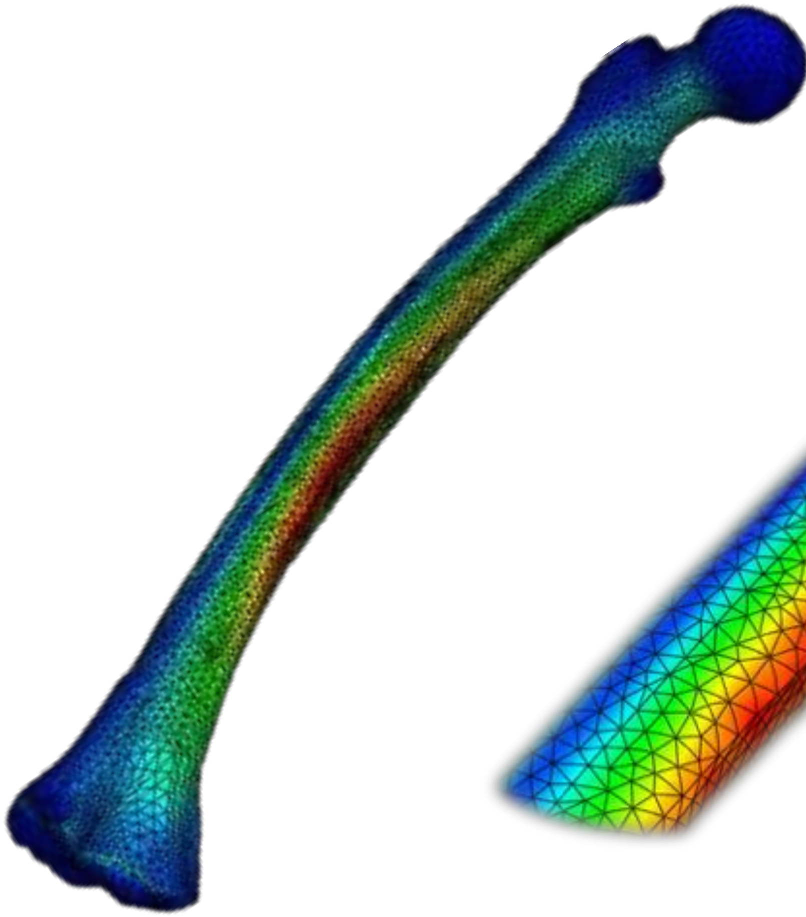

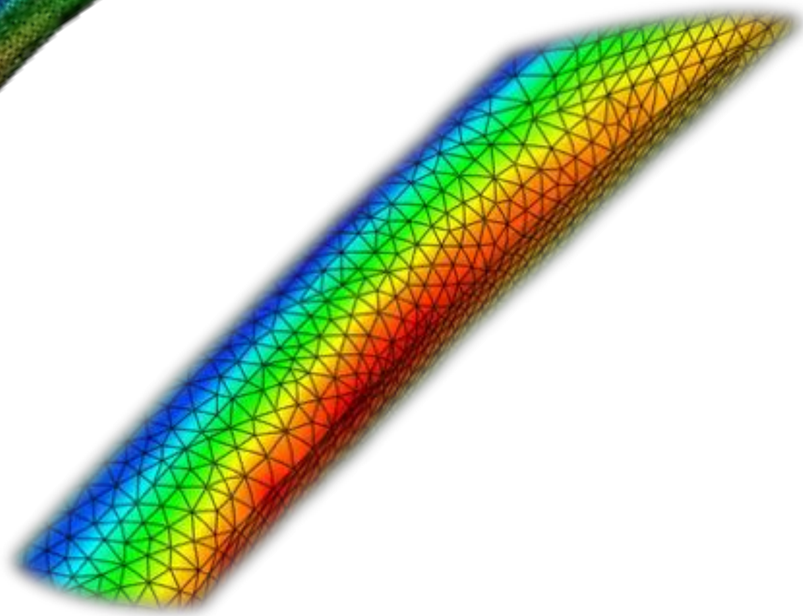

ME 461: Finite Element Analysis

The Development and Analysis of Human Femur Bone

Group Members:

Beytullah Aydogan, Eric Swartz, Ryan Tracy

PennState College of Engineering

Department of Mechanical and Nuclear Engineering, University Park, PA

Table of Contents

Executive Summary

We will be evaluating a human femur bone using ABAQUS. The overall goal of this project is to analyze and compare the stresses, strains, and displacements that occur on the top and bottom faces of the femur. The femur is described as the bone that extends from the hip to the knee and it is a very important bone in human body. It is the longest, heaviest, and strongest bone in entire body which means that there can be lots of stresses and strains applied to it at any given time. Moreover, the femur supports all of the body's weight during many activities like running, jumping, walking, and standing. Therefore, extreme forces act on the femur which makes it classified structurally as a major component of the appendicular skeleton [1].

The team decided to perform three experiments on the femur bone in ABAQUS. The first analysis consists of a vertical surface force applied at the head of the bone with the bottom of the bone being fixed. The value of the force applied was 2500N and was found from an on line reference [11]. The next analysis involved cutting the femur bone in half and fixing the cut-off end while applying pressure forces to the head of the femur. Finally, the last analysis used the whole femur as well as a block that was created. The block was given a displacement so that it would cause a reaction on the femur. In each experiment, stress and displacement were measured and compared to the other experiments as well as any online resources that were used.

Acknowledgements

Due to the extent of work and complexity of this project, our team would like to thank Professor Kraft for his help and guidance along the way. Without him we would not have gotten accurate results and our knowledge of finite element analysis would be minimal. Thank you again.

List of Figures

Section 1: Background and Project Plan

As stated earlier, we will be evaluating a human femur bone. The overall goal of this is to evaluate the stress and strain that occurs on the top and bottom faces of the structure. The femur is known as the bone of the thigh or upper hind limb. It is very important bone in the human body. It is also the longest, heaviest, and strongest bone in entire human body. In fact the average length of a male femur bone is 48 cm (18.9 in) and 2.34 cm (0.92 in) in diameter. The femur can also support up to 30 times the weight of an adult which leads us to question the stresses and strains that the femur endures. Moreover, the femur supports all of the body's weight during many activities like running, jumping, walking, and standing. Therefore, there are extreme forces acting on the femur which is why it is classified structurally as a major component of the appendicular skeleton [1].

In order to analyze the bone, a 3D model must first be created. Since there are many intricate parts of the bone, we thought that it would be best to download an already created model of the femur. So, a femur bone CAD model was downloaded from GrabCAD and used during the evaluation. The general plan of approach to meet the objective is to place a fixed constraint and variable boundary condition on the top and bottom of the bone. By using a finite element model, we will be able to find reactions of the femur when different loads are applied. Of course those loads depend on a person's body mass index (weight, height) so we decided to just analyze the average size person's femur. Also, it should be considered that the loads and finite element model results will change if a person is running, jumping, or lifting. In order to do all of this, the SolidWorks model from GrabCAD will be imported into ABAQUS where the bone will become meshed, followed by a finite element analysis. For reference, a human femur bone is shown in Figure 1.

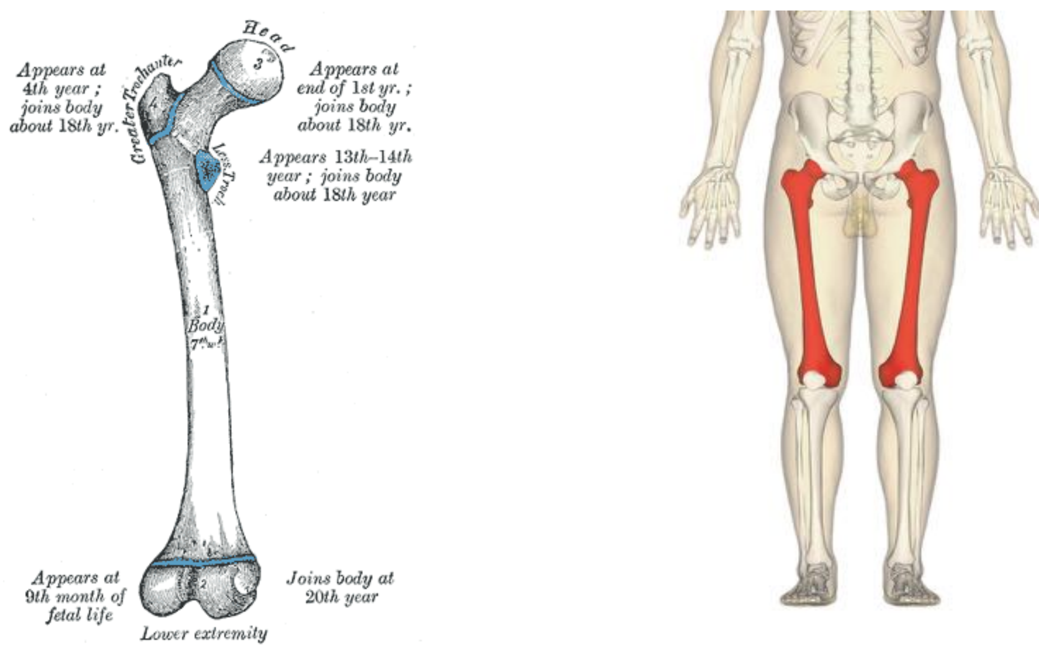

Figure 1: Femur bone [4]

Section 2: Development and Description of the CAD Geometry

CAD Geometry

We also considered different CAD designs. One of the CAD designs was scanned CAD data given in Figure 2 [3]. By doing this, the team was able to compensate for unwanted results and benefit by seeing the effects of the different and more accurate CAD model. As stated above and in order to save time and to simplify the process, our team decided to use a pre-existing model of the femur bone from a website known as GrabCAD, which is an online source designed to help engineering teams manage, view, and share CAD files. Since the femur bone clearly consists of very complex geometry, we decided that this was the best route to take and one that would yield the best results. As seen in Figure 2 below, we assigned some general dimensions in millimeters and eliminated unnecessary dimensions that caused clutter.

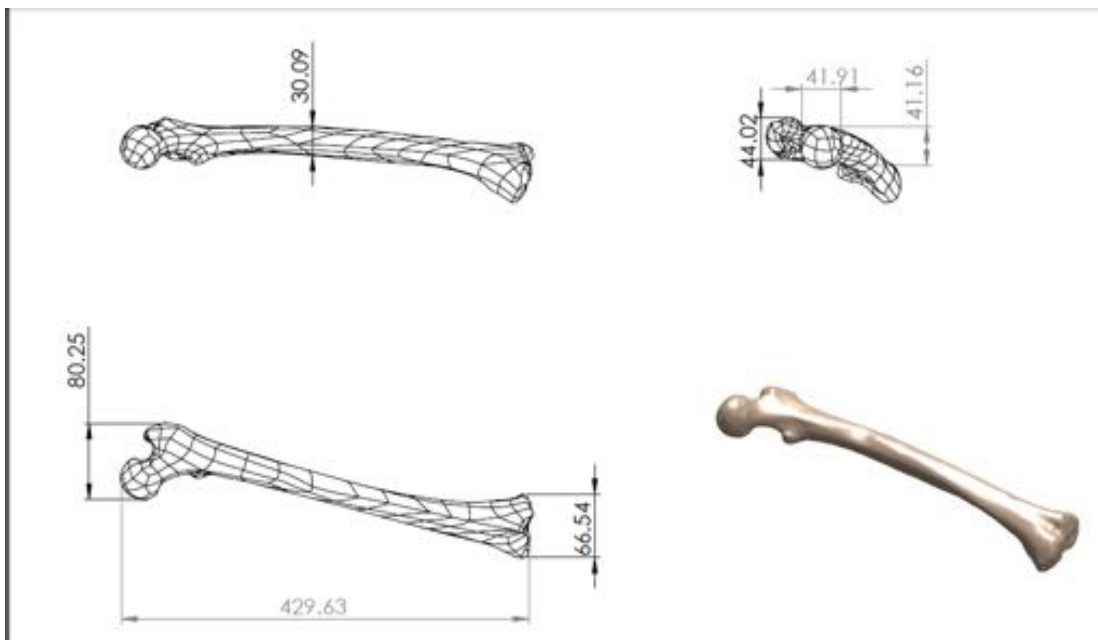

Figure 2: Scanned CAD model (mm)

Material Properties

There are many material properties that are found within the actual human femur bone. Since the true femur bone consists of multiple different areas that include various densities and elasticity, we needed a reliable or feasible means of simplifying our model, for it would have been very challenging if we were to consider these individually, as seen below in Figure 3 [9]. In an attempt to generalize our model, research was conducted and it was found that all bone typically and commonly consists of two primary materials or, in this case, “biomaterials.” These two types in medical terms are referred to as cancellous bone and cortical bone. Cancellous bone, or sometimes referred to as “trabecular bone,” is the more spongy or flexible part of bone, and is at the core of all bone structures. With having small gaps or air pockets as part of the material make up, cancellous bone is typically less dense. In contrast, cortical bone is the stronger part of bone, and is the located on the outer surface surrounding the cancellous bone. With having a higher density, this particular type of bone is the stronger of the two, and naturally gives bone its strength. Considering these two types of bone, we decided to generalize the femur by assuming that all cortical bone was of the same density and same elasticity everywhere. This was also true for our assumption of cancellous bone. So if we think of the femur bone as being a homogenous, isotropic, and linearly elastic material, we can take the Young's modules to be 17 GPa for cortical bone [5, 6, 7, 8] and 1500 MPa for cancellous or trabecular bone [5, 6]. We also assigned a Poisson's ratio, and it was defined as 0.33 for both [5, 6, 7, 8].

Parameter	Cortical bone	Trabecular bone
Hounsfield Unit (HU)	2200	800
Density(g/cm^3)	2.0208	1.3712
Modulus of Elasticity(MPa)	E_1= 6982.9	E_1= 2029.4
	E_2= 6982.9	E_2=2029.4
	E_3= 18155	E_3= 3195.3
Poisson's Ratio	υ_{12} =0.4	υ_{12} =0.4
	υ_{23} = 0.25	υ_{23} = 0.25
	υ_{31} = 0.25	υ_{31} = 0.25
Shear Modulus (GPa)[16]	G_{12}= 4.69	G_{12}= 4.69
	G_{23}= 5.61	G_{23}= 5.61
	G_{31}= 7.68	G_{31}= 7.68

Figure 3: Material Properties Table [9]

Loads

When determining which loads should be applied on the femur, as well as where those loads should be applied, we found a piece of literature written by J. P. Paul that highlighted the loads experienced by the femur bone when a person is walking and standing (shown in Figure 4 below) [10]. As seen in Figure 4, it is apparent that this, too, would complicate our analysis to a much greater extent than anticipated.

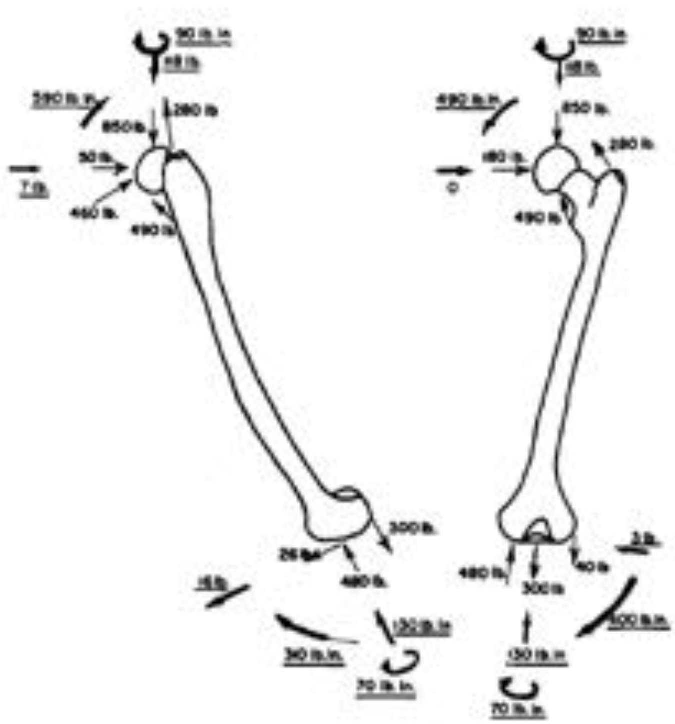

Figure 4: Loads on Femur [10]

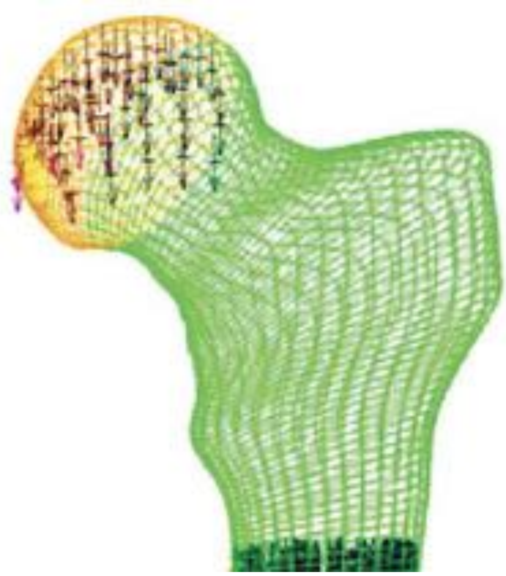

Figure 5: Surface Loads on Femur[11]

Instead of using this kind of loading presented by J. P. Paul on the femur, we found that there is a way to concentrate these loads and, as a result, simplify our model and analysis greatly. That particular loading is shown here in Figure 5 where it shows a concentrated vertical load acting downward on the ball or hip joint of the femur. In that piece of literature, they applied a 2500 N vertical force on the femur [11].

Investigating this further, some studies use different loading systems. For instance, N. Garijo and at al. used three forces on the femur head [12]. Location of the loads is given in Figure 6 and the magnitudes of these forces given in Figure 7.

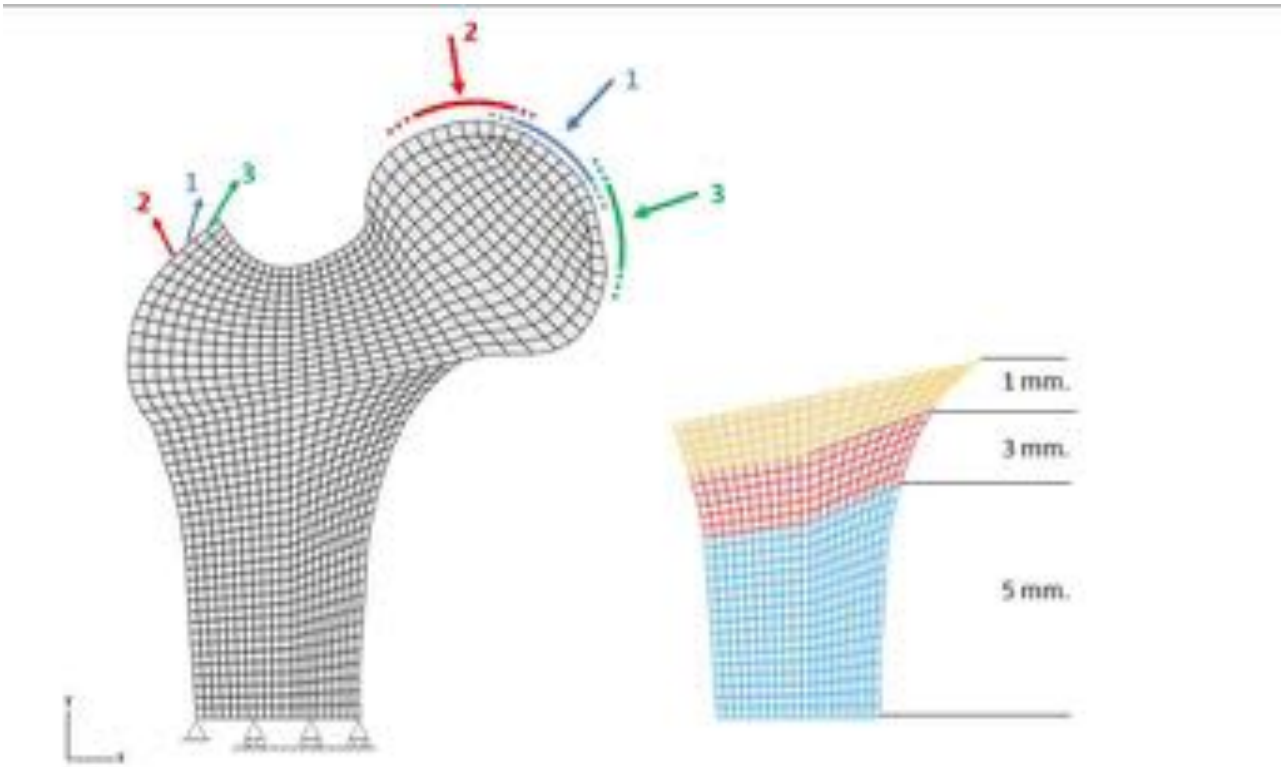

Figure 6: Three Loads locations on the head [12]

Loading conditions considered.

Mean value and orientation of the forces for the three load cases considered. Angles refer to the vertical direction

Case	Cycles/day	Load value at the head		Reaction force at the abductor muscle	
		Magnitude (N)	Angle (°)	Magnitude (N)	Angle (°)
1	6000	2317	24	703	28
2	2000	1158	−5	351	−8
3	2000	1548	56	468	35

Range of variation of the loading parameters

Parameter	Variability	Variations for each parameter
Force Magnitude	±15%	20
Angle	±5%	10
Position	Indicated in Fig. 2	5

Figure 7: The magnitudes of the three forces [12]

Section 3: Development of Finite Element Meshes

In order to increase our skills in ABAQUS and meshing, each team member created their own mesh of the femur bone. This allowed us to analyze three different meshes and see which one would work the best. Each mesh was created by using the same procedure. First the femur bone file was imported into ABAQUS as a single part. Next the material properties were defined as in Section 2 and a section was created for the bone. Then the properties were assigned to that section and an assembly was created. A static, general analysis step was used and finally a mesh was created using different global seed sizes for each team member.

Mesh 1

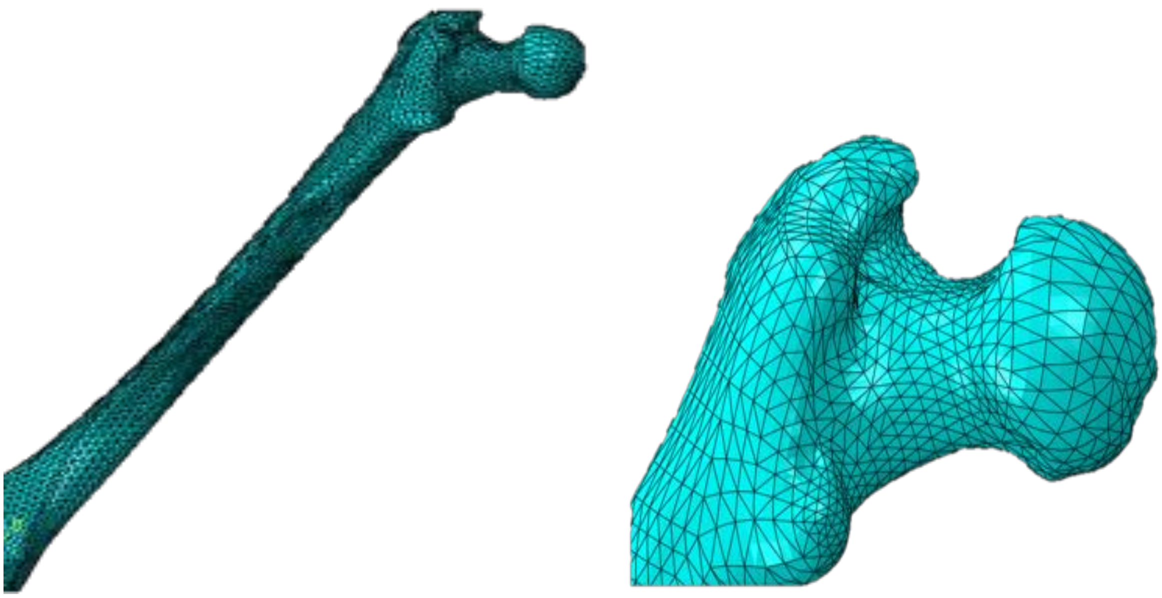

Figure 8: Global seed size of 0.18. Meshed femur #1

Mesh 2

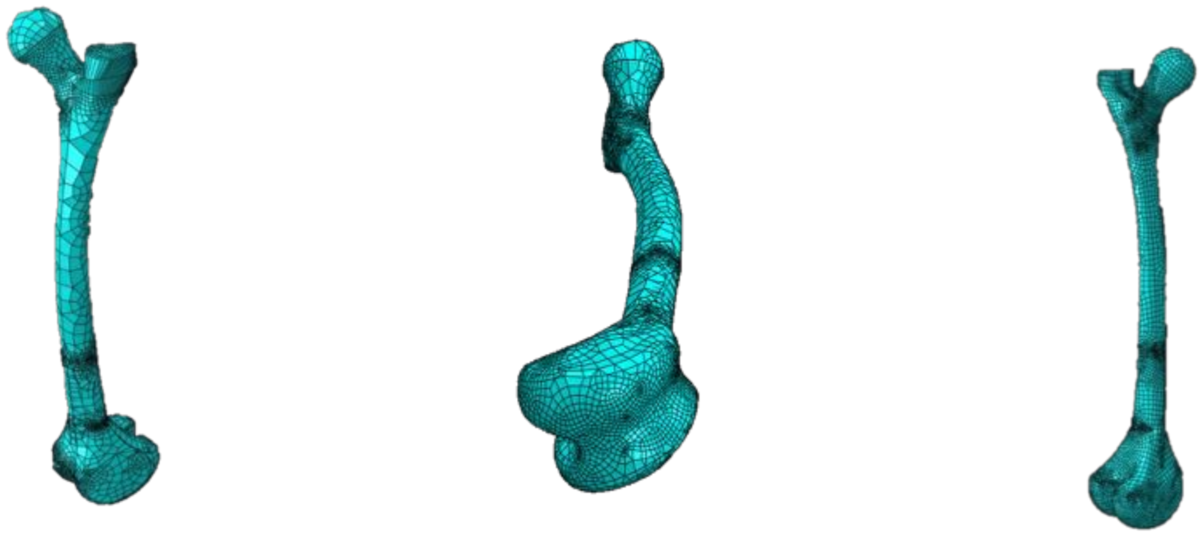

Figure 9: Global seed size of 20. Meshed Femur #2

Mesh 3

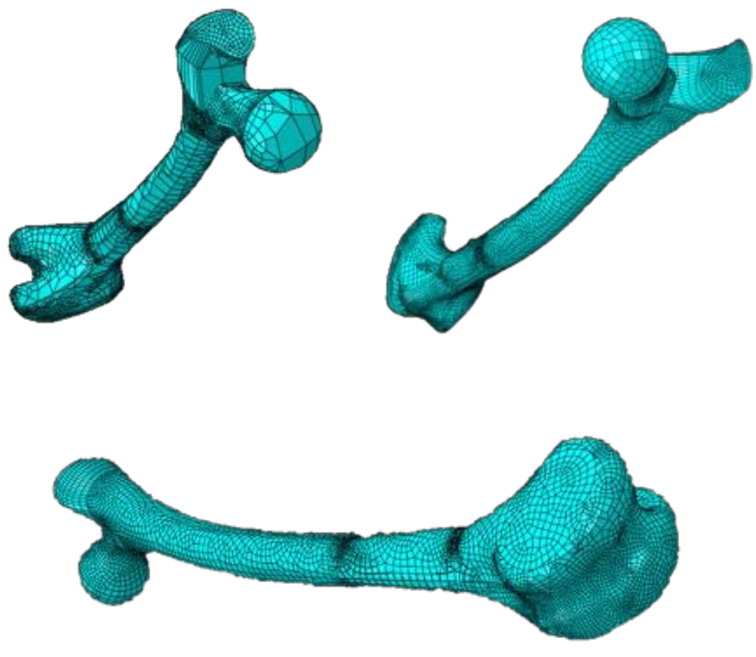

Figure 10: Global seed size of 5. Meshed Femur #3

Section 4: Development and Description of the Model Assembly and Boundary Conditions

Since our objective was to only analyze the stresses and strains that occur on the femur, there was only one part that was evaluated in ABAQUS, therefore only a basic assembly was created. After that, a vertical surface force was applied on the head of the femur bone at 2500 N and boundary conditions were applied at the bottom face as shown below in Figure 11. This applied force value was taken from the previously cited paper [11].

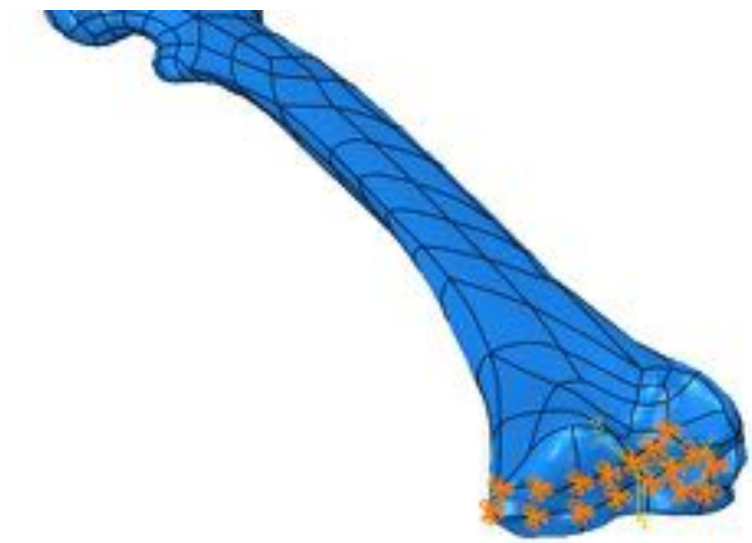

Figure 11: Fully fixed boundary condition at the bottom of the femur bone

After the loads, restraints, and mesh were applied to the femur, a "Job" was created and submitted. We monitored the results and ran a simulation on the bone and the outcomes for stress and displacement are shown below in Figures 12-17.

Figure 12: Results of basic simulation – initial position

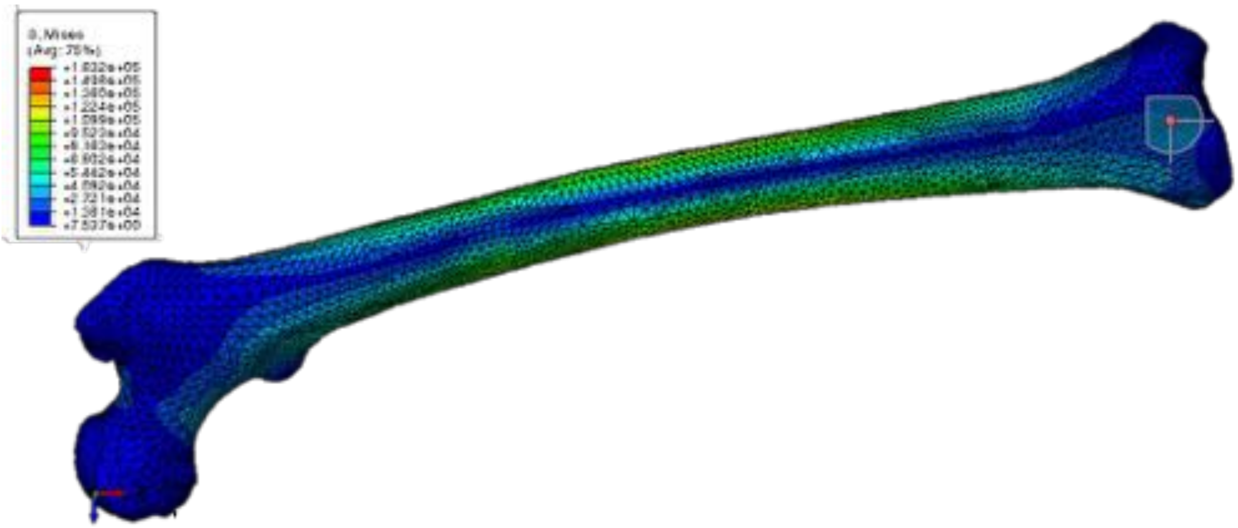

Figure 13: Results of basic simulation –last position

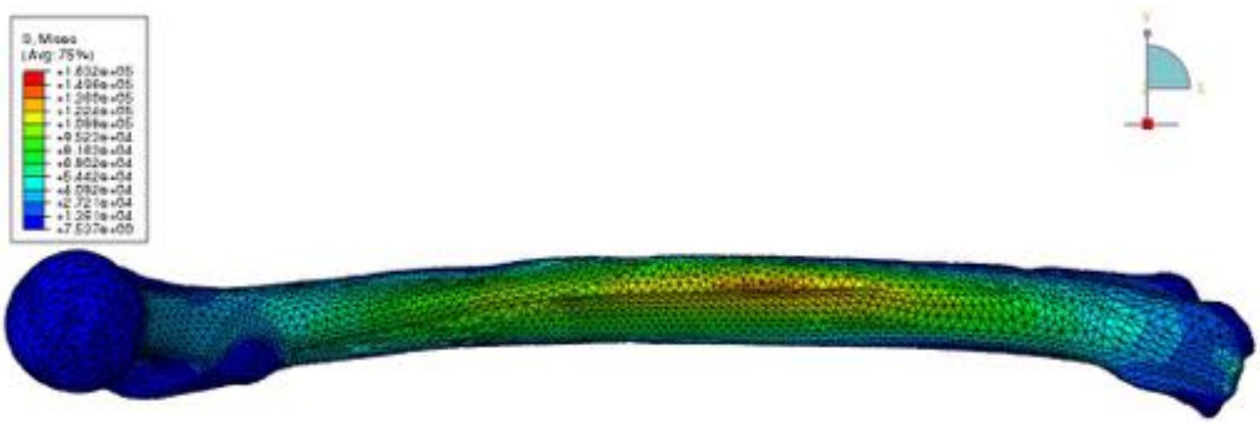

Figure 14: Results of basic simulation –side

As you can see from the figures above, with the bottom fixed, the head of the femur bone is displaced which causes a deformation in the middle of the femur. The reddish color means that there is a higher value of stress in that area where as the blue means little stress is occurring at those points. By looking at the close-up of the femur in Figure 15, the stresses become very clear and it's obvious that the middle of the bone is where the stresses occur.

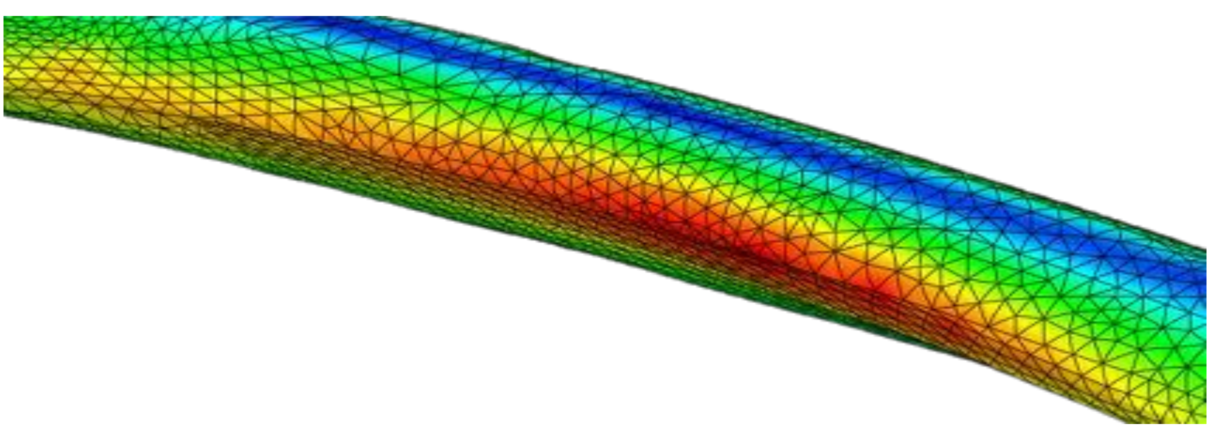

Figure 15: Close-up (Stress point)

The next measurement that was simulated and analyzed was displacement. As stated before, the only displacement occurred in the head of the femur because the bottom end was fixed with boundary conditions. The figure below confirms this by showing the red and orange areas as areas with the most displacement and the blue areas with little to no displacement.

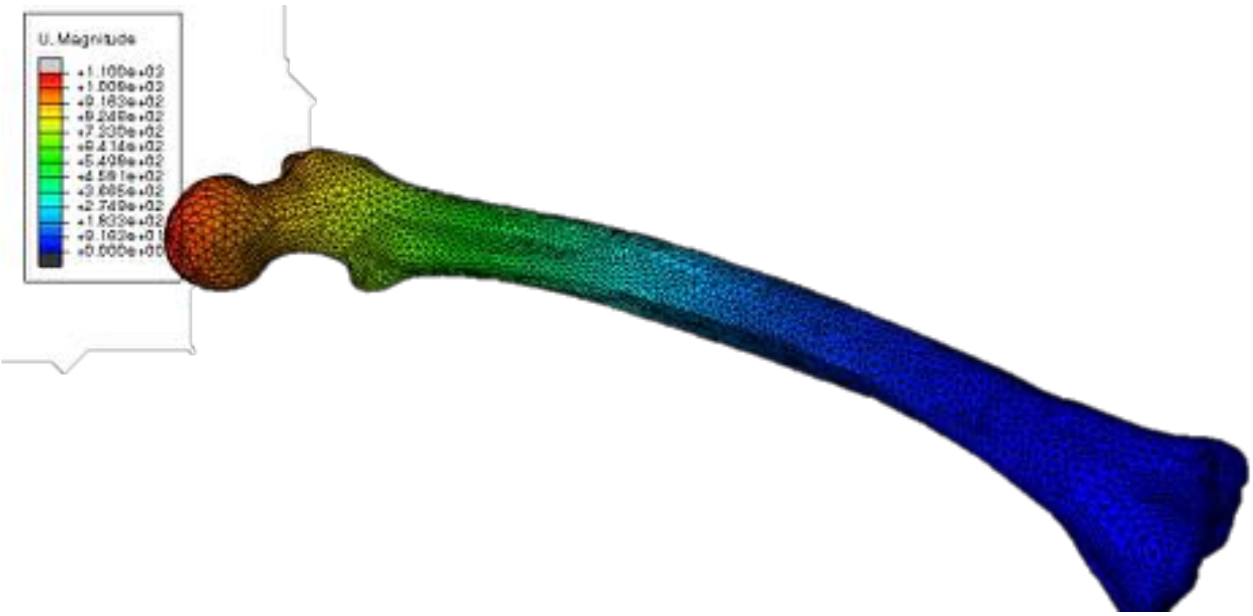

Figure 16: Displacement simulation

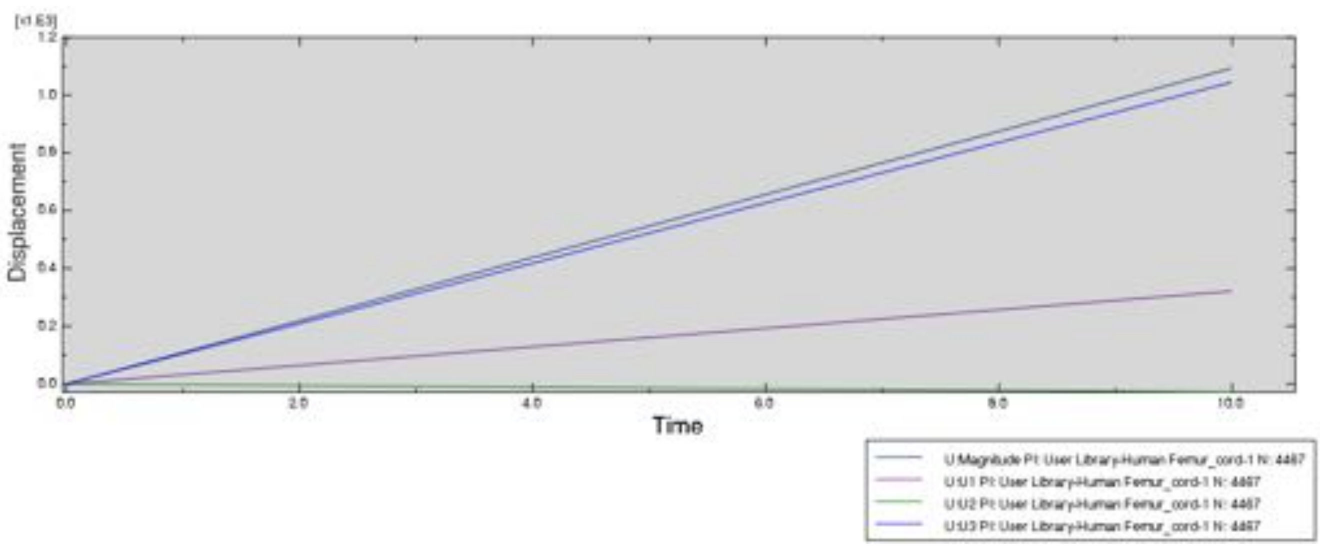

Figure 17: Displacement vs. Time

Section 5: Development and Description of Model Interactions

In order to understand the femur reactions and compare with previous results, the femur bone was cut in half. The boundary conditions remained the same as the cut-off end was fixed, however there was a slight change in the interactions applied at the head of the bone. Instead of using a vertical surface force, pressure forces were applied to the head of the femur as shown below in Figure 18. The magnitude of the force remained the same as the previous analysis (2500N). Pressure forces can be seen more clearly in Figure 19.

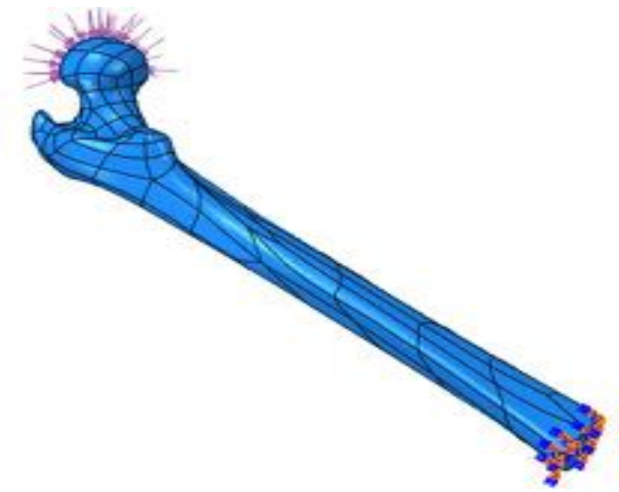

Figure 18: Fixed cut femur bone

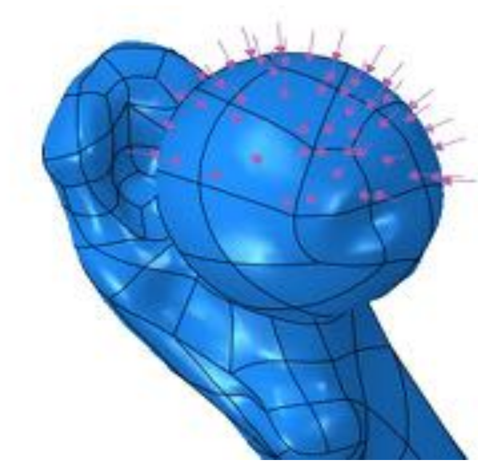

Figure 19: Pressure forces 2500N

After the pressure loads and the boundary conditions were created and applied, a finer mesh was created and used for the femur bone (Figure 20).

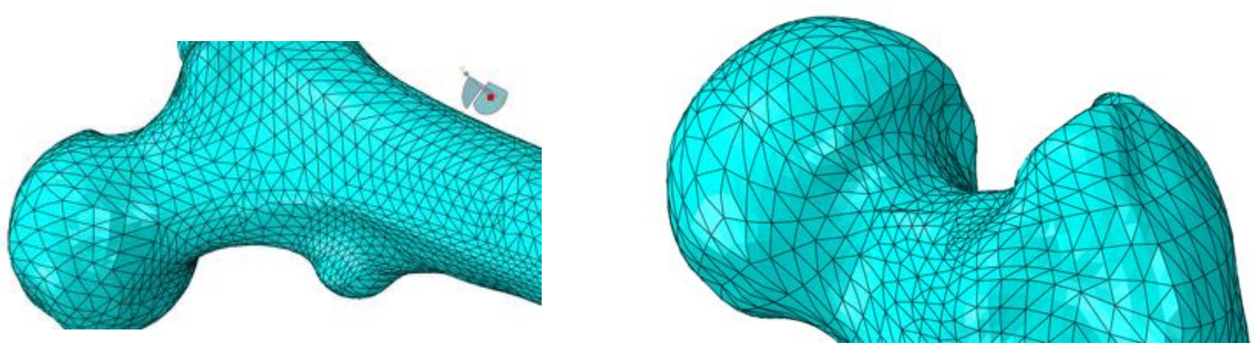

Figure 20: Fine mesh

Again, stress and displacement were the two characteristics being measured and the results are shown below in Figures 21-23. As you can see from Figure 21, the majority of the stress is occurring in the same place as did the previous experiment. Moving onto the displacement analysis, it is also clear to see that the most displacement once again occurs at the head of the femur bone.

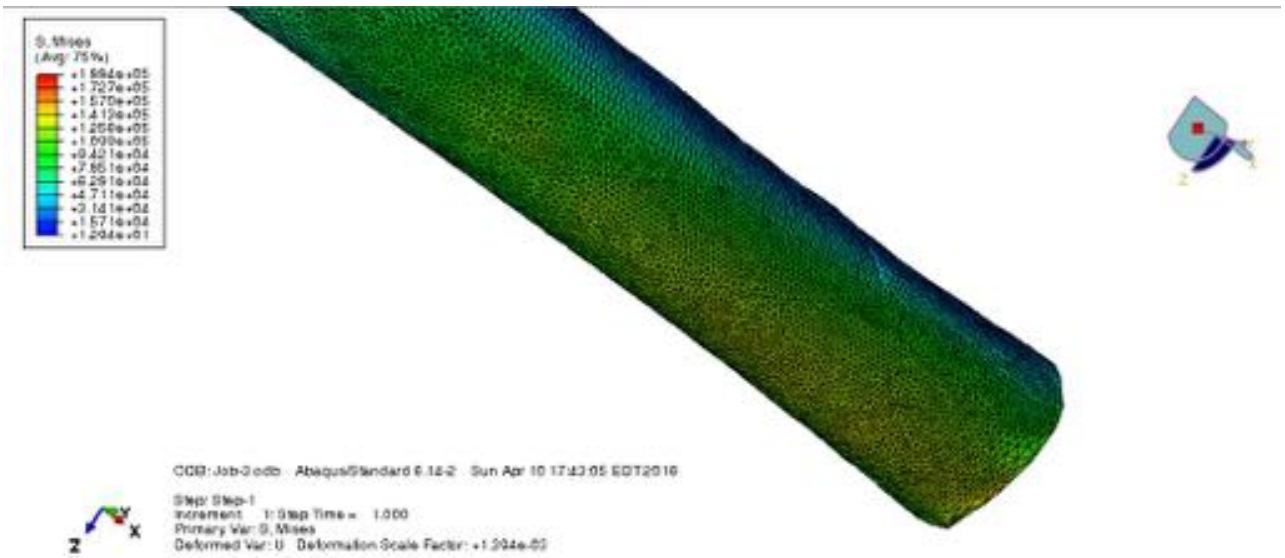

Figure 21: Stress results

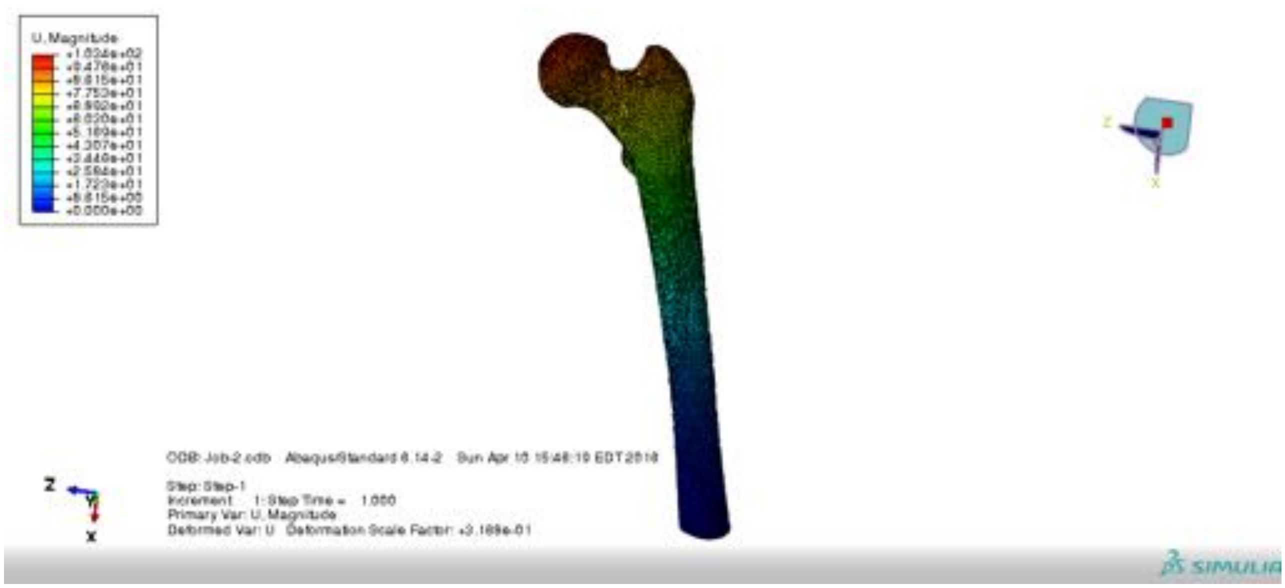

Figure 22: Displacement Results

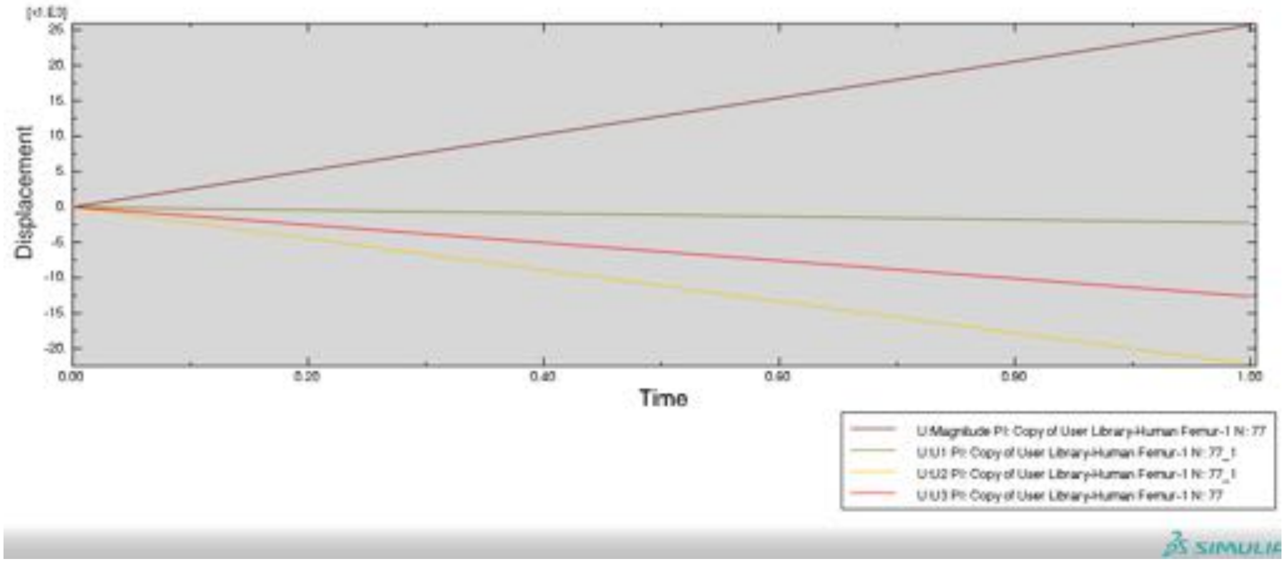

Figure 23: Displacement vs. Time

The final analysis performed by the team involved using the complete femur bone like in the first experiment. The difference this time is that a block was created in ABAQUS. We added the block to the top of the bone and applied a displacement to the block in order to induce some kind of reaction on the bone. The results are shown in Figure 24 below. By looking at the Figure 24, the trend continues in that the maximum stress occurs at the middle of the bone.

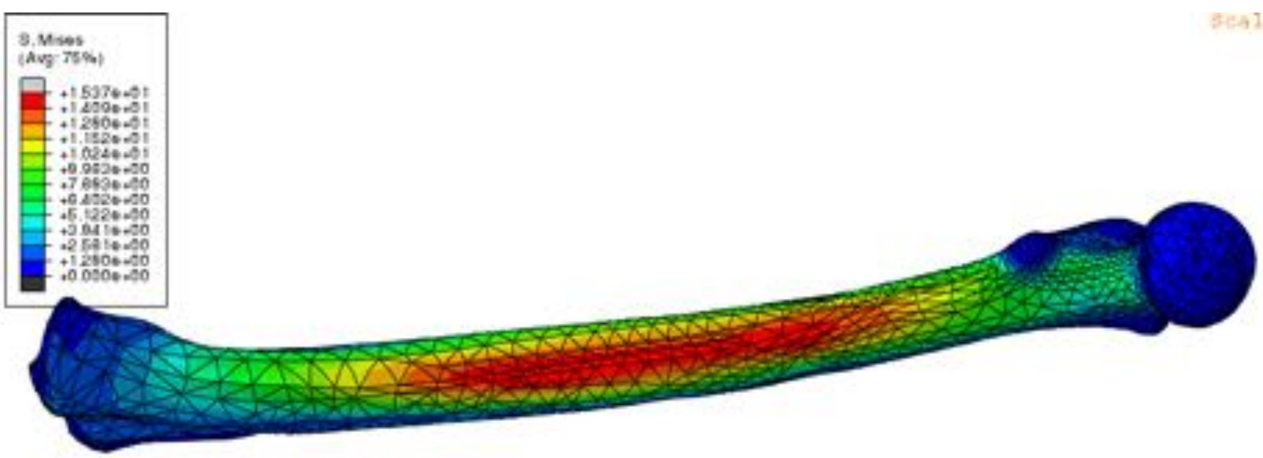

Figure 24: Stress distributions

.

Section 6: Analysis of Finite Element Model

Our simulation was run mostly in static, general and in each of the three analyses we only used one step. Since we simplified the loads being applied on the femur our ABAQUS analysis was not very complicated and didn't require much time to run. In fact, the run time to complete the analysis only took about 5 minutes or less; so an extra processor was not needed to run our analysis.

Section 7: Summary of Major Findings

Overall we found that replicating the human femur bone in computer analysis is very hard to perform. So, simplification of the loads applied needed to be included or else we would have taken a long time to figure out how to correctly apply all the loads. Most of the time was spent researching the femur bone and the material properties as well as planning for how an analysis was going to be performed. Our major results and findings are as follows:

- The results we found were very similar when compared to the references and online resources we used.
- Every experiment shows the same reactions where the maximum stresses occur at the middle of the femur and the displacement occurs at the head.
- The material structure and properties of a human bone are tough to replicate.
- The bone has both compressive and tensile stresses occurring at the same time.
- In our results, the bone head displaces in the x, y, and z directions.
- When we cut the bone in half the displacement was less compared to the analysis on the full bone as expected.

Section 8: Works Cited

[1] "Inner Body." [Online]. Available: https://www.innerbody.com/image_skelfov/skel25_new.html#full-description.

[2] "GrabCAD." [Online]. Available: https://grabcad.com/library/femur-bone.

[3] "3D femur." [Online]. Available: www.3dcontentcentral.com/Search.aspx?arg=femur.

[4] "Fermur Wikipedia." [Online]. Available: https://en.wikipedia.org/wiki/Femur.

[5] K. Polgár, H. S. Gill, M. Viceconti, D. W. Murray, and J. J. O'Connor, "Strain distribution within the human femur due to physiological and simplified loading: finite element analysis using the muscle standardized femur model.," *Proc. Inst. Mech. Eng. H.*, vol. 217, no. 3, pp. 173–189, 2003.

[6] G. N. Duda, M. Heller, J. Albinger, O. Schulz, E. Schneider, and L. Claes, "Influence of muscle forces on femoral strain distribution.," *J. Biomech.*, vol. 31, no. 9, pp. 841–6, 1998.

[7] M. A. R. and Taylor, M. E., Tanner, K. E., Freeman and A. L. Yettram, "Stress and strain distribution within the intact femur: compression or bending?," *Med. Engng Phys.*, vol. 18, pp. 122–131, 1996.

[8] R. P. Lengsfeld, M., Kaminsky, J., Merz, B. and Franke, "Sensitivity of femoral strain pattern analyses to resultant and muscle forces at the hip joint," *Med. Engng Phys.*, vol. 18, pp. 70–78, 1996.

[9] a E. Yousif and M. Y. Aziz, "Biomechanical Analysis of the human femur bone during normal walking and standing up," *IOSR J. Eng.*, vol. 2, no. 8, pp. 9–12, 2012.

[10] J. P. Paul, "Load actions on the human femur in walking and some resultant stresses," *Exp. Mech.*, vol. 11, no. 3, pp. 121–125.

[11] L. Voo and M. Armand, "Stress fracture risk analysis of the human femur based on computational biomechanics," *Johns Hopkins APL*, vol. 25, no. 3, pp. 223–230, 2004.

[12] N. Garijo, J. Martinez, J. M. Garcia-Aznar, and M. A. Perez, "Computational evaluation of different numerical tools for the prediction of proximal femur loads from bone morphology," *Comput. Methods Appl. Mech. Eng.*, vol. 268, pp. 437–450, 2014.