CALIFORNIA POLYTECHNIC STATE UNIVERSITY
AEROSPACE ENGINEERING DEPARTMENT
SAN LUIS OBISPO, CA

Effect of Multifunctional Material on the Mechanical Behavior of Composite Structure Using Finite Element Analysis

Nicholas A. Romonoski

Acknowledgements:
I would like to thank Kodi Rider as well as Dr. Elghandour for their help with the use of COSMOS/M.

Table of Contents

List of Figures

List of Tables

Effect of Multifunctional Material on the Mechanical Behavior of Composite Structure Using Finite Element Analysis

Nick Romonoski
California Polytechnic State University, San Luis Obispo, CA 93401

This project will focus mainly on the effect of multifunctional material on the mechanical behavior of composite structure using finite element analysis. The structure that will be analyzed is that of a bicycle frame. Using traditional bicycle frame geometry, the frame is broken down into its components and analyzed individually with its respective loading. The models created will be verified using theoretical analysis. Once material characteristics for the multifunctional materials have been collected, they will then be applied to the same models to predict the mechanical behavior of the structure.

I. Introduction

Driven by increasing environmental awareness, many companies have made advancements in the incorporation and development of many natural and organic fibers in their products. Examples of this increased use of natural fibers can be seen in the automotive industry, as well as bicycle industry. Automakers use natural fibers as a component of a cars interior, such as natural fiber reinforced thermosets and thermoplastics in door panels, package trays, seat backs and trunk liners. Bicycle makers such as Calfee Design have started production of a bike, which uses bamboo and hemp as the main materials of the frame. The increased use of these materials are rooted in a desire to find a low cost and low weight alternatives to more traditional fibers, like fiberglass or carbon fiber. These natural fibers, which can be comprised of flax, jute, bamboo, hemp, or kenaf as well as other agriculture products, have signaled the start of a so-called “green” industry that has enormous potential. These organic materials have many advantages over traditional materials, which include a lower weight, greater availability, thermal and acoustic insulation, and ease of recycling. Organic materials are also carbon dioxide neutral, which means that when they are burned the fibers reportedly give off no more carbon dioxide than they consumed while growing. This should be considered a major factor due to the relative lack of knowledge on how composite structures are going to be disposed of at the end of their life. More advantages can be seen in the energy consumption used to produce natural fibers compared to glass fibers. On average natural fibers suitable for composites is around 60 percent lower in energy consumption than glass fibers. The main advantage of multifunctional materials, natural fiber and epoxy, is that they are environmentally friendly and may possibly replace existing materials used in composites.

Therefore, this project will focus mainly on the effect of multifunctional material, natural fibers with organic epoxy, on the mechanical behavior of composite structure using finite element analysis. The structure that will be analyzed is that of a bicycle frame. To do this a traditional bicycle frame geometry will be broken down into its components and analyzed individually where the applied loading due to the frame geometry will be taken into account. Each component will be modeled using finite element software and verified with theoretical analysis. Once all of the components of the frame have been modeled and analyzed, the parts will be assembled and an analysis of the complete bicycle frame will be completed. During this process, the effect of the multifunctional material will be seen in the frames mechanical behavior. A general overview of the Cosmos/M program and the process required to correctly create the models will be described and displayed in the latter sections.

II. 3-Point Bending Model

The finite element analysis was performed in the Cal Poly structures lab using the labs computers equipped with Cosmos/M, a finite element analysis program. The process used to find the stress and strain distributions in the structure is a relatively simple one. First the structure needs to be accurately modeled in the FEA program. Then the material characteristics need to be determined, and an appropriate mesh needs to be created for the structure. The boundary conditions and the applied loads are then applied, and finally the model was run through Cosmos/M and the results were determined.

In order to model the structure, dimensions were determined. For the following cases an arbitrary length and width pipe was created to do the analysis. The dimensions will not matter as much at this point since only the process of modeling and analyzing of each case is done correctly. Once these are verified the exact dimensions of the components can be taken into account, using the same boundary conditions and the new material properties. Figure 1 below shows the pipe geometry and its dimensions that will be modeled for the analysis. Figure 2 below also shows the model of the pipe in Cosmos/M.

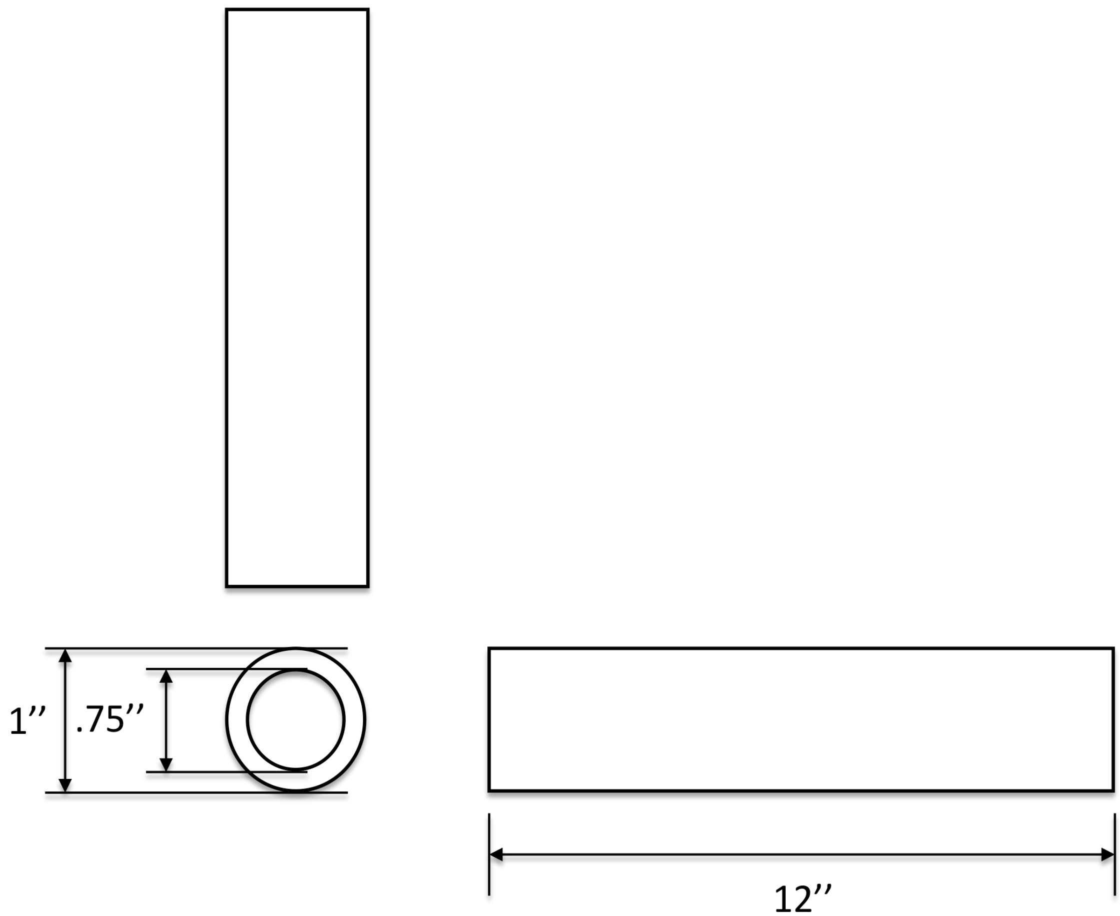

Figure 1: Pipe Dimensions

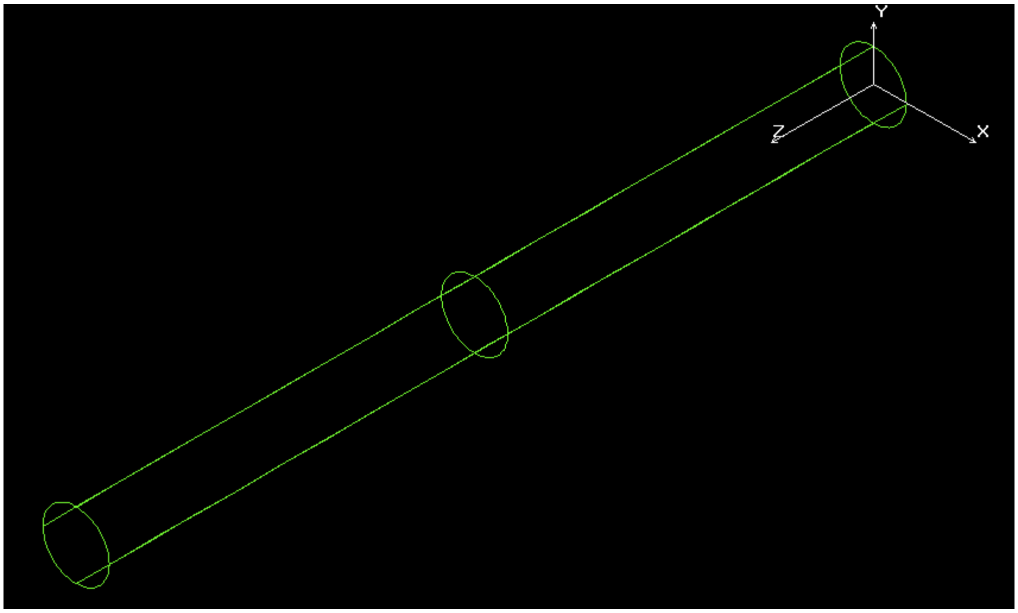

Figure 2: Pipe Geometry Model

Now that all of the dimensions have been determined and taken into account, the appropriate model for the structure was made in the finite element analysis program. The first case that will be analyzed is 3-point bending. Using the geometry previously established, the loads and boundary conditions for a pipe in 3-point bending will be used. Since the geometry has been created the material properties and element type can be set. For this model, the pipe material will be set as Aluminum 2024. With that being said the modulus of elasticity, E, is 11.7 Msi and Poisson's Ratio is 0.33. As far as the element type, it will be defined as a 'SHELL4'. 'SHELL4' designates a four-node thin shell element, and is used for areas or two-dimensional surfaces. Since the pipe is modeled as a two-dimensional surface rather than a solid structure this element group is appropriate. Characteristics of 'SHELL4' are that it is a quadrilateral element for linear elastic materials, and will also assume small deflections. Due to the nature of this element group a 'Real Constant' will need to be set designating the thickness, temperature gradient, foundation stiffness, material angle, among others. For this case though, the thickness only needs to be entered, 0.125 inches thick, and the other options left to their default values.

After the model has been created and the material and element properties have been set we can continue and start building the mesh that will be used to analyze the structure. The mesh determines how many nodes and elements are going to be analyzed for the structure. In general, the more nodes and elements there are the more accurate the results will be. Although, when the amount of nodes and elements are increased the computational time for the model also increases. There is a certain limit at which the amount of nodes and increase of accuracy is worth the increase in computational time. In this project a lower number of nodes and elements will be used due to the difficulty in determining which element number corresponds to a location on the structure, while also keeping the computation time low. Figure 3 below shows the mesh created for the pipe that will undergo 3-point bending. The mesh created consists of a total of 2,000 elements and 2,243 nodes. A critical note is that in the creation of the mesh, the nodes connecting each surface need to be overlapped. The reason for this is to create a structure that would act as one solid rather than numerous individual surfaces. Just overlapping the nodes isn't enough to make the program think that it is a single structure. To fix this the overlapping nodes were merged, effectively connecting all of the surfaces together.

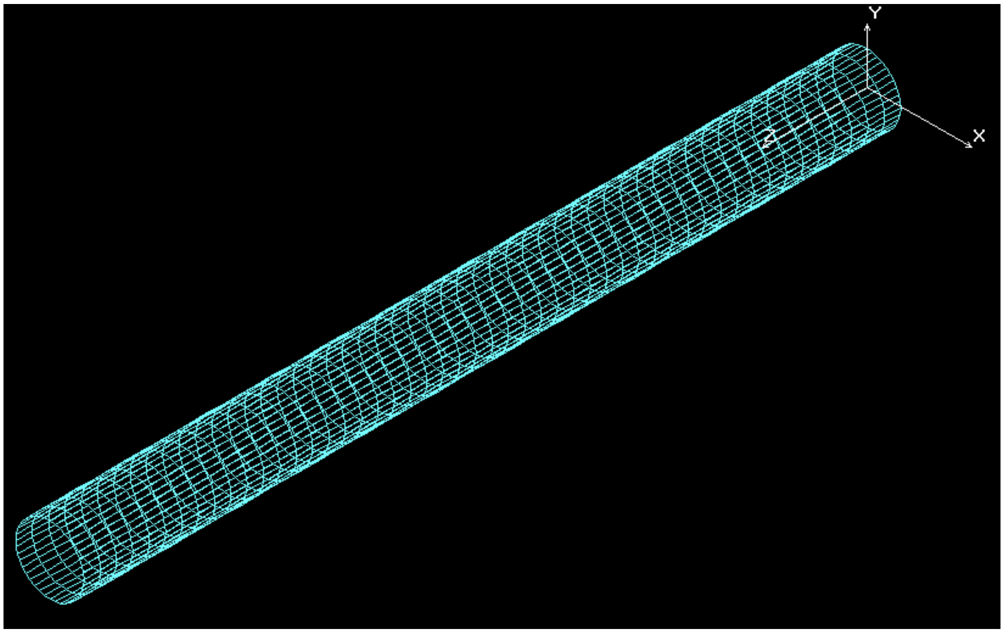

Figure 3: 3-Point Bending Model Mesh

Now that the meshing has been created the constraints and loads can now be applied. It is critical that the constraints are put in the correct directions. If they aren't correct, there is a possibility that you are analyzing a completely different structure then the one that was intended, therefore you won't get good results. With that being said the constraints for the pipe were placed along its centerline and the edges. The centerline along the pipes lengthwise direction and surface was constrained in the z-direction restricting its movement in that direction along that curve. The pipe was also constrained from moving in the y-direction by placing a constraint on the bottom of the pipe where it would be in contact with the supports if it were actually being tested in the Instron machine. Figure 4 below shows the constraints placed on the pipe for the case of 3-point bending.

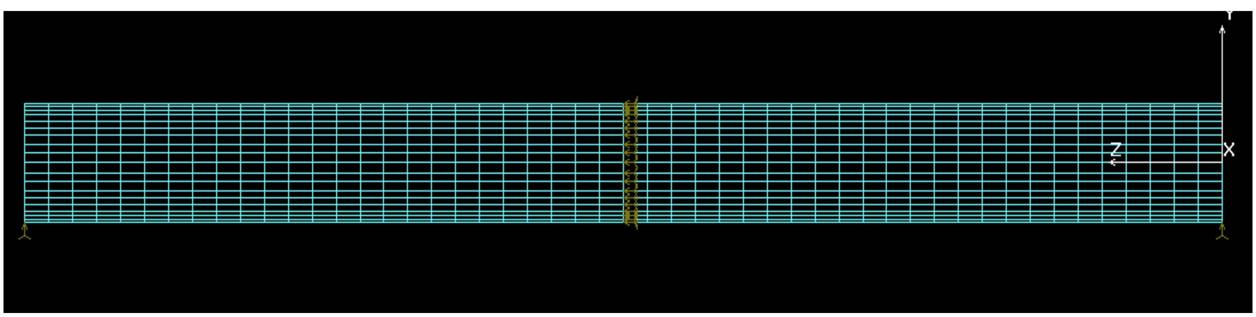

Figure 4: 3-Point Bending Model Constraints

Once the constraints are set, the loads can be applied in the same manner. For the loads it is important that they are placed in the correct direction or else you can run into the same problem as putting the constraints in the wrong place. Using the 3-Point Bending case, the load of an arbitrary 10 pounds was applied at the centerline in the y-direction. In reality, if this were to be tested, this is where the load would be applied. In the case of a bicycle frame, the loads will not be applied this way but the component will feel the same result in that it is bending. Figure 5 below shows the load applied to the pipe.

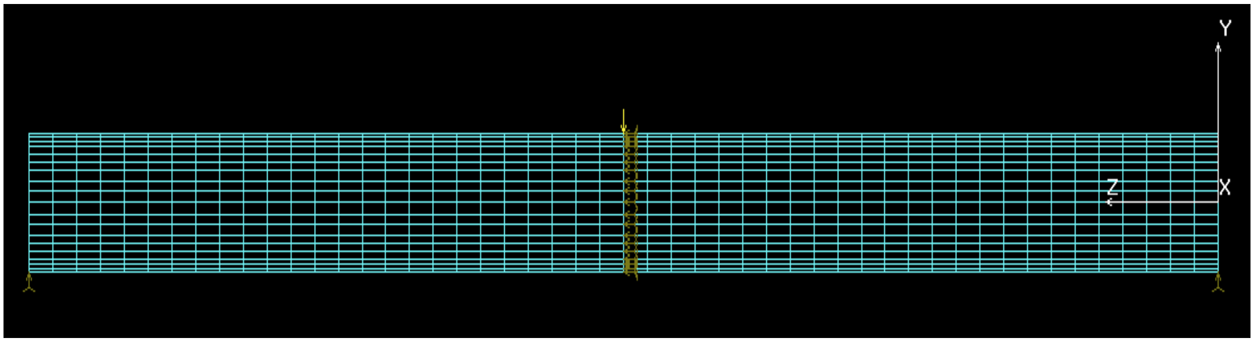

Figure 5: 3-Point Bending Model Loading

The model for the structure has now been completed and the analysis can be performed. For this particular structure, static analysis will be done, therefore the static analysis options need to be set, and then the analysis can be completed. Doing so will find the displacement and stresses of each node that was created in the mesh. The stresses and strains of every created element will also be calculated.

Now that the previously made model has been completed and ran through the Cosmos/M solver we can look at the results. Cosmos/M provides an abundant amount of information such as deformation along with the stress and strain distribution of the model. Looking at the deformed pipe, figure 6 below, one can notice that the deformation is symmetrical. The loading and the boundary conditions are completely symmetrical therefore the deflection will be symmetrical. A complete list of every nodal and elemental stress can be provided upon request.

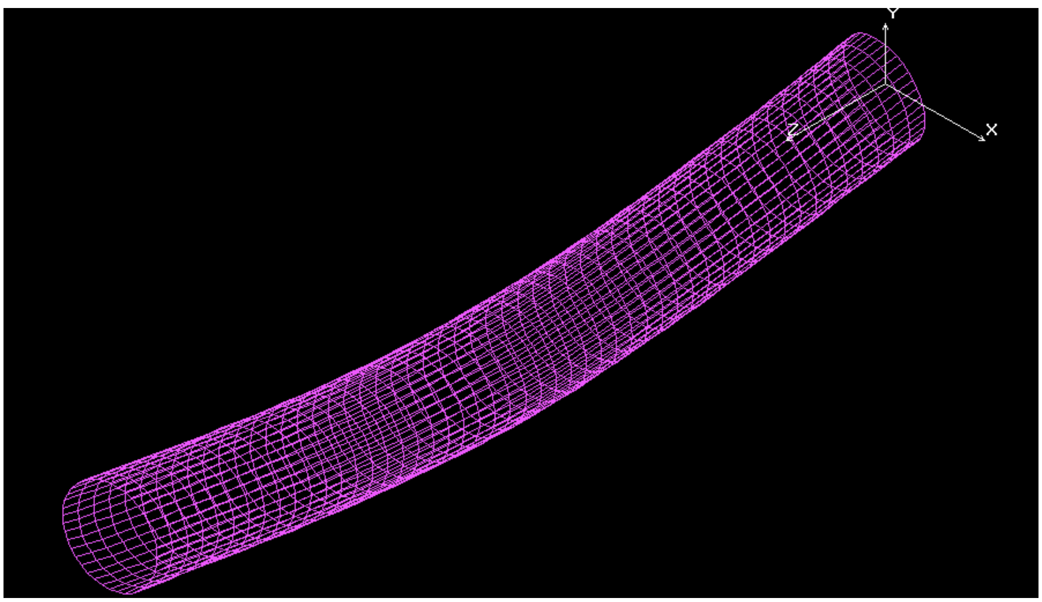

Figure 6: 3-Point Bending Model Deformation

III. 3-Point Bending Model Validation

Now that the model for the 3-Point Bending case has been made and analyzed, we must verify it to make sure that it is giving us the correct values. This can be done easily using theoretical analysis. In order to do this assumptions must first be made to simplify the analysis. First, the pipe loading is assumed to only be acting in the linear elastic region. In other words, the yielding stress has not been reached and thus no plastic deformation. Second, the beam is assumed to be bending in one plane. Essentially, this is only a 2 dimensional problem, so there is no twisting involved either. Third, the pipe is entirely isotropic, so the material is the same throughout the entire pipe. Fourth a 10 pound load is applied to the pipe of length 12 inches. Fifth, the pipe is initially straight before loading.

In the case of the 3-Point Bending model, it is assumed that the stress is caused entirely by bending. This stress can be found from

$$\sigma = \frac{Mc}{I} \tag{1}$$

where M is the moment, c is the distance from the neutral axis, and I is the moment of inertia about the axis the beam is bending.

The centroid can be found using the following equation, but it is known that for a pipe the centroid lies at the center.

$$\bar{x} = \frac{\sum A_i \bar{x}_l}{\sum A_i} \tag{2}$$

where A_I and $\bar{x}_l$ are the respective areas and centroid of each piece of the geometry respectively. When applied to the geometry, and as stated earlier the centroid is found to be located at the center of the pipe.

The moment of inertia of the pipe can be found using

$$I = \frac{\pi}{4} r^4 \tag{3}$$

for a circle, or more specifically for a pipe

$$I = \frac{\pi}{4}\left(r_o^{\,4} - r_i^4\right) \tag{4}$$

where r_o is the outer radius and r_i is the inner radius. Applying this to our case the moment of inertia results in

$$I = \frac{\pi}{4}\left(\left(\frac{1}{2}\right)^4 - \left(\frac{0.75}{2}\right)^4\right) = 0.033556 \text{ in}^4 \tag{5}$$

The moment is simply the reaction at one end, 5 pounds, multiplied by the moment arm. For the point of interest, and the point that we will be comparing/validating, the moment arm is 6 inches since it is located at the center of the pipe. Therefore, the moment for this particular location is found to be 30 in-pounds. Also for the point that is going to be compared, the distance from the neutral axis is going to be 0.5 inches since it is located at the outer edge of the pipe and the centroid is at the center of the pipe. The culmination of these values resulted in a bending stress of 447.016 pounds per square inch, shown below.

$$\sigma = \frac{Mc}{I} = \frac{(30in * lb)(0.5in)}{(0.033556in^4)} = 447.016\frac{lb}{in^2} \tag{6}$$

We can now use this value to compare what was found using the finite element method. If we look at the list of stresses found for the 3-Point Bending model it was found that the stress due to bending was 471.6 pounds per square inch, at the half span of the pipe on the outer edge. Using the percent error formula we find the percent error to be 5.5 percent. This is an acceptable value for the error, and may be attributed to the assumption of there being no shear stress in the model.

IV. Buckling Model

Using the same process for the 3-point bending model, we can model the same pipe but now in a buckling situation. The geometry and dimensions, as well as the material and element group will remain the same from case to case. To restate these, the pipe is 12 inches long with an outer diameter of 1 inch and a thickness of 0.125 inches. The material is Aluminum 2024, and the element group 'SHELL4' for a four-node thin shell element. For this particular case, a different sized mesh was created and can be seen below in figure 7.

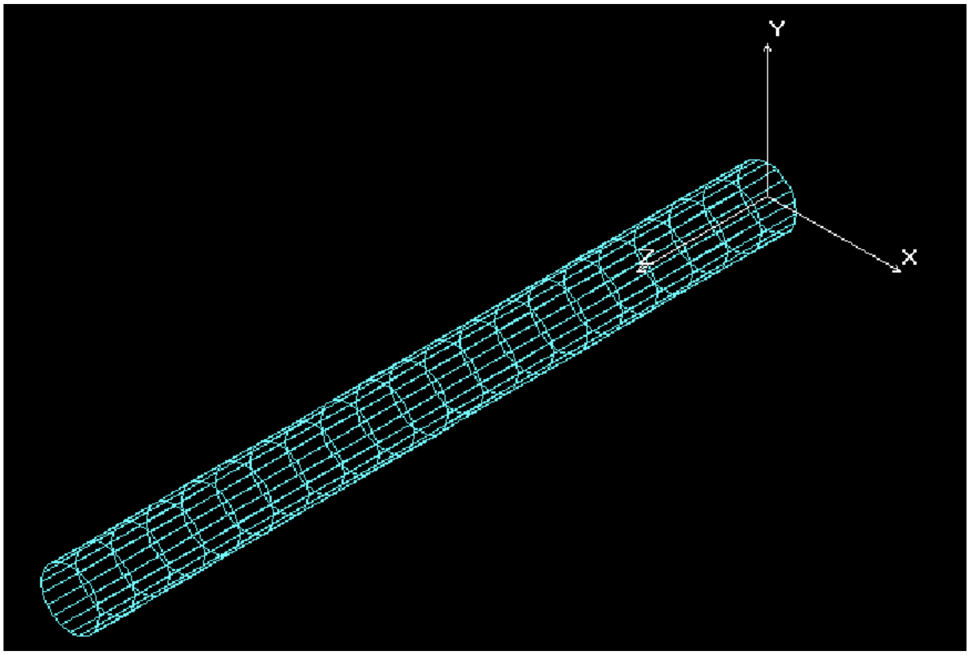

Figure 7: Buckling Model Mesh

For this case the boundary conditions and loads are going to be different. The constraints for this case are going to be set along both edges of the pipe. For the edge set at the origin, the pipe will be constrained in all 3 x, y, and z directions as if it were completely fixed. For the opposite edge, it will be constrained from movement in the x and y direction, making it a pinned end. Figure 8 below shows the constraints set on the pipe in the case of buckling. Now that the constraints are set the loads were placed. The loads for this case are set at the pinned end to create compression. A load of 20 pounds was placed along the edge, resulting in a load of 1 pounds per node. Figure 9 below shows the loaded column. It should be noted that the applied load as well as the geometry of the pipe is arbitrary at this point, since only the process and boundary conditions for each model are trying to be validated.

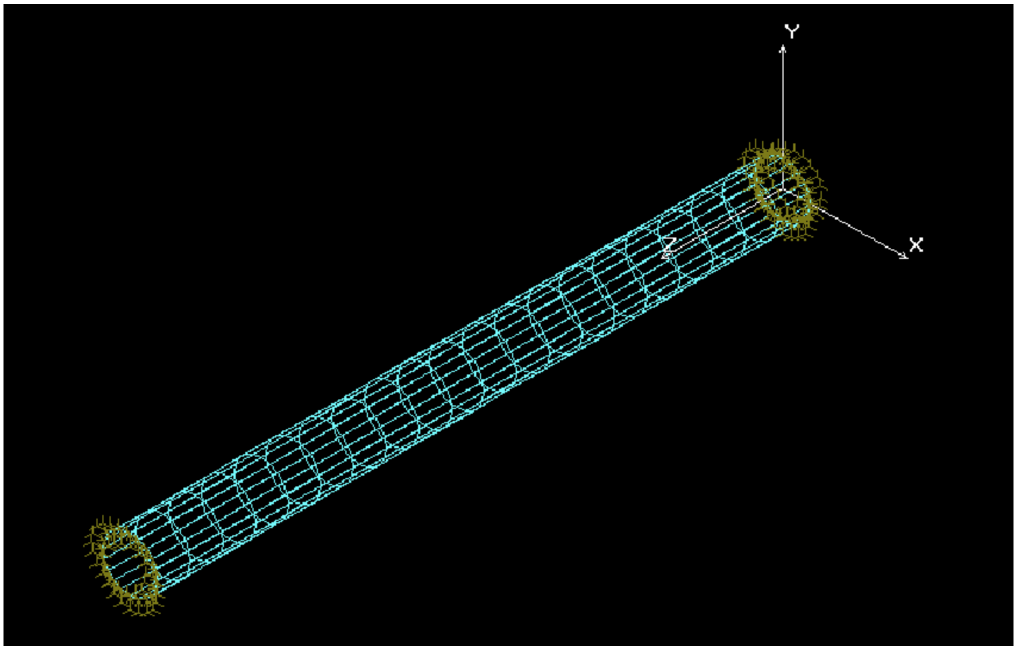

Figure 8: Buckling Model Constraints

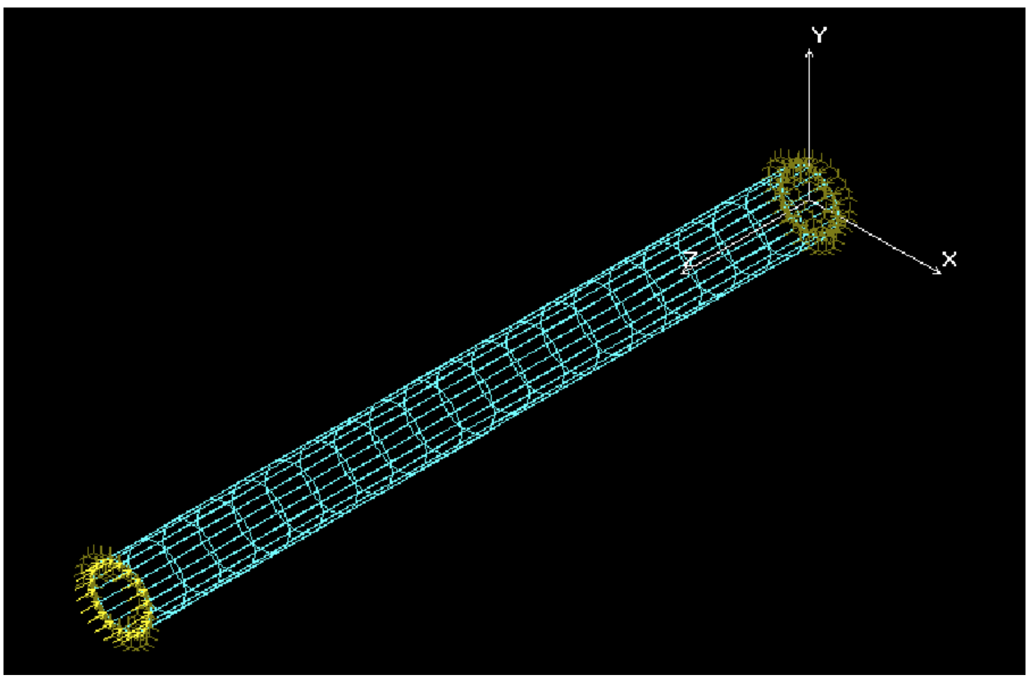

Figure 9: Buckling Model Loading

The model for the structure has now been completed and the analysis can be performed for the buckling case. For this particular structure, buckling analysis will be done, therefore the buckling analysis options need to be set, and then the analysis can be completed. Doing so will restrict us from finding the stresses and strains, but will allow us to find the deflections and Eigen values, which will help find the critical load.

Now that the previously made model has been completed and ran through the Cosmos/M solver we can look at the results. Looking at the deformed pipe, figure 10 below, one can notice the buckling in the pipe. A complete list of every nodal and element deflection can be provided upon request.

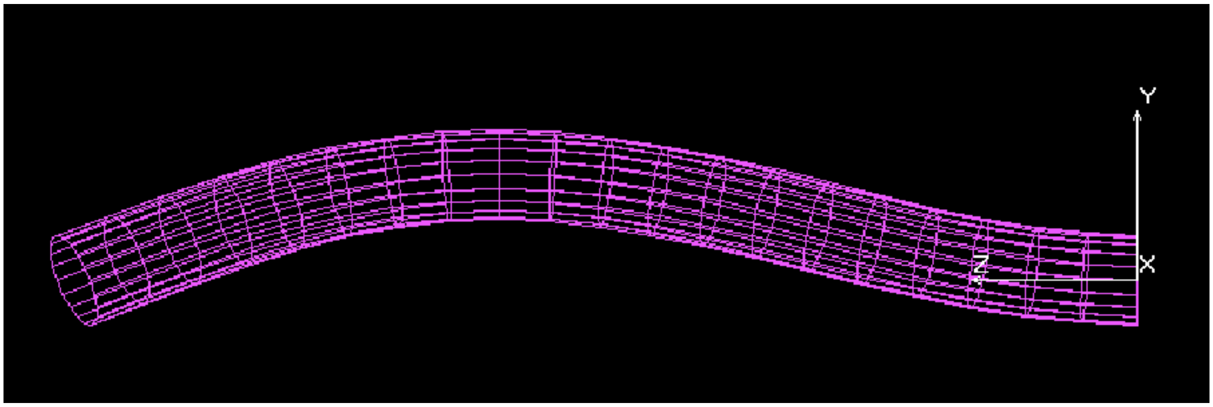

Figure 10: Buckling Model Deformation

V. Buckling Model Validation

Now that the model for the buckling case has been made and analyzed, we must validate it to make sure that it is giving us the correct values. This can be done easily using theoretical analysis. In order to do this assumptions must first be made to simplify the analysis. First, the pipe loading is assumed to only be acting in the linear elastic region. In other words, the yielding stress has not been reached and thus no plastic deformation. Second, the beam is assumed to be bending in one plane. Essentially, this is only a 2 dimensional problem, so there is no twisting involved either. Third, the pipe is entirely isotropic, so the material is the same throughout the entire pipe. Fourth a 20 pound load is applied to the pipe of length 12 inches. Fifth, the pipe is initially straight before loading.

Using the equation shown below we can determine what the critical load that will cause the pipe to buckle.

$$F = \frac{\pi^2 EI}{(kL)^2} \tag{7}$$

where E is the modulus of elasticity, I is the area moment of inertia, L is the length of the pipe, and k is the column effective length factor. For this case, where one end is completely fixed and the other is pinned, the k will be equal to 0.7. The length, modulus of elasticity, and area moment of inertia are known from earlier on in this report. To repeat though, the length is 12 inches, E is equal to 11.7 Msi, and the I is 0.033556 in^4.

$$F = \frac{\pi^2 EI}{(kL)^2} = \frac{\pi^2 (11.7Msi)(0.033556in^4)}{(0.7*12in)^2} = 54915.8lbs \tag{8}$$

Now using the Eigen value found by Cosmos/M and multiplying it by the applied force we can determine the critical load that the finite element method provides.

$$F = EigenValue * P = 0.339037x10^4 * 20lbs = 67807.4lbs \tag{9}$$

Now that both of the critical loads have been found we can compare them to each other to determine the percent error. Using the percent error formula it is found that the finite element model is 23 percent different.

VI. Tension Model

Using the same process for the 3-point bending model and buckling model, we can model the same pipe but now in a tension situation. The geometry and dimensions, as well as the material and element group will remain the same from case to case. To restate these, the pipe is 12 inches long with an outer diameter of 1 inch and a thickness of 0.125 inches. The material is Aluminum 2024, and the element group 'SHELL4' for a four-node thin shell element. For this particular case, a different sized mesh was created and can be seen below in figure 11.

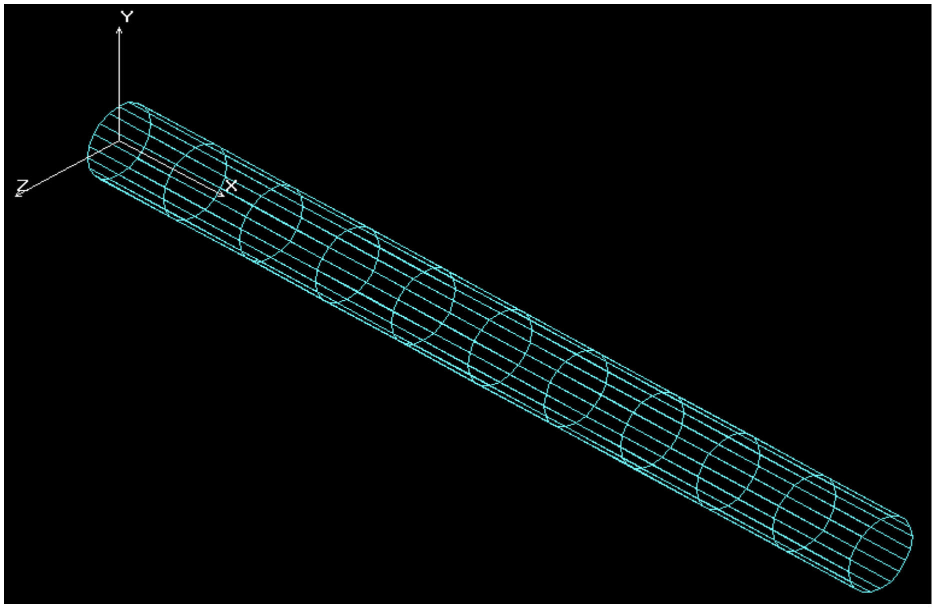

Figure 11: Tension Model Mesh

For this case the constraints are going to be different from the previous two cases. For the tension model, the end at the origin is going to be fixed while the other end is free to move. Boundary conditions were also applied along the pipe on the horizontal axis to constrain the pipe from moving in the vertical direction, and along the vertical axis to constrain the pipe from moving in the horizontal direction. In this case, this is constraining the pipe to a fixed surface rather than putting tension, and pulling from both sides of the pipe. Figure 12 displays the constraints placed on this particular model and can be seen below. For this case, the constraints aren't going to be the only things that are different. For the tension case, a tension force needs to be applied, rather than a compression force in the case of buckling, or a point load at the center in the 3-point bending case. Figure 13 below shows the location and direction of the applied load. A load of 100 pounds is distributed among each node on the edge, resulting in a total of 20 nodes with 5 pounds of force each. It should be noted that the applied load as well as the geometry of the pipe is arbitrary at this point, since only the process and boundary conditions for each model are trying to be validated.

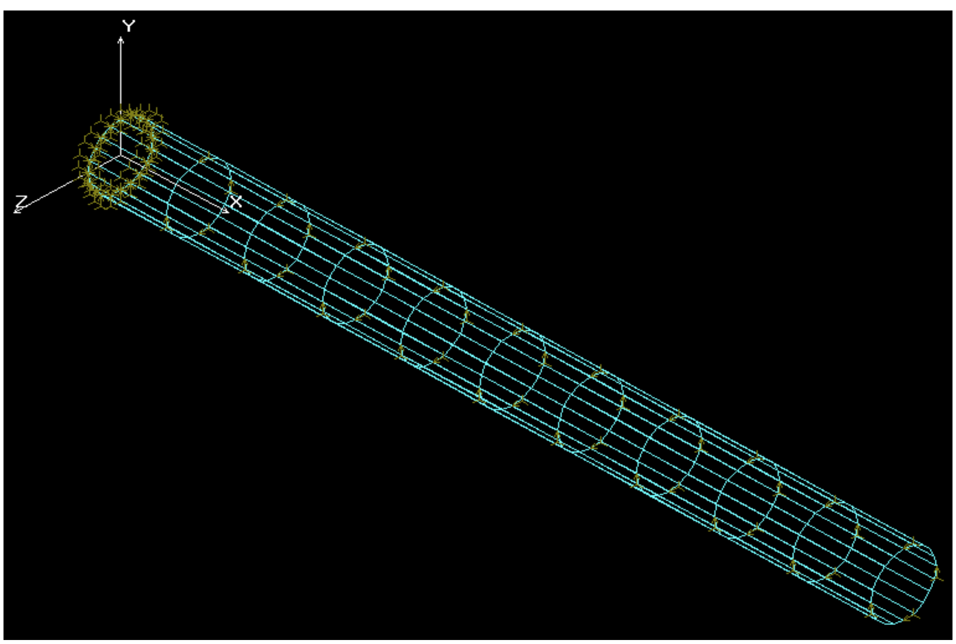

Figure 12: Tension Model Constraints

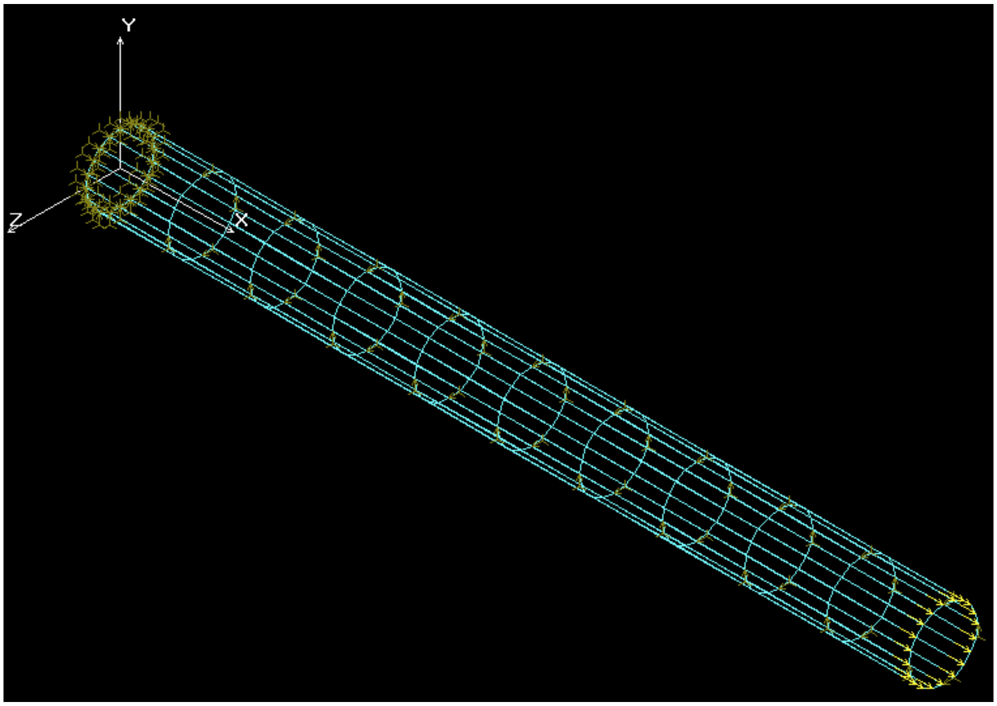

Figure 13: Tension Model Loading

The model for the structure has now been completed and the analysis can be performed. For this particular structure, static analysis will be done, therefore the static analysis options need to be set, and then the analysis can be completed. Doing so will find the displacement and stresses at each node that was created in the mesh. The stresses and strains of every created element will also be calculated.

Now that the previously made model has been completed, and ran through the Cosmos/M solver we can look at the results. Cosmos/M provides an abundant amount of information such as deformation along with the stress and strain distribution of the model. Looking at the stress distribution in the pipe, figure 14, one can notice that the deformation is symmetrical. The loading and the boundary conditions are completely symmetrical therefore the stress distribution and deformation will be symmetrical around the z-axis. A complete list of every nodal and element stress can be provided upon request. One can also notice Poisson's effect taking place. The pipe is being elongated in the z-direction, therefore the pipe will shrink in the x and y-direction. This is also why there is a higher stress at the tip of the pipe when compared to the rest of the body. The smaller cross sectional area of the pipe leads to a higher stress than that of the rest of the pipe that has a larger cross sectional area. This can be easily seen in the following section where the normal stress equations are shown in full detail.

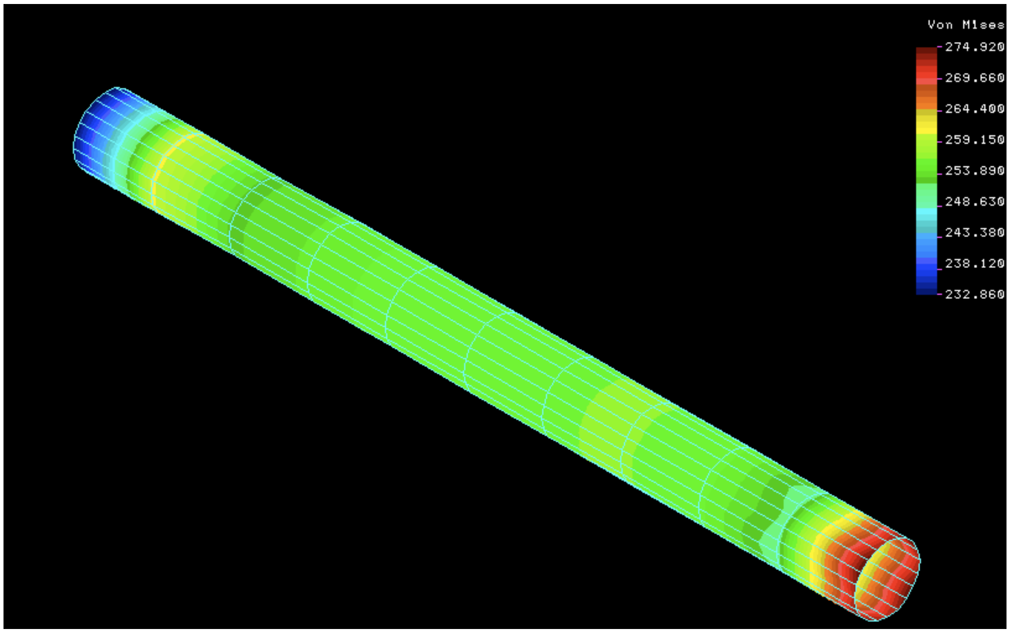

Figure 14: Tension Model Stress Distribution

VII. Tension Model Validation

Now that the model for the tension case has been made and analyzed, we must validate it to make sure that it is giving us the correct values. This can be done easily using theoretical analysis. In order to do this assumptions must first be made to simplify the analysis. First, the pipe loading is assumed to only be acting in the linear elastic region. In other words, the yielding stress has not been reached and thus no plastic deformation. Second, the beam is assumed to be bending in one plane. Essentially, this is only a 2 dimensional problem, so there is no twisting involved either. Third, the pipe is entirely isotropic, so the material is the same throughout the entire pipe. Fourth a 100-pound load is applied to the pipe of length 12 inches. Fifth, the pipe is initially straight before loading.

Assuming that there is no stress due to bending, and that the only stress on the pipe is normal stress we can use the following equation

$$\sigma = \frac{F}{A} \tag{10}$$

where F is the applied force, and A is the cross sectional area of the pipe. Using the equation for the area of a circle we can determine the cross sectional area of the pipe.

$$A = \pi\left(\left(\frac{1}{2}\right)^2 - \left(\frac{0.75}{2}\right)^2\right) = 0.343612 \text{ in}^2 \tag{11}$$

Using the cross sectional area and the applied force we can determine the normal stress on the pipe.

$$\sigma = \frac{F}{A} = \frac{100lbs}{0.343612in^2} = 291.026psi \tag{12}$$

This stress is for a location on the pipe where the cross sectional area hasn't changed, therefore we must compare that value to one under the same conditions on the finite element model. With that being said, we can't use a nodal stress at the end of the pipe because it has started to deform, which results in a smaller cross sectional area. So looking at a node located at half the length of the pipe Cosmos/M outputs a nodal stress of 255.7 psi. If we use the

percent error formula we find that the finite element estimation is approximately 12 percent different from the theoretical value.

VIII. Compression Model

Using the same process for the 3-point bending model, buckling model, and tension model we can model the same pipe but now in a compression situation. The geometry and dimensions, as well as the material and element group will remain the same from case to case. To restate these, the pipe is 12 inches long with an outer diameter of 1 inch and a thickness of 0.125 inches. The material is Aluminum 2024, and the element group 'SHELL4' for a four-node thin shell element. For this particular case, a different sized mesh was created and can be seen below in figure 15.

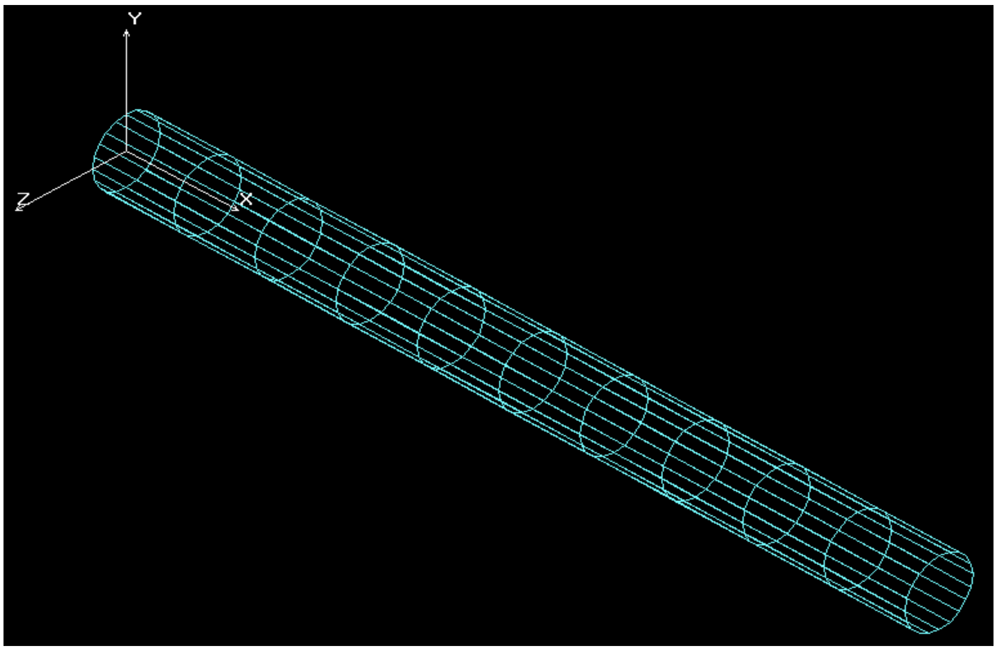

Figure 15: Compression Model Mesh

For this case the constraints are going to be different from the previous cases of 3-point bending and buckling. For the compression model, the end at the origin is going to be fixed while the other end is free to move in the z-direction. This is constraining the pipe to a fixed surface rather than compressing the pipe from both sides. Boundary conditions were also applied along the pipe on the horizontal axis to constrain the pipe from moving in the vertical direction, and along the vertical axis to constrain the pipe from moving in the horizontal direction. As one may notice, these are the same boundary conditions that were applied to the tension model. Figure 16 displays the constraints placed on this particular model and can be seen below.

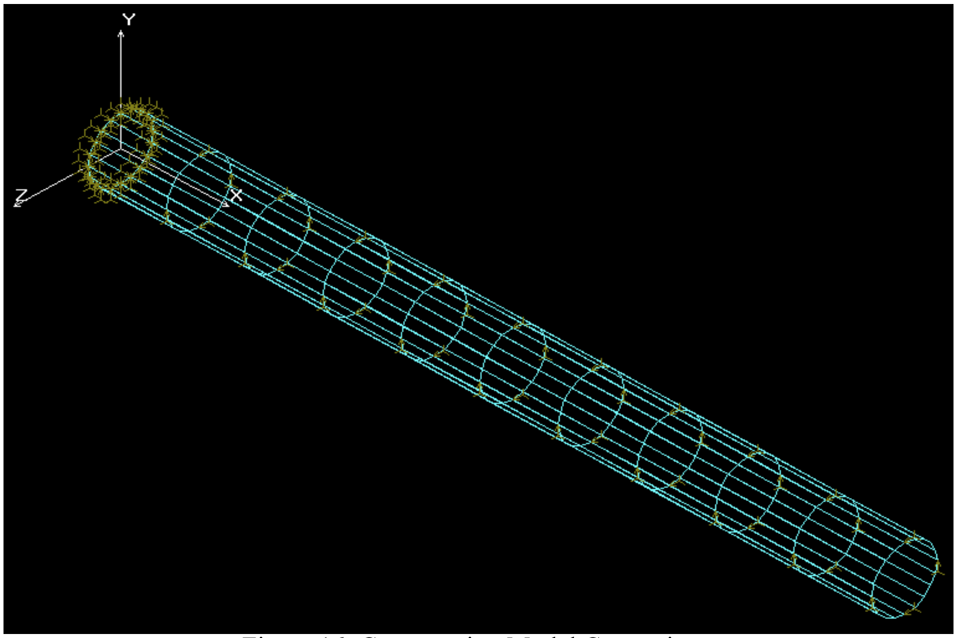

Figure 16: Compression Model Constraints

For this case the constraints aren't going to be the only thing that are different. For the compression case, a compression force needs to be applied, rather than a point load at the center in the 3-point bending case or a tension force for the tension case. As one may notice, the compression and tension cases are very similar except for the direction of the forces being applied. Figure 17 below shows the location and direction of the applied load. A load of 20 pounds is distributed among each node on the edge, resulting in a total of 20 nodes with 1 pound of force each. It should be noted that the applied load as well as the geometry of the pipe is arbitrary at this point, since only the process and boundary conditions for each model are trying to be validated.

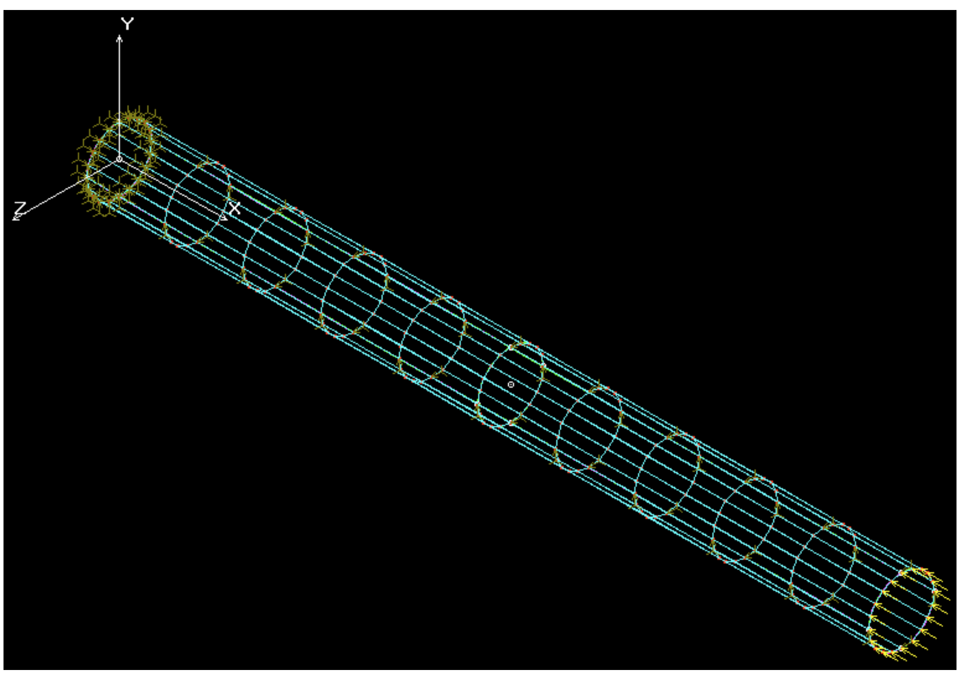

Figure 17: Compression Model Loading

The model for the structure has now been completed and the analysis can be performed. For this particular structure, static analysis will be done, therefore the static analysis options need to be set, and then the analysis can be

completed. Doing so will find the displacement and stresses at each node that was created in the mesh. The stresses and strains of every created element will also be calculated.

Now that the previously made model has been completed, and ran through the Cosmos/M solver we can look at the results. Cosmos/M provides an abundant amount of information such as deformation along with the stress and strain distribution of the model. Looking at the stress distribution in the pipe, figure 18, one can notice that the deformation is symmetrical. The loading and the boundary conditions are completely symmetrical therefore the stress distribution and deformation will be symmetrical around the z-axis. A complete list of every nodal and element stress can be provided upon request. One can also notice Poisson's effect taking place. The pipe is being compressed in the z-direction, therefore the pipe will expand in the x and y-direction for the section of the pipe that isn't constrained in those directions. This is also why there is a higher stress near the tips of the pipe when compared to the rest of the body. The smaller cross sectional area of the pipe leads to a higher stress than that of the rest of the pipe with a larger cross sectional area. This can be easily seen in the following section where the normal stress equations are shown in full detail.

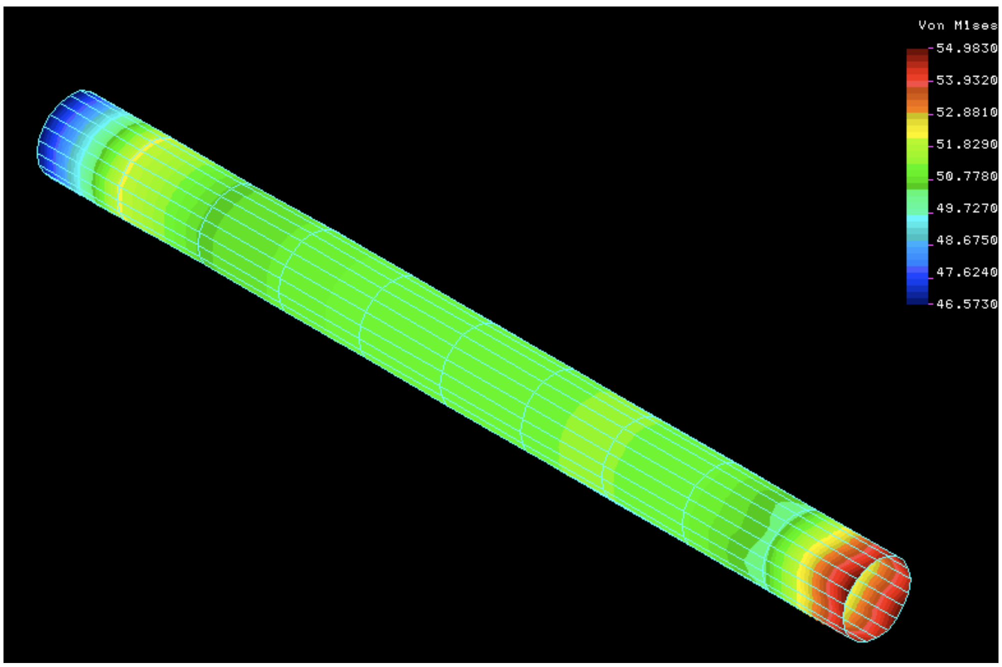

Figure 18: Compression Model Stress Distribution

IX. Compression Model Validation

Now that the model for the compression case has been made and analyzed, we must verify it to make sure that it is giving us the correct values. This can be done easily using theoretical analysis. In order to do this assumptions must first be made to simplify the analysis. First, the pipe loading is assumed to only be acting in the linear elastic region. In other words, the yielding stress has not been reached and thus no plastic deformation. Second, the beam is assumed to be bending in one plane. Essentially, this is only a 2 dimensional problem, so there is no twisting involved either. Third, the pipe is entirely isotropic, so the material is the same throughout the entire pipe. Fourth a 20 pound load is applied to the pipe of length 12 inches. Fifth, the pipe is initially straight before loading.

In a similar fashion to the tension validation, we assume that there is no stress due to bending, and that the only stress on the pipe is normal stress. Therefore we can use the following equation to determine the normal stress

$$\sigma = \frac{F}{A} \tag{13}$$

where F is the applied force, and A is the cross sectional area of the pipe. Using the equation for the area of a circle we can determine the cross sectional area of the pipe.

$$A = \pi\left(\left(\frac{1}{2}\right)^2 - \left(\frac{0.75}{2}\right)^2\right) = 0.343612 \text{ in}^2 \tag{14}$$

Using the cross sectional area and the applied force we can determine the normal stress on the pipe.

$$\sigma = \frac{F}{A} = \frac{20lbs}{0.343612in^2} = 58.2052psi \tag{15}$$

This stress is for a location on the pipe where the cross sectional area hasn't changed, therefore we must compare that value to one under the same conditions on the finite element model. With that being said, we can't use a nodal stress at the end of the pipe because it has started to deform, which results in a smaller cross sectional area. So looking at a node located at half the length of the pipe Cosmos/M outputs a nodal stress of 51.13 psi. If we use the percent error formula we find that the finite element estimation is approximately 12 percent different from the theoretical value.

X. Initial Models Comparison

After modeling all four cases and performing the analysis in the finite element program, they were validated by means of theoretical analysis. Table 1 below shows the percent difference between each models finite element estimation and what was determined using theoretical analysis. The largest percent difference was found to be approximately 23 percent, while the lowest percent difference was 5.5 percent. Looking at the percent differences, the finite element program has done an adequate job of estimating each case.

Table 1: Model Accuracy

Case	Percent Difference
3-Point Bending	5.5 %
Buckling	23 %
Tension	12 %
Compression	12 %

Now that we have found the percent difference between the theoretical and finite element analysis an attempt can be made to improve the models accuracy. In order to increase the accuracy of the model, the mesh will need to be dealt with for each case. If the mesh of the model is refined a more accurate result will be determined, therefore reducing the error. It should be noted though that at a certain point the computational time of the model would become unnecessary for the increase in accuracy. With that being said, numerous cases varying the number of elements would need to be done in order to determine the point at which the increase in computational time becomes unnecessary. Also the assumptions made in the theoretical analysis can be changed in order to model a more realistic case, such as that there is no shear in the 3-point bending case. At this point though, the percent difference isn't horrible, and each case modeled the way it is now may be used to estimate the mechanical behavior of the natural fiber composite to a reasonable degree.

XI. Frame Analysis

Using the same method of modeling the previous cases, the actual 2 – dimension bicycle frame can be created in Cosmos/M. The first step in modeling the frame is to define its geometry. A standard bicycle frame geometry was used and can be seen below in figure 19. Once the geometry has been defined, it can be created in the finite element program. After this is done, the appropriate element group can be set.

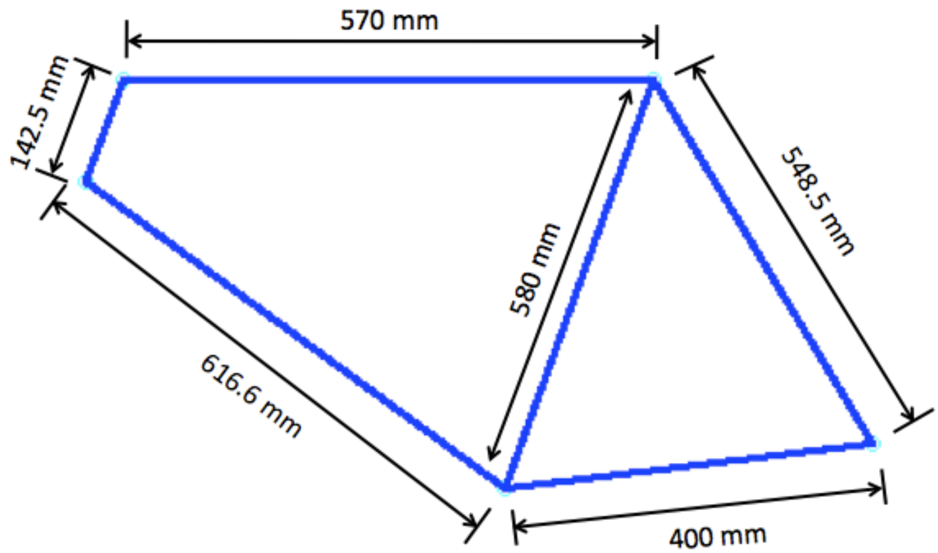

Figure 19: Frame Geometry

For this case the 'BEAM2D' element group will be used. The 'BEAM2D' will be used for this model due to the fact that a frame is being analyzed as well as the assumed 2 – dimension nature. The characteristics of the 'BEAM2D' element group include a linear elastic assumption, as well as a small displacement formulation. Once this element group and its default options have been set a 'Real Constant' must be applied due to the nature of the element. The real constant for this specific element group consists of the cross sectional area of the member as well as its moment of inertia. For this particular model, using the same 1-inch diameter and 0.125-inch thick pipe, the cross sectional area was set to 0.3436 in^2 and a moment of inertia of 0.03356 in^4. From here the material was selected to be Aluminum 2024, which was also used in the previously created models.

Now that the geometry, element group, real constant, and material properties have been set the mesh can be created, the boundary conditions/constraints and loads can be applied. For this model each tube of the frame is defined to consist of 10 elements. Once the elements and their respective nodes are created, the nodes must be merged in order to create a continuous frame structure. The reason for doing this is to ensure that the finite element software recognizes that each member is part of a larger, connected frame rather than individual members that are not connected and floating in space. A figure of the mesh and the elements/nodes can be seen below in figure 20.

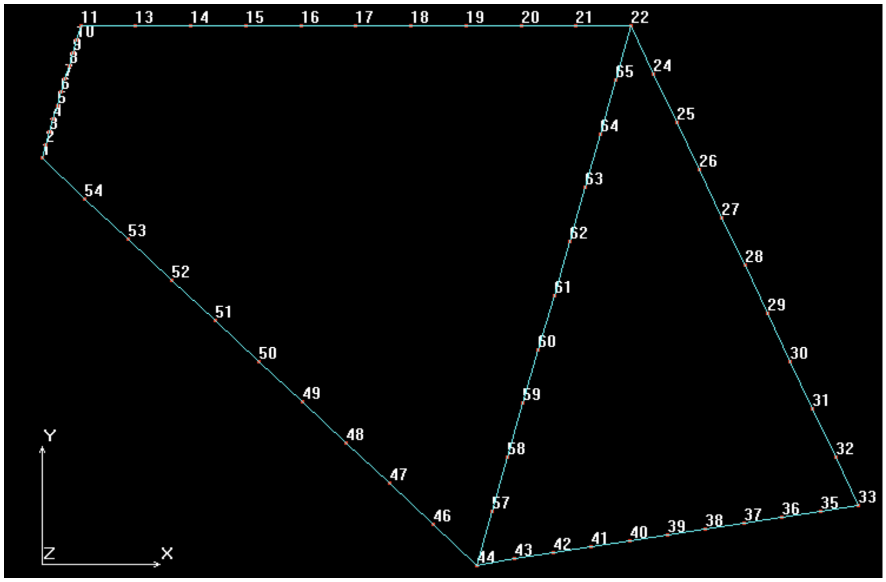

Figure 20: Frame Mesh

It should be noted that for this particular model only the frame will be analyzed, if the wheel/tire interaction were to be included different boundary conditions and elements would need to be added. Although, since the frame is only being looked at we can constrain the bicycle frame at the head tube and at the seat stay/chain stay connection. With that being said the frame will be constrained in the vertical direction at these locations, as well as the head tube being constrained in the horizontal direction. The single horizontal constraint will allow the bicycle frame to deform in the x-direction by ensuring that the frame doesn't translate during loading. The last set of constraints that will be applied will be located at each tube connection. At these locations the frame will be constrained in the z-direction to ensure that there won't be any twisting due to the loading and that the model remains a 2-dimensional analysis. The previously stated constraints can be seen applied to the model in figure 21 below.

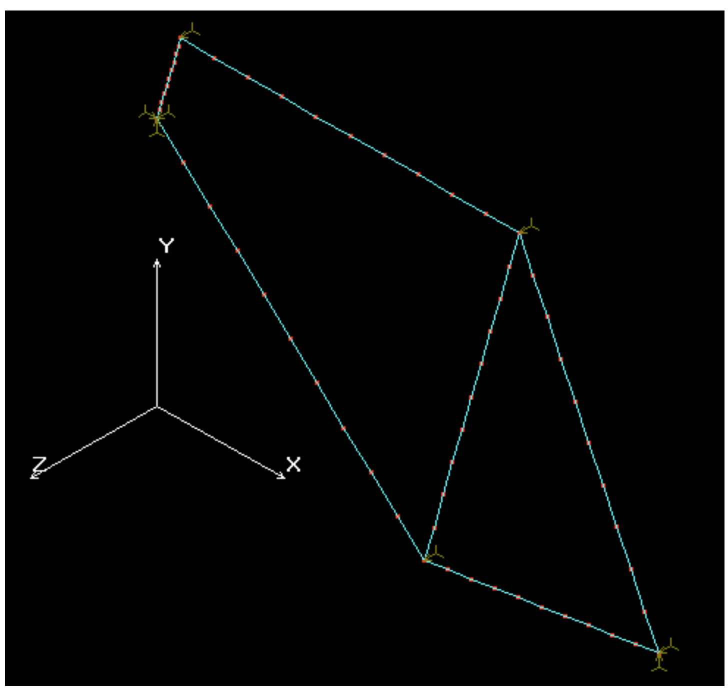

Figure 21: Frame Constraints

With the boundary conditions and constraints now applied the loads can be applied to the model. The case that will be analyzed is that of an arbitrary 100 pound load on the head post as well as a 100 pound load on the seat post. This loading is to simulate a rider sitting on the bicycle while also leaning forward on the handlebars. With the case now established, two vertical loads of 100 pounds each are applied to the previously stated locations. Figure 22 shown below, displays the applied loads on the bicycle frame and their locations.

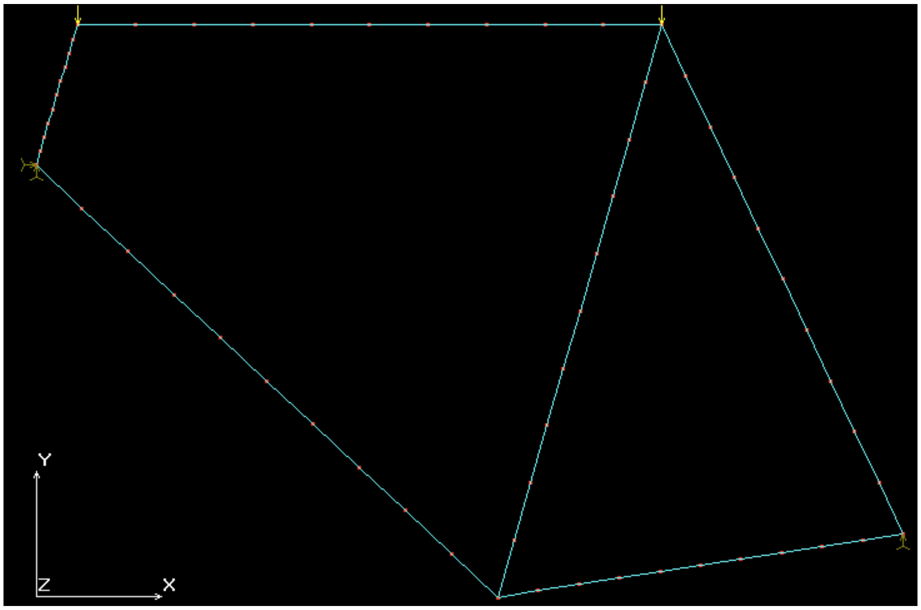

Figure 22: Frame Loading

Now that all of necessary steps have been taken to create the model it can be analyzed. For this particular model static analysis will be performed. Therefore, in order to complete the analysis the static analysis options are set to their default values and settings and the model can be ran through the Cosmos/M solver. In this process the displacement, response, and reactions are calculated for each node as well as the shear/moment values and beam end force. The beam end force values will be useful in the comparison of the finite element results to theoretical results in order to validate the model. After the analysis is completed plots of the deformed frame can be shown. This deformed frame shape can be seen below in figure 23.

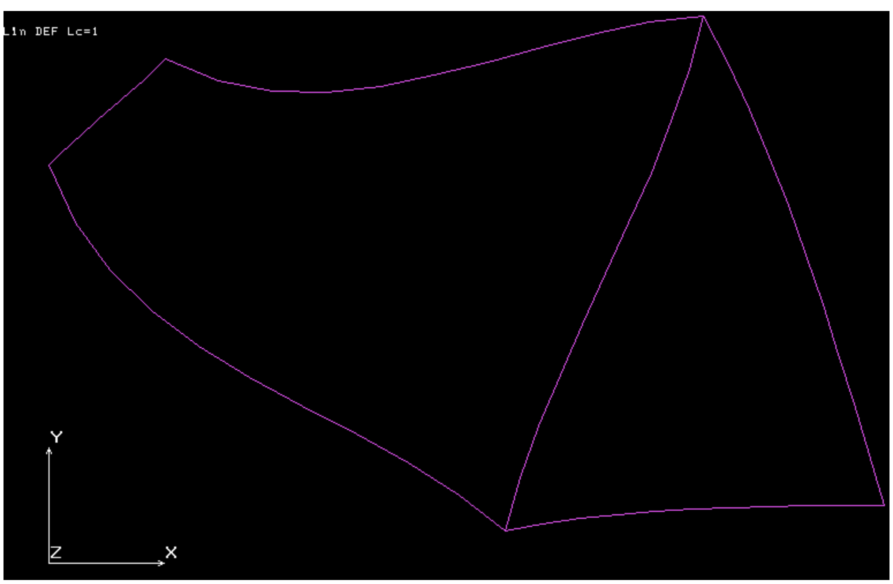

Figure 23: Deformed Frame

XII. Frame Validation

Now that the model for the frame has been made and analyzed, we must verify it to make sure that it is giving us accurate results. This can be done easily using theoretical analysis. The axial loads in each member of the frame will be analyzed in order to validate the finite element model. In order to do this assumptions must first be made to simplify the analysis. First, the pipe loading is assumed to only be acting in the linear elastic region. In other words, the yielding stress has not been reached and thus no plastic deformation. Second, the beam is assumed to be bending in one plane. Essentially, this is only a 2-dimensional problem, so there is no twisting involved either. Third, the pipe is entirely isotropic, so the material is the same throughout the entire pipe. Fourth a 100 pound load is applied to the head tube and seat post. Fifth, the tubes are initially straight before loading.

To determine the axial loads in each member, a code written in Matlab was created. In this code the geometry of the frame was inputted as well as the same loads applied to the finite element model. Once this is inputted, the geometry and loading is used to determine the reactions at both of the vertical constraints. This is done simply by using the sum of the moments and sum of the forces and setting them equal to zero for static analysis.

$$\sum Forces = 0 \tag{16}$$

$$\sum Moments = 0 \tag{17}$$

After this is applied the theoretical reactions are found to be 123.0351 pounds and 76.9649 respectively for the head tube and chain stay/seat stay connection. Comparing these values to that outputted by the analyzed finite element model, 123 pounds and 76.96 pounds for the head tube and chain stay/seat stay connection respectively, a percent difference of 0.03 and 0.006 is found. So far in the sense of determining the reaction forces of the model the finite element software is doing a superb job.

Moving onto the comparison of the theoretical and finite element axial forces requires more work. The Matlab code previously stated uses a method of joints in order to determine the axial forces of each member and gives a singular value. Although, Cosmos/M displays the axial stress as components, therefore the magnitude of each members respective components needs to be determined. This can be done easily by using equation 18 shown below.

$$|F| = \sqrt{(F_x)^2 + (F_y)^2} \tag{18}$$

After employing the previous equations and determining the resulting magnitudes of the axial stress, that stress is multiplied by the cross sectional area of the tubes. This cross sectional area was defined earlier when the real constant for the 'BEAM2D' element group was set. We can multiply the axial stress by the area to determine the forces by rearranging the axial stress equation.

$$\sigma = \frac{F}{A} \tag{19}$$

Once this is completed the theoretical and finite element results for the axial force in each member can finally be completed. With that being said the percent difference between the two ranged from 4.12 percent to 11.4 percent. All of the results can be seen below in figure 24 and table 2.

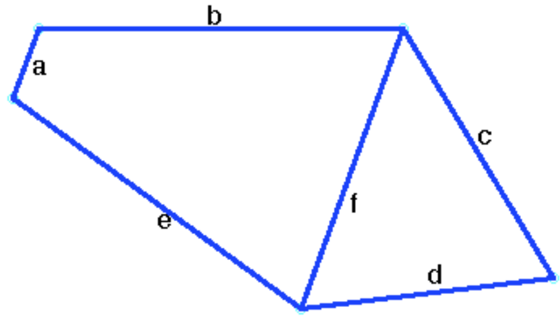

Figure 24: Frame Members

Table 2: Frame Member Axial Load Comparison

Member	Percent Difference
a	4.13 %
b	6.69 %
c	8.75 %
d	8.56 %
e	8.03 %
f	11.43 %

XIII. Applied Bamboo

Now that the previous cases have been ran and verified using known materials, the models can now be altered to analyze the natural, organic materials. Before this was done though, extensive research was performed in order to determine viable material characteristics for bamboo. In this process many types of bamboo were found, leading to various characteristic values for each species and type of bamboo. The bamboo that will be investigated and used in this report is that of Moso bamboo. The material characteristics for this particular species were found to be 13.85 GPa or 2.009 Msi for the longitudinal elastic modulus and 1.69 GPa or 245.1 ksi for the circumferential elastic modulus[1]. The longitudinal-circumferential Poisson's ratio was found to be 0.30[1]. A new mode of analysis will also be used in order to determine more accurate results as well as a prediction for how the bamboo tubes will deflect when loaded. The particular analysis that will now be employed is non-linear analysis. The type of analysis in Cosmos/M allows the user to define a load-time curve in which the applied load can be varied with time. This is beneficial because the user can more accurately model what would take place in experimental test. The applied load is increased at some defined rate rather than the load being instantaneously applied, which is the case with static analysis. The non-linear analysis also allows us to see how the material deflects over time as the load is being applied. With this type of analysis there does not need to be a linear deflection curve, and it can be rather non-linear.

XIV. Bamboo 3-Point Bending Model

The applied bamboo and non-linear analysis will be completed in a way similar to that of the static analysis cases. For these models, the geometry, element group, any real constants, and material properties will be defined first. For this model the same element group of 'SHEL4' will be used. The 'SHELL4' element characteristics include a 2-dimensional element that is linear as well as small displacement. Once this is set, the real constant can be defined in the same way as earlier. For this model the element thickness will remain 0.125 inches.

With the geometry, element group, and real constant set the material properties as well as the mesh can be defined. The material properties that need to be defined will be the modulus of elasticity in the x and z direction as well as the Poisson's ratio for the x-z, y-x, and y-z directions. These material properties can be found and taken from research papers as stated earlier. After the material characteristics are defined the mesh can be set. For this case the

mesh was set to have 10 elements in the longitudinal direction, while having 20 elements around the perimeter of the tube. Figure 25 shown below displays the mesh applied to the 3-point bending case.

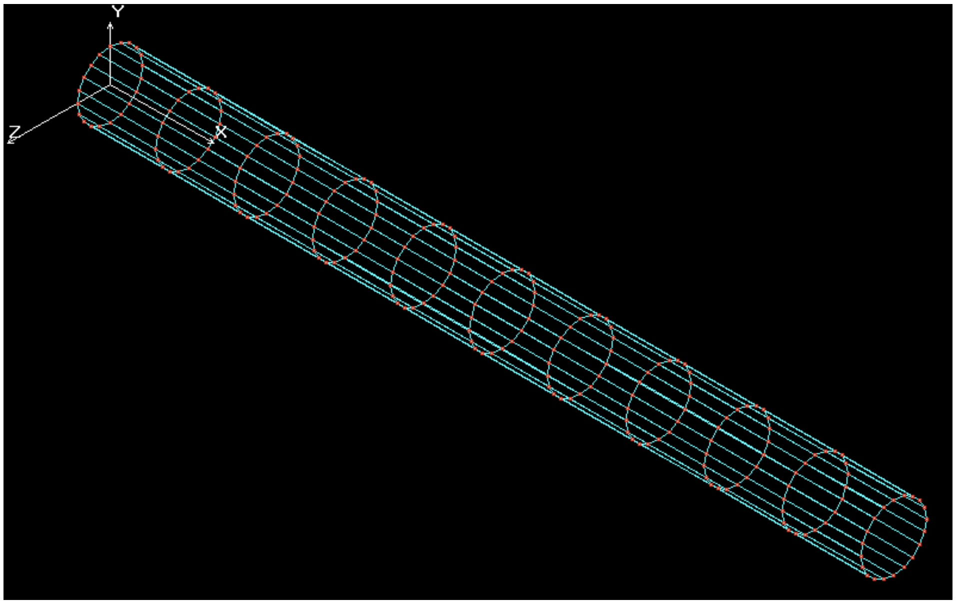

Figure 25: Bamboo 3-Point Bending Mesh

The next step in creating the non-linear analysis model is to define the boundary conditions. The boundary conditions for the bamboo 3-point bending model will be the same as the earlier 3-point bending case. The tube will be constraint at both edges in the y-direction, which simulate the two supports in a 3-point bend test. The tube will also be constrained at the vertical centerline to restrict the center of the tube from translating from its initial location. A rotation boundary condition will also be set to ensure that the tube does not twist allowing for pure bending. These boundary conditions can be seen in figure 26.

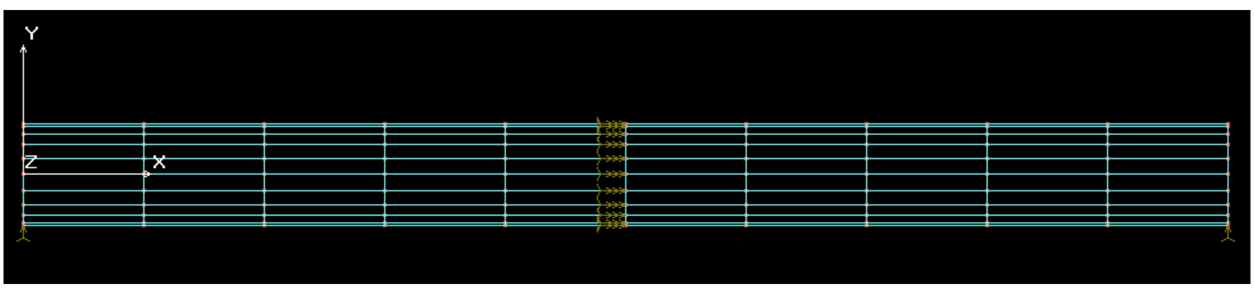

Figure 26: Bamboo 3-Point Bending Constraints

From this point is where the usual process of creating the finite element model changes. In order to perform a non-linear analysis the load-time curve must be defined. To do this the interval of time is first set. For this case the time will range from 0 to 100 in increments of 1 second. With this set the respective loads at each time can be set. Again, for this model there will be no load at the initial 0 second condition but will linearly increase to 1,000 pounds at 100 seconds. With the load-time curve set it is then activated. From here the location of the load will be set. It is crucial that the load-time curve is set and activated before the location of the load in order for the finite element program to recognize where the time dependent load will be applied on the model. With that being said, the load will be applied at the center of the top surface of the tube. It should be noted that the applied load is set to a unit load of 1 pound. Figure 27 displays this below.

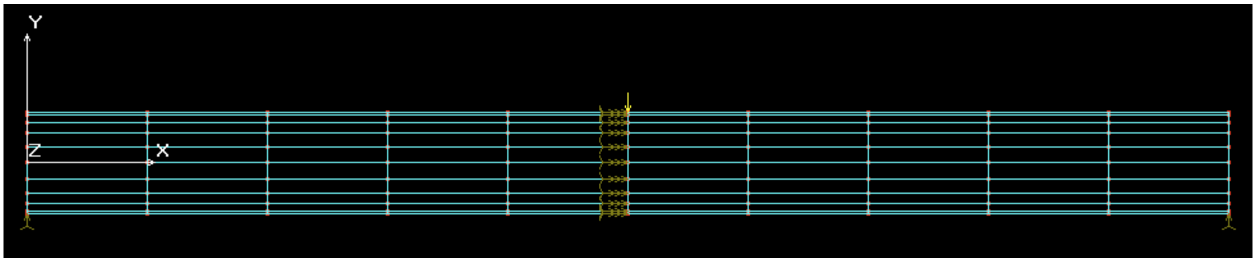

Figure 27: Bamboo 3-Point Bending Load

At this point the model can be ran through the Cosmos/M solver and results can be found. For this particular analysis the nodal and elemental stress and strain distributions as well as the deflection can be seen for every time interval that has been set in the load-time curve. Figure 28, shown below displays the stress concentration at the 100^{th} step. At this point a 1,000 pound load has been applied to the tube.

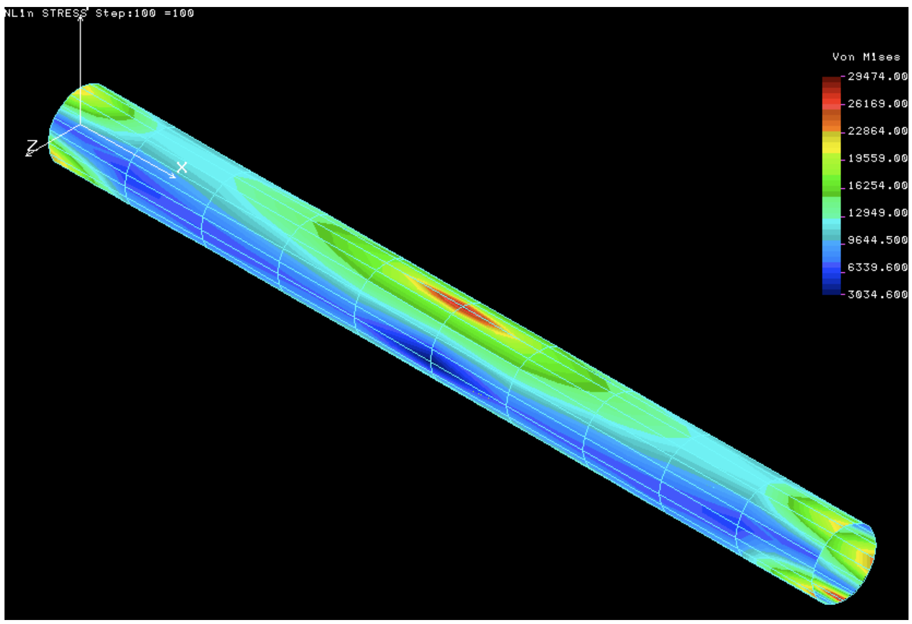

Figure 28: Bamboo 3-Point Bending Stress Distribution

XV. Bamboo 3-Point Bending Validation

Now that the model for the bamboo 3-point bend case has been made and analyzed, an attempt must be made to verify it to make sure that the model is giving us the correct values. This can be done using theoretical analysis. For this case the stress distribution should not be effected by the new material properties, therefore a similar approach to the original 3-point bending model will be used. In order to do this assumptions must first be made to simplify the analysis. First, the pipe loading is assumed to only be acting in the linear elastic region. In other words, the yielding stress has not been reached and thus no plastic deformation. Second, the beam is assumed to be bending in one plane. Essentially, this is only a 2 dimensional problem, so there is no twisting involved either. Third, the pipe has the same dimensions of 12 inches long with a 1 inch diameter that is 0.125 inches thick. lastly, the pipe is initially straight before loading.

In the case of the 3-Point Bending model, it is assumed that the stress is caused entirely by bending. This stress can be found using equation 1. Similarly the centroid and moment of inertia can be found using equations 2 and 3. For this validation, the 100^{th} time step will be analyzed. With that being said the load will be 1000 pounds. The moment is simply the reaction at one end, 500 pounds, multiplied by the moment arm. For the point of interest, and the point that we will be comparing/validating, the moment arm is 6 inches since it is located at the center of the pipe. Therefore, the moment for this particular location is found to be 3,000 in-pounds. Also for the point that is going to be compared, the distance from the neutral axis is going to be 0.5 inches since it is located at the outer edge

of the pipe and the centroid is at the center of the pipe. The culmination of these values resulted in a bending stress of 44701.4 pounds per square inch, shown below.

$$\sigma = \frac{Mc}{I} = \frac{(3{,}000in * lb)(0.5in)}{(0.033556in^4)} = 44{,}701.4\frac{lb}{in^2} \tag{20}$$

We can now use this value to compare what was found using the finite element software. If we look at the list of stresses found for the 3-Point Bending model at the 100th time step it is found that the stress due to bending is 29,474 pounds per square inch, at the half span of the pipe on the outer edge. Using the percent error formula the percent error to be 34 percent. This error may be attributed to the different type of analysis. For this model non-linear analysis was used rather than static.

XVI. Bamboo Tension Model

For the bamboo tension model the same method for the 3-point bending was used. The geometry was defined, 12 inch length and 1 inch diameter, than the element group, 'SHELL4' and respective real constants, 0.125 inch thickness. From here the material characteristics for bamboo are inputted to the finite element software. At this point, the mesh for the model can be generated. For this particular model the mesh was made to consist of 10 elements along the longitudinal axis and 20 elements along the tubes cross sectional perimeter. The next step is to define the boundary conditions for the model. For the tension model the pipe was constrained in all directions at one end. Boundary conditions were also applied along the pipe on the horizontal axis to constrain the pipe from moving in the vertical direction, and along the vertical axis to constrain the pipe from moving in the horizontal direction. The constraints for this model can be seen in figure 29 below.

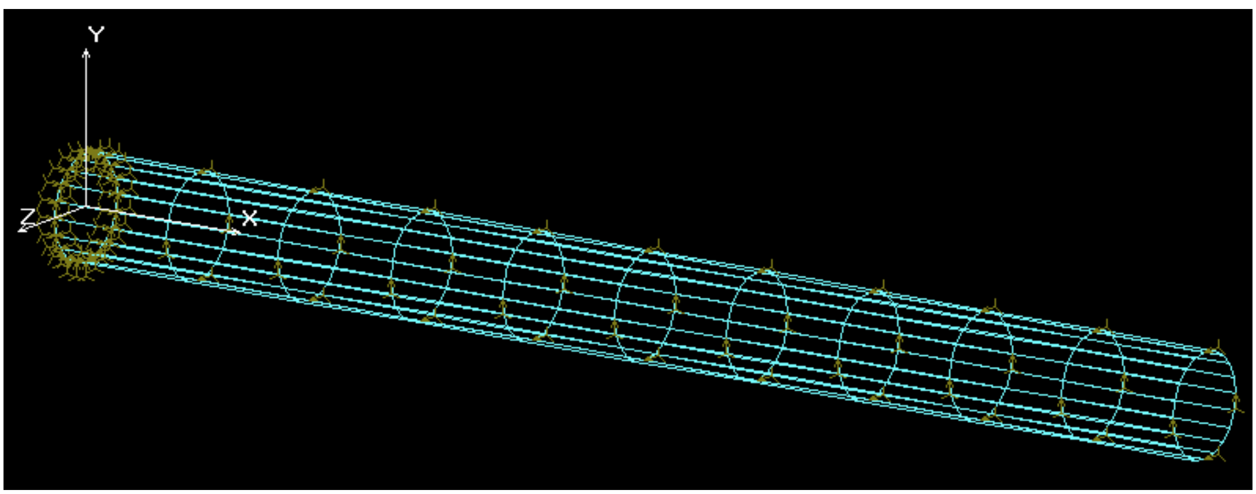

Figure 29: Bamboo Tension Model Constraints

The next step is to define the load-time curve for the model. The load-time curve for the bamboo tension model will be the same as the bamboo 3-point bending model. The time spans from 0 to 100 in increments of 1 second, with the load being zero at the initial step and 1,000 pounds at the 100th increment. The load-time curve is activated and the tension load is then applied to the model at each node on the unconstrained edge. Each load is made to be 1/20 of a pound, which makes a total load of 1 pound over 20 nodes. At this point the model is completed and can be ran through the Cosmos/M solver. Figure 30 shows the tension model with the loads applied.

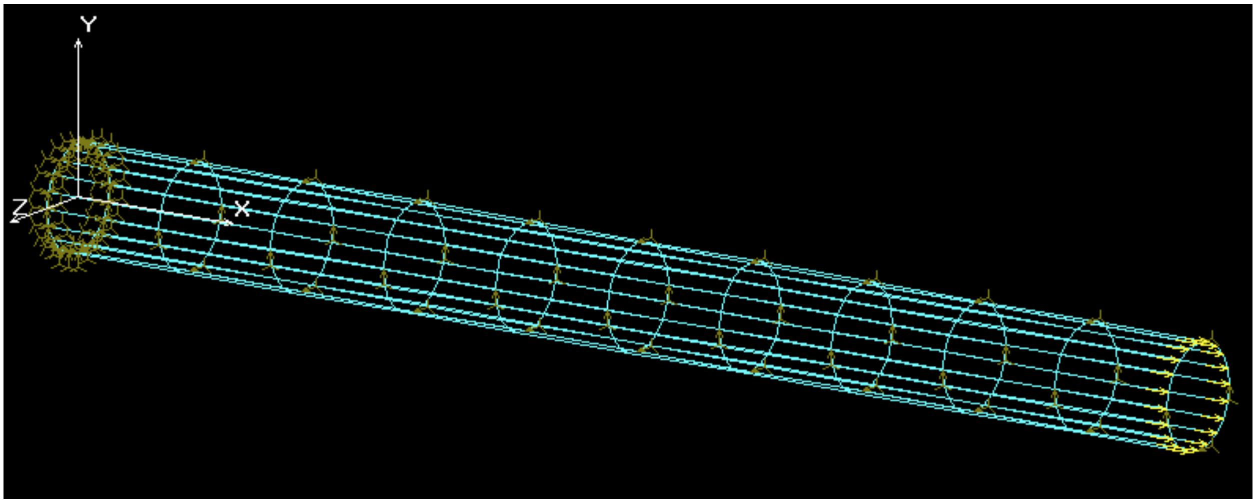

Figure 30: Bamboo Tension Model Loading

With the model completed and ran through the Cosmos/M solver the results can be investigated. Figure 31 below shows the stress distribution in the tube due to the tension load at the 100th time step, which is a 1,000 pound load. Again, symmetry can be seen throughout the model as well as a general decrease in stress from where the load is applied to the fixed end. There is an interesting area of a lighter stress seen by the orange area though.

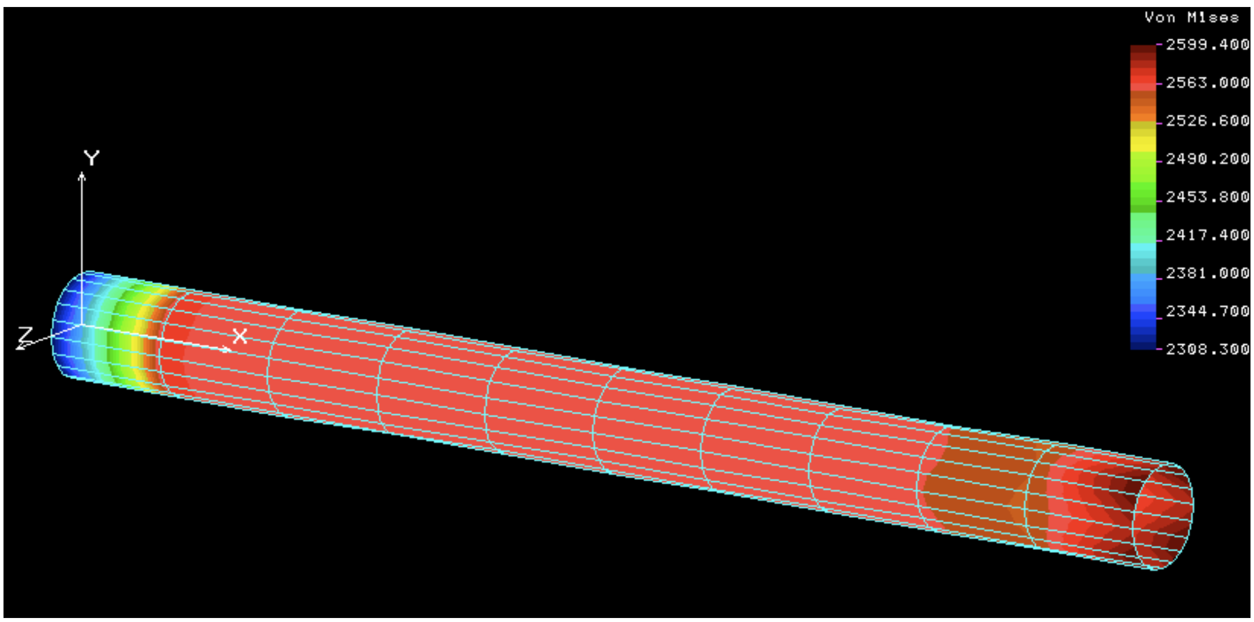

Figure 31: Bamboo Tension Stress Distribution

XVII. Bamboo Tension Validation

The validation for the bamboo tension model will be done in the same manner as the original tension model. As stated earlier, the material characteristics will not effect the tensile stress of the tube. With that being said, the same assumptions will be used. To restate, the pipe loading is assumed to only be acting in the linear elastic region. In other words, the yielding stress has not been reached and thus no plastic deformation. Second, the beam is assumed to be bending in one plane. Essentially, this is only a 2 dimensional problem, so there is no twisting involved. Third, the pipe has the same dimensions of 12 inches long with a 1 inch diameter that is 0.125 inches thick. lastly, the pipe is initially straight before loading.

For this validation, the 100th time step of a 1,000 pound will be used. Using equation 10 the tensile stress can be found. In this case F will be 1,000 pounds while the area will remain the same.

$$\sigma = \frac{F}{A} = \frac{1{,}000 lbs}{0.34612 in^2} = 2{,}889.2 \frac{lb}{in^2} \tag{21}$$

This stress is for a location on the pipe where the cross sectional area hasn't changed, therefore we must compare that value to one under the same conditions on the finite element model. With that being said, we can't use a nodal stress at the end of the pipe because it has started to deform, which results in a smaller cross sectional area. So looking at a node located at half the length of the pipe Cosmos/M outputs a nodal stress of 2,563 psi. Using the percent error formula it is found that the finite element estimation is approximately 11.3 percent different from the theoretical value.

XVIII. Bamboo Compression Model

The bamboo compression model will be relatively easy to develop since it is essentially the same as the tension model. With that being said, everything will remain the same with the exception of the direction of the applied load. Since this is a compression case, the load will be in compression rather than tension. This is shown in figure 32. From here the model is ran through the Cosmos/M solver and the results are outputted.

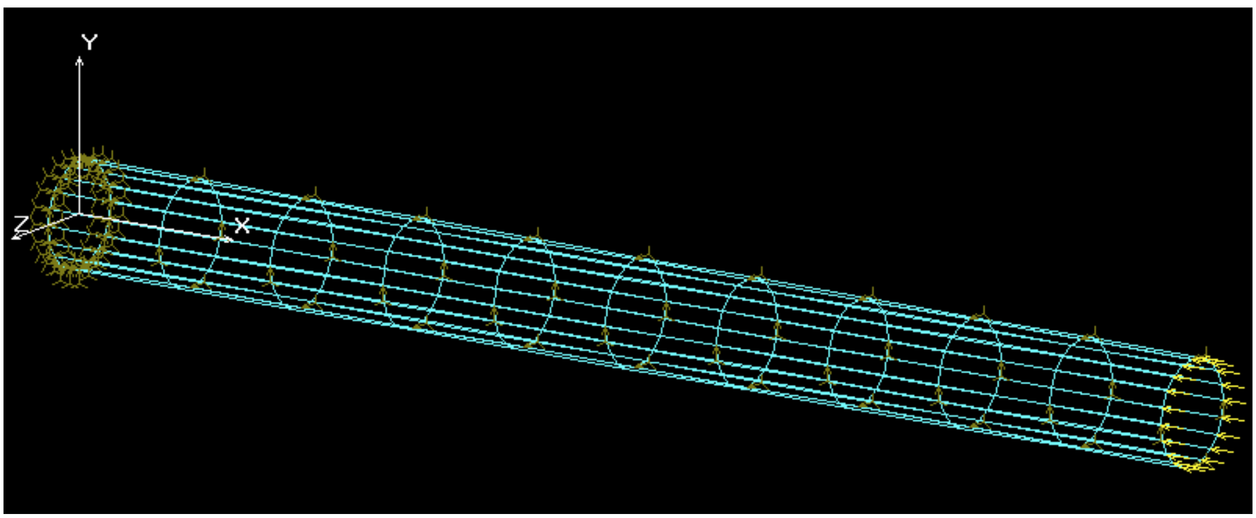

Figure 32: Bamboo Compression Model Loading

With the model ran, the stress distribution can be shown. As expected the stress distribution for the compression case is the same as the tensile case. The stress distribution for the 100th time step can be seen below in figure 33. The stress distribution is symmetrical as well as decreasing from the loaded end to the constrained end. This makes sense because the constrained end will begin to have a larger cross sectional area, resulting in a smaller stress value.

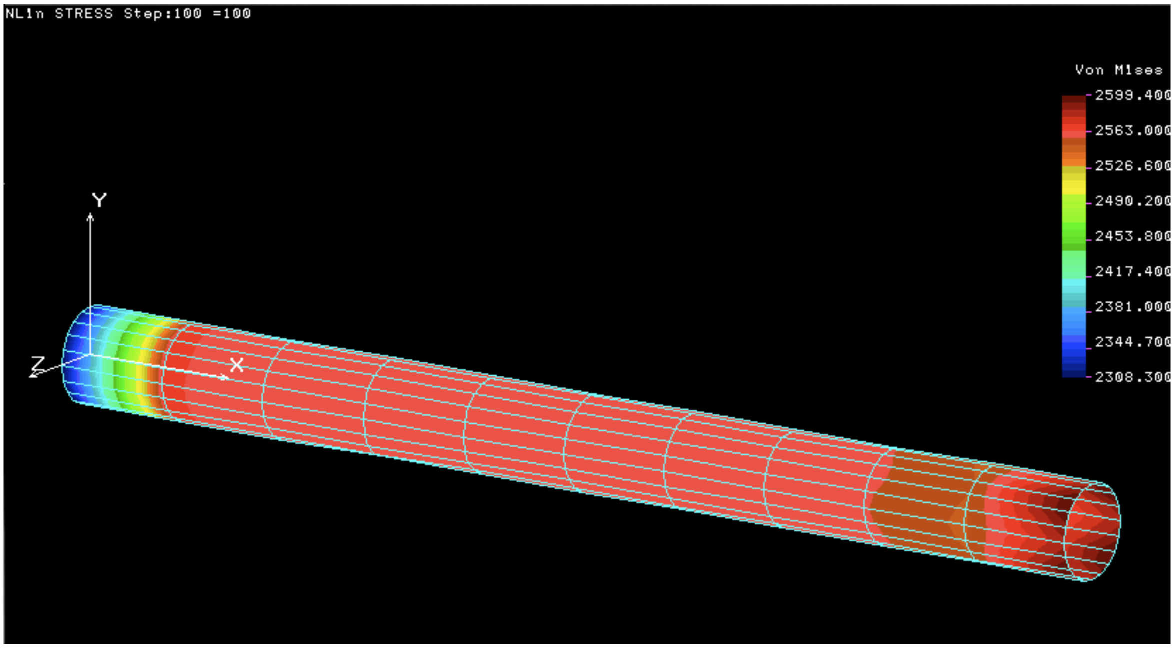

Figure 33: Bamboo Compression Model Stress Distribution

XIX. Bamboo Compression Validation

The validation for the compression model will be essentially the same as the tension model. Using the same assumptions as previously stated and the same equations the theoretical stress is found to be 2,899.2 psi. This is shown in equation 22.

$$\sigma = \frac{F}{A} = \frac{1{,}000\,lbs}{0.34612 in^2} = 2{,}889.2 \frac{lb}{in^2} \tag{22}$$

This stress is also for a location on the pipe where the cross sectional area hasn't changed, therefore we must compare that value to one under the same conditions on the finite element model. Looking at a node located at half the length of the pipe Cosmos/M outputs a nodal stress of 2,563 psi. Using the percent error formula it is found that the finite element estimation is approximately 11.3 percent different from the theoretical value, the same as the tension case.

XX. Bamboo Model Comparison

After modeling the cases and performing the analysis in the finite element program for the bamboo material, they were validated by means of theoretical analysis. Table 3 below shows the percent difference between each of the bamboo models finite element estimation and what was determined using theoretical analysis. The largest percent difference was found to be approximately 34 percent, while the lowest percent difference was 11.3 percent. It should be noted that the buckling case was not done for bamboo since there isn't a buckling option within the non-linear analysis.

Table 3: Bamboo Model Comparison

Case	Percent Difference
3-Point Bending	34 %
Tension	11.3 %
Compression	11.3 %

With the bamboo cases verified we can now look at the deflection characteristics of each case over time. These plots are useful since they show how the structure will deflect due to the time dependent load.

XXI. Deflection Characteristics

Since the bamboo models have been verified to be relatively accurate, we can expect the following deflection curves to be fairly representative of what would actually occur. The time-deflection curves for all three cases, 3-point bending, tension, and compression are shown bellow in figures 34, 35, and 36 respectively. For the 3-point bending case the deflection in the y-direction is plotted for the node located at the center of the top surface of the pipe, where the load is applied. The tension and compression cases plot the deflection of a node on the loaded end of the tube in the x-direction. These plots only display the deflection for each case to a load of 1,000 pounds. If a larger load is desired the load-time curve can be reset to take into account the desired load. By increasing the load, one may investigate the bamboo deflection throughout the entire linear elastic region. Also, the finite element software is capable of analyzing the bamboo's plastic characteristics, although additional material characteristics would need to be entered. For this project though, only the linear elastic region up to a load of 1,000 pounds has been investigated. A note to be made about the tension and compression deflections is that they are equal and opposite of each other. The tension case elongates the same amount as the compression case compresses due to the time dependent load.

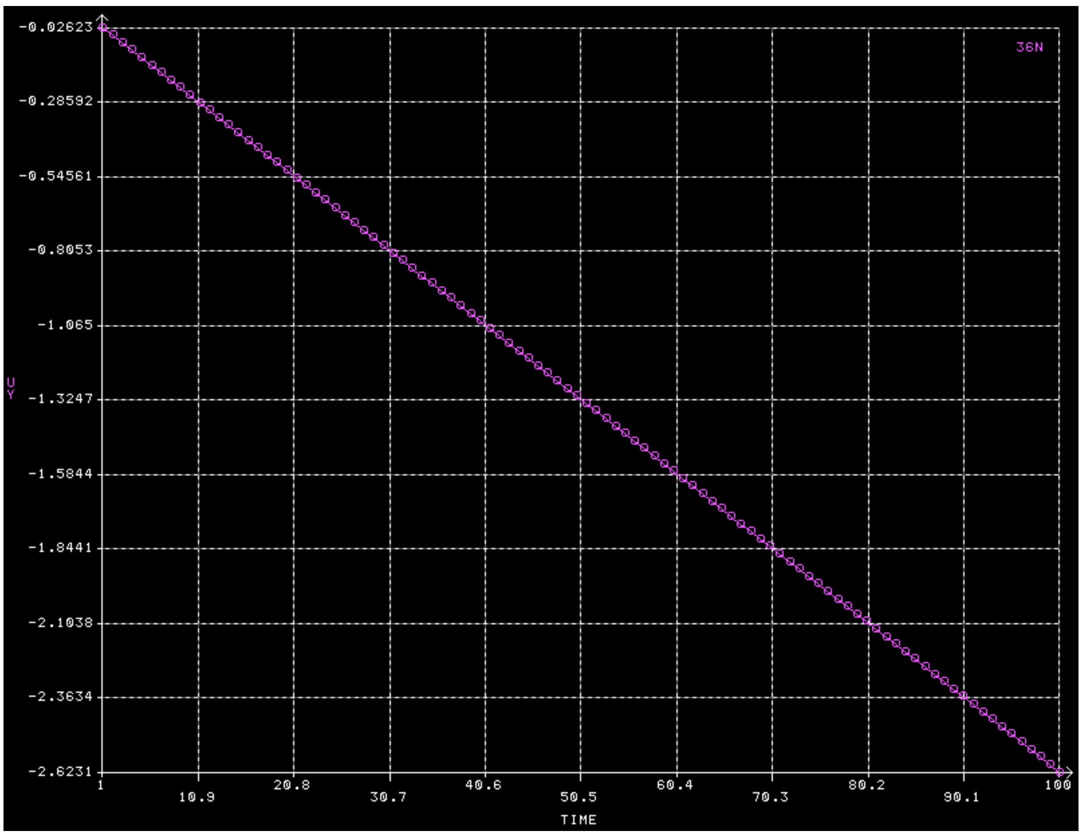

Figure 34: Bamboo 3-Point Bending Deflection Curve

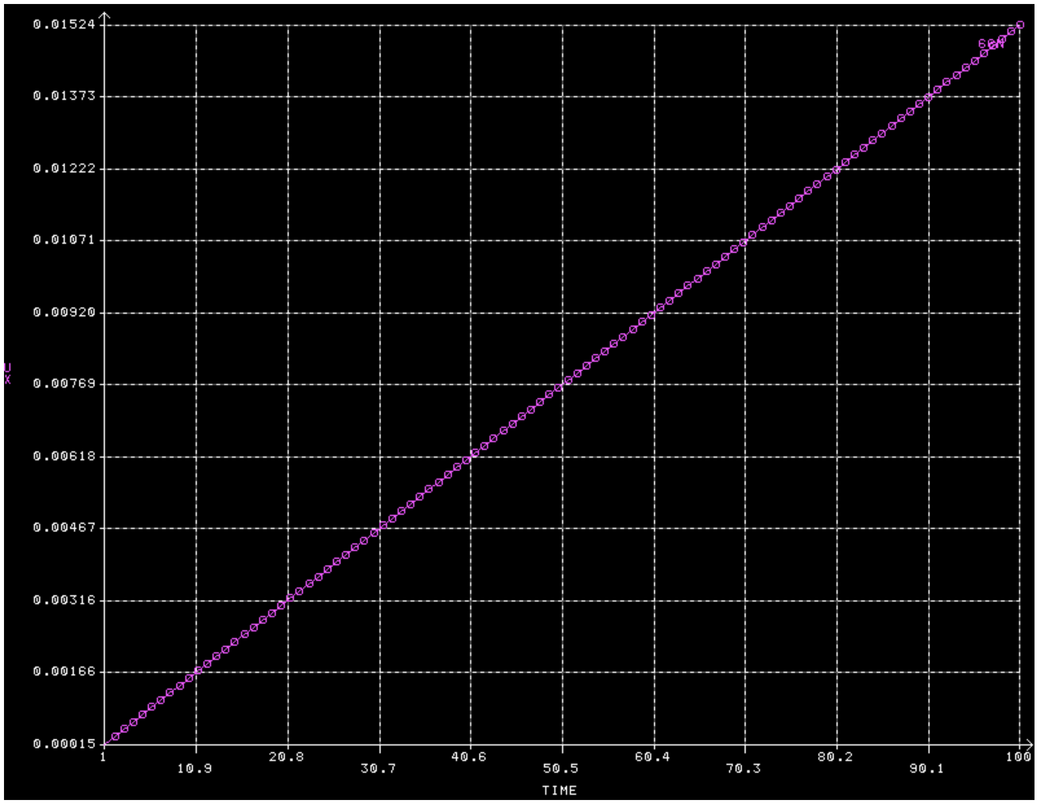

Figure 35: Bamboo Tension Deflection Curve

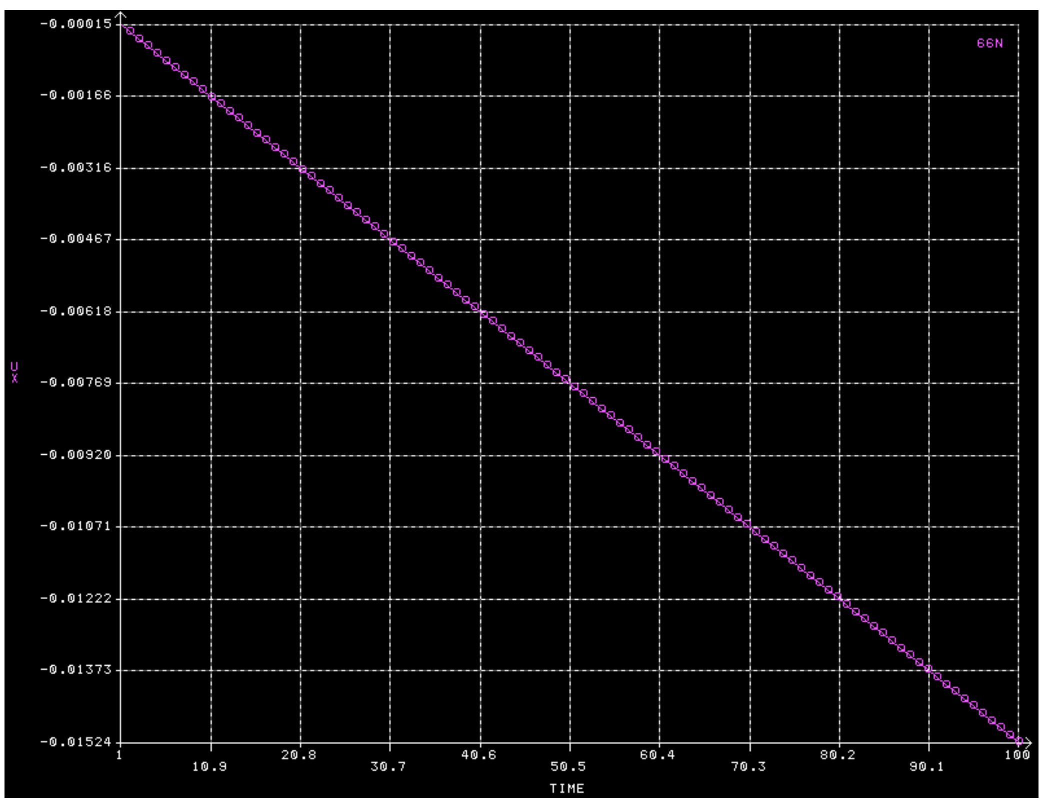

Figure 36: Bamboo Compression Deflection Curve

XXII. Conclusion

To conclude, the finite element analysis method of estimating the mechanical behaviors for each case resulted in a maximum percent difference of 34% and a minimum percent difference of 5.5%. As stated earlier, efforts may be made to make the models more accurate. The meshes for each case will be refined to try and achieve a higher degree of accuracy, as well as using more realistic assumptions for the theoretical analysis. At this point though, the accuracy of each model is decent and may be taken forward if a better model cannot be made. Using the bamboo material characteristics the stress distributions and deflection curves for each of the cases were found. For the applied bamboo cases though, a non-linear analysis approach was taken in order to find more accurate results that closely model what would happen during experimental test. The deflection curve for each case was plotted against time which includes a linearly increasing load from 0 pounds to 1,000 pounds. From these plots we are able to see how the bamboo material will react to the applied load. This gives us the ability to determine if the found deflections are allowable.

XXIII. References

[1]Elghandour, Eltahry. “Aero 433 Lab Procedures” Lab Procedure.
[2]Megson, T.H.G. “Aircraft Structures for Engineering Students” 4th Edition. Elsevier Ltd. 2007.
[3]Simon, Frederic. Kutnik, Magdalena. Le Bayon, Isabelle. Gatimel, Vincent. Moisset, Gerald “Evaluation of an Optimized Industrial Process Aiming to Improve the Natural Durability and Mechanical Behavior of Bamboo for Outdoor Use” PowerPoint. Queenstown. 2011.
[4]Garcia, J.J. Ghavami, K. Torres, L.A. “A Transversely Isotropic Law for the Determination of the Circumferential Young’s Modulus of Bamboo with Diametric Compression Tests”. Pontificia Universidade Catolica do Rio de Janeiro, Brasil. 2007.
[5]Bhalla, Suresh. “Scientific Design of Bamboo Structures” Indian Institute of Technology. New Delhi. 2007.
[6]Kumar, Rakesh. Jain, Seema. “Mechanical Behaviour of Bamboo and Bamboo Composite” Indian Institute of Technology. New Delhi. 1992.
[7]Naik, NK. “Mechanical and Physico-Chemical Properties of Bamboos” Indian Institue of Technology. Bombay.