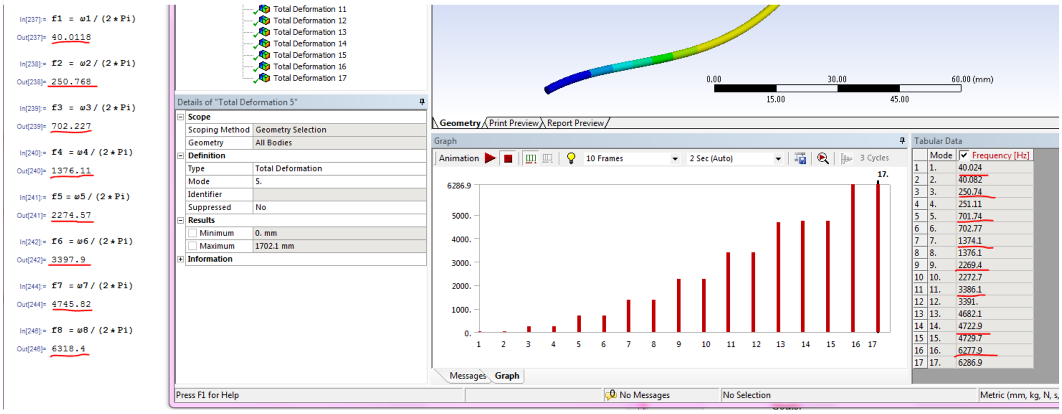

Figure 1: This is the comparison between the analytical results (Mathematica) and computational results (ANSYS) for a fused silica wire with the diameter of 2 mm and a length of 200 mm. In Appendix A, the tutorial for how to make a simple cantilever bar with a circular cross section is shown. The same process to make Figure 1 is described in depth.

One of the main goals is to get the analytical results to match closely to the computational results. This will allow the models to be more accurate when actually making the full mirror suspension systems. There are parts of the program that need to be accounted, for such as gravity. With this in mind, continual tests are being made comparing a gravitational force being applied and a non-gravitational force when analyzing the static structural design. Each model that is being produced, have slight changes each time such as the amount of force being applied to one of the faces if it's a pendulum or where the fixed support is located at.

In order to match the criteria for fused silica, the material characteristics were manually inputed into the analysis each time. Table 2 shows the multiple characteristics of fused silica.

Characteristic	Symbol	Value
Young's Modulus	E	7.2E10 N/m^2
Density	ρ	2200 kgm^{-3}
Specific Heat Capacity	C	770 J/kgK
Thermal Conductivity	k	1.38 W/mk
Linear Thermal Expansion		3/9E-07 K^{-1}
Poisson's Ratio		0.17
Reference Temperature		21 °C

Table 2: The fused silica characteristics.

The goals for the future are to accurately describe each model and understand step-by-step what ANSYS computationally produces each time with small variations that are applied to multiple models. Also transfer functions will be used to describe pendulum motions. The goal is to find a way to have ANSYS accurately describe these transfer functions and modes. Using the techniques learned, the models of the pendulum will change to represent the dimensions of the mirror suspensions. The approach to obtain the future goals is to continue testing and practicing the capabilities of ANSYS. Then once further along, the main goal will then become making the suspension systems using the actual representations.

6 The Main Challenges in Using ANSYS

The most challenging aspect of the project is to become familiar enough with the technique to accurately describe each part of the model. Another challenge will be to use unfamiliar programs such as the program that is currently being used: ANSYS. The main challenges of ANSYS is making models that converge without having error messages. The amount of time it takes to process a more complicated model is not worth the time if at the end there are multiple errors that stop the calculation. Another challenge is finding out the potential of ANSYS when it comes to what functions it can produce and what actual processes it needs to perform to match the suspension system criteria.

The future challenges that can be anticipated is the boundary conditions for ANSYS. The more complex the model is, if there are not enough constraints, ANSYS will not converge and finish the computations. This poses a problem because of the extreme disproportions of the actual fused silica wire compared to the 40kg mirror.

7 Appendix A - A Tutorial for a Simple Model of ANSYS

A Simple Model of the Modal Analysis of a Cantilever Cylinder Cross Section in ANSYS tutorial

1. Open the ANSYS workbench
2. Drop and drag the "Modal" analysis system into the project schematic
3. Right click on "Engineering Data" and edit the material. There are predetermined properties found in the "Engineering Data Sources" (Right click on the description or press the books in the top right hand corner). For this tutorial, fused silica was created with these parameters:

Young's Modulus E: $7.2E10Nm^{-2}$ (Isotropic Elasticity) Density p: $2200kgm^{-3}$ (Density) Specific Heat c: $770J/kgK$ (Specific Heat) Isotropic Thermal Conductivity k: 1.38 W/mK (Isotropic Thermal Conductivity) Coefficient of Thermal Expansion: $3.9E-7$ (Isotropic Secant Coefficient of Thermal Expansion) Reference Temperature: 21 C (Isotropic Secant Coefficient of Thermal Expansion) Poisson's Ratio: 0.17 (Isotropic Elasticity)

In order to put in the parameters, just drag and drop from the toolbox. Also make sure the filter is turned off (that's the filter symbol next to the engineering data sources/books in the top right hand corner) this will allow a user to few all the parameters.

4. Return to the project, and open/edit the geometry. The cantilever that was used for this tutorial is a cylinder with a diameter of 2 mm with a length of 200 mm, shown in Figure 2.

How to Create a Geometry:

- Open up the geometry
- Decide which plane you want to start drawing in: XY Plane, ZX Plane, YZ Plane
- Click on "new sketch" in the 4th row from the top after "Sketch1" it's blue sketch
- Whatever plane you decide to choose, press the "Look at Face/Plane/Sketch" icon in the top right hand corner (The person looking at a face)
- This will allow the view to go to the view where you want to sketch on
- Click on "Sketching" which is next to the "Modeling" right above the "Details View"
- Here will be Several Options

- Draw: This is where you can decide what shape you want to use
- Modify: This is the extras such as fillets, chamfers and corners
- Dimensions: This is where you can select what type of dimensions you will be using
- Constraints: This is where you can constrain lines or connect lines
- When you want to connect lines together, click on each end and make them "Coincident" this way the shape will connect when you change dimensions
- Settings: This is where you can show a grid or snap grid
- Sketch Color:
- Teal: Underdefined: The sketch does not have enough dimensions to constrain and tell the program what exactly the shape is
- Blue: Defined: The sketch is fully defined and does not need any more dimensions
- Red: Overdefined: The sketch has repeating dimensions that causes errors
- For this tutorial, draw a circle. I made mine coincident to the middle by pressing the vertex in the middle and one of the axis and then repeat the process with the second axis to center the circle
- Dimensions: The diameter is 2 mm, under dimensions to display the number, click "Display" and then "Value" instead of "name"
- To change the dimensions, the "Details View" in the bottom left hand corner will say D1 and from there you can change the diameter
- From there in order to create the length of the cylinder, press "Extrude" at the top (4th row) towards the right. From there it will ask what geometry, click on your sketch and click "Apply". Make sure the "Operation" is "Add Material" to Extrude.
- Then choose a "Depth" in this case it's 200 mm. Then click "Generate" the lightning bolt next to the "New Sketch" symbol.
- Your cylinder should look like Figure 2.

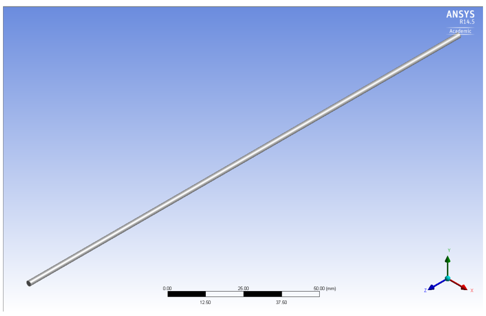

Figure 2: This is the geometry of the 2 mm in diameter cylinder with a length of 200 mm

5. Return to the project and open the Model tab. With that open, go to the subtab of ”Geometry”, named ”Solid”. Under Solid go to ”Assignment” under ”Material” to assign Fused Silica or whatever material you decide to choose.

6. Go to the ”Mesh” tab. In this case the ”Element Size” under the ”Sizing” tab is 0.04 and the ”Relevance Center” as ”Fine”. Right click on the mesh and press ”Generate mesh”.

7. Under the Modal tab, go to the Analysis Settings and change the ”Max Modes to Find” for ANSYS to calculate. In this tutorial, the amount of modes that were used was 17.

8. Right Click the Modal and press on ”Fixed Support”. This will make the bar a cantilever bar, once the geometry is set to fix one face of the bar. Press on one of the faces and click on ”apply” on the Geometry.

9. Right click on the solution and press ”Solve”. Let Ansys run the modes through.

10. On the Solution tab, the ”Tabular Data” is listed but the Total Deformation has not been listed. In order to do so, select all the frequency of the modes in the column and right click and press ”Create Mode Shape Results”.

11. Once loaded, the total deformation will have a lightening bolt next to each entry, right click and press ”Solve” or ”Evaluate All Results”. This will make all the entries have a green check mark.

12. This will give the user the ability to animate each entry. This is down by clicking the play (sideways triangle button). ANSYS will run through the simulation for that mode that is selected, shown in Figure 3.

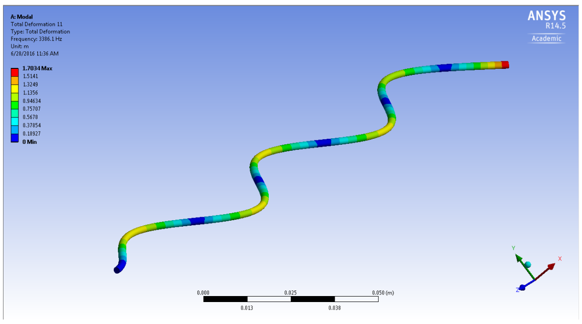

Figure 3: This is the simulation of the 200 mm long cantilever bar using ANSYS. Looking at the different modes.

13. Using Mathematica (or another computational system) input the analytical solution for a Cantilever bar fixed to one end.

$w_n = (k_n L)^2 Sqrt(EI/mL^4)$ Where: w_n is the frequency measured in radians/second E = Young’s Modulus I = moment of cross section $I_{rectangle} = ba^3/12$ (b and a are the sides) $I_{circle} = (\pi/64)(d^4)$ (d is the diameter) m = mass per unit length L = length of the bar k_n relates to the amount of nodes $k_1 L = 1.875$, $k_2 L = 4.694$, $k_3 L = 7.855$, $k_4 L = 10.996$, $k_5 L = 14.137$ n greater than 5: $k_n L = (2n - 1)(\pi/2)$ Convert w_n to f measured in Hz

14. Compare only to the bar actually bending, not twisting or contracting. There are modes that are the same due to the symmetry of the bar. In Figure 4 the underlined frequencies compare to the analytical calculations (Mathematica) and the computational calculations (ANSYS).

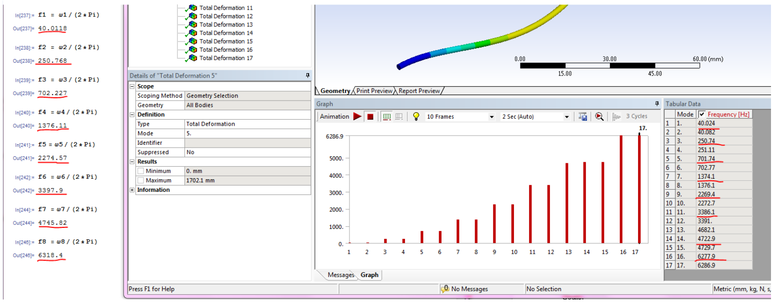

Figure 4: This is the comparison between the analytical results (Mathematica) and computational results (ANSYS) for a fused silica wire with the diameter of 2 mm and a length of 200 mm.

8 References

Abbott, B. P., et al. (2016). Observation of Gravitational Waves from a Binary Black Hole Merger. Physical Review Letters.

Arain, M.A., and Mueller, G. (2008). Design of the Advanced LIGO recycling cavities. Opt. Express Optics Express, 16(14), 10018. doi:10.1364/oe.16.010018

Aston, S. M., Barton, M. A., Bell, A. S., Beveridge, N., Bland, B., Brummitt, A. J., . . . Wilmut, I. (2012). Update on quadruple suspension design for Advanced LIGO. Class. Quantum Grav. Classical and Quantum Gravity, 29(23), 235004. doi:10.1088/0264-9381/29/23/235004

Heptonstall, A. W. (2004). Characterisation of mechanical loss in fused silica ribbons for use in gravitational wave detector suspensions (Unpublished master's thesis). University of Glasgow.

Kumar, R. (2013). Aspects of suspension design for the development of advanced gravitational wave detectors (Unpublished master's thesis). University of Glasgow.

Weinstein, A. (2012, June 21). Status of the Search for Gravitational Waves [The description of the Laser Interferometer Gravitational-Wave Observatory (LIGO) detectors around the world and the status of detecting Gravitational waves through technical advancements.].

What are Gravitational Waves? (n.d.). Retrieved July 05, 2016, from https://www.ligo.caltech.edu/page/what-are-gw