ME 461:
Finite Element
Analysis

Fall | 2015

The Development and Analysis of an FEA Simulation on an Artificial Knee

Daniel Offley, Joseph Martin, Harrison Wostein

PennState
College of Engineering

Department of Mechanical and Nuclear Engineering, University Park, PA

Table of Contents

Table of Contents 2
Executive Summary 3
Acknowledgements 4
List of Figures 5
Section 1: Background and Project Plan 6
Section 2: Development and Description of the CAD Geometry 7
Section 3: Development of Finite Element Meshes 9
Femoral Implant 9
Meniscus Plate 10
Tibial Implant 11
Mesh Quality 11
Section 4: Development and Description of the Model Assembly and Boundary Conditions 12
Total Knee Implant 12
Meniscus Model 14
Section 5: Development and Description of Model Interactions 14
Total Knee Implant 14
Meniscus Model 16
Section 6: Analysis of Finite Element Model 16
Simplified Simulation 16
Meniscus Model 18
Section 7: Summary of Major Findings 18
Section 8: Works Cited 21
Appendix 1: Figures 21
Appendix 2: Fatigue Life Calculations 24

Executive Summary

Artificial knees are an increasingly utilized technology. They allow for patients to regain mobility due to injury, pain, or joint damage. Unfortunately, due to material limitations and properties, these artificial knees have a finite lifespan (10-20 years). Therefore, there is continual and essential research into improving the materials used in the joints.

Our team's project focuses on creating a model of the artificial knee joint, then simulating cyclical loading on the joint to failure. By simulating "wear and tear" on the joint from normal use, we will be able to determine failure points and simulated lifetimes of the artificial knee. Additionally, by varying material properties, new and different materials can be used in the model to test for use in the implants. The goal is that these new materials would increase the fatigue life of the joint.

Our team began by importing CAD geometry of the knee implant. Next, each part of the assembly was separated and meshed. This was successful; however, significant hurdles were encountered in programing the interactions of the parts combined with cyclical loading. In order to solve this problem, the geometry was simplified to a 10x10 cube, applying the correct material properties, initial conditions, and loading conditions to it. After successful test runs of the representative element, the same conditions were applied to the plastic polymer piece, or meniscus plate, of the knee implant. Therefore, the model simulated the cyclic loading condition of an artificial knee patient walking.

The final product of this study was a FEA model of the meniscus plate (part subject to most wear) of the knee geometry. The model included a cyclic loading condition and estimated the lifetime of the part to be approximately 6.604×10^{62} years, essential an infinite life. Future improvements to the model could include the addition of frictional effects, more accurate loading conditions, analysis of different failure modes, and varying material properties.

Acknowledgements

Acknowledgements for this paper and study include:

Dr. Rueben Kraft and Dooman Akbarian for teaching us a lot about FEA and helping our team troubleshoot the project.

GrabCAD for supplying us with the artificial knee SolidWorks files.

List of Figures

Figure 1: Artificial knee replacement and CAD representation…………………………………… 6
Figure 2: Fatigue Life Equations……………………………………………………………... 7
Figure 3: Geometry of CAD representation………………………………………………….. 8
Figure 4: Femoral Implant CAD Geometry………………………………………………….. 9
Figure 5: Mesh of the femoral component…………………………………………………… 10
Figure 6: Mesh of the meniscus plate………………………………………………………… 10
Figure 7: Imported CAD geometry of the tibial implant……………………………………... 11
Figure 8: Mesh quality………………………………………………………………………... 12
Figure 9: Parts of the CAD geometry with mesh…………………………………………….. 13
Figure 10: Rigid body constraint……………………………………………………………. 13
Figure 11: Meniscus plate boundary conditions……………………………………………….. 14
Figure 12: Total knee implant interactions………………………………………………….. 15
Figure 13: Cyclic load applied to the femoral implant……………………………………..... 15
Figure 14: Amplitude and boundary conditions of simplified model……………………………. 17
Figure 15: Stress and deformation of the cube at time step……………………………………… 17
Figure 16: Cyclic stress from element centroid of meniscus………………………………….. 18
Figure 17: Contour plot of meniscus at 5.1 seconds…………………………………………... 18
Figure 18: Stress along the Z axis path through the meniscus plate……………………………... 19
Figure 19: Cutaway view of the part highlighting the element that XY data was generated……… 23
Figure 20: Contour stress plot of minscus cutaway……………………………………………. 23
Figure 21 Cutaway view of meniscus highlighting the stress XY data path…………………….. 24
Figure 22: Modal frequencies for the first 3 eigenvalues of meniscus part……………………… 24
Figure 23: Contour plot at each modal frequency of the meniscus part………………………….. 25

Section 1: Background and Project Plan

Knee arthroplasty, knee replacement surgery, is the process of cutting away bone and cartilage from the femur, tibia, fibula, and patella, and replacing it with an artificial joint. Osteoarthritis is the leading reason for knee replacements. Unfortunately, currently marketed artificial knees last between 10-20 years depending on the patient[2].

The artificial knees are made of titanium or cobalt-chromium based alloys (metal parts), and high molecular weight polyethylene (plastic parts). Implant components are fixed to the bone via a bone cement, polymethylmethacrylate.

There is a specific need for long lasting artificial knee implants (30-40 years) due to early onset osteoarthritis in kids and young adults, mostly due to sports injuries. A longer lasting knee replacement would mean fewer surgeries over a patient’s lifetime.

Initially, the goal of this project was to determine which materials produce the least wear on the artificial joint due to cyclic loading (normal usage by a patient). By understanding how the materials respond to cyclical loading, the best material could be chosen to extend the life of artificial knees. Specifically, the group focused on the meniscus plate of the knee implant. Geometry for the artificial knee was provided from a free online CAD model database.

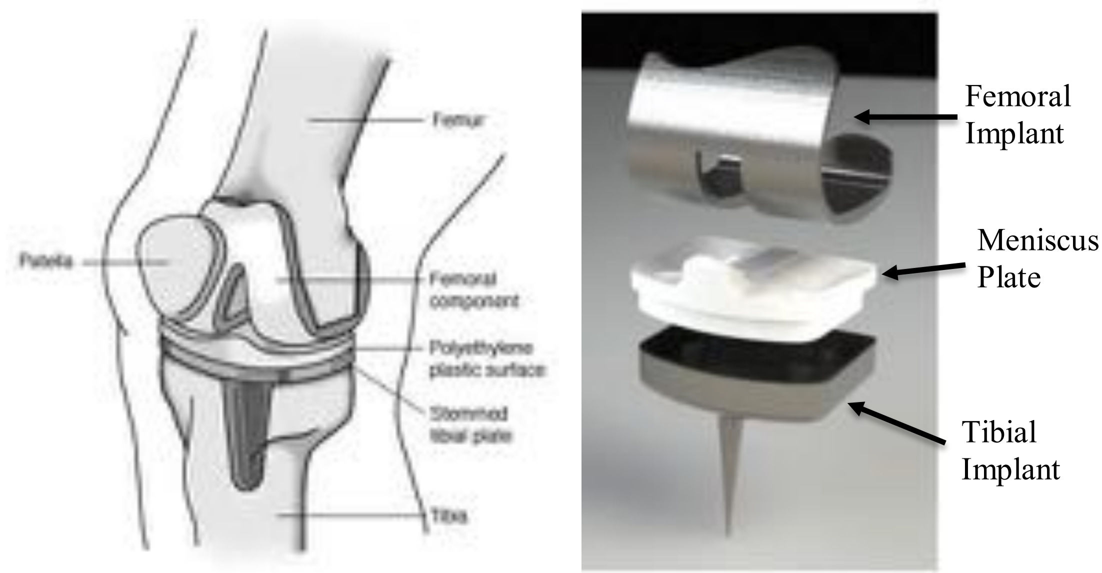

Figure 1: Figure on left demonstrates how an artificial knee replaces the human knee joint. The top and bottom components of the knee are cemented to the bone. The CAD geometry on the right will be used in the FE analysis.

Due to complications throughout the project, the team’s goal evolved to consider whether our FEA model could accurately simulate the lifetime of an artificial knee, specifically the meniscus plate made of plastic. Our goal was to model the knee accurately to show that it fails at around

10 years of time. The plan was to accurately model the meniscus plate and determine the minimum and maximum stresses that occur on the plate during cyclic loading. From this output of the model, the amplitude stress σ_a could be determined and input into equations for fatigue life as shown in Figure 2. Thus, from fatigue life calculations the total amount of time before failure could be determined and compared to experimental data showing a life of 10-20 years on average[2].

$$S_n = aN^b$$

$$S_n = S_m \quad at \ N = 10^3$$
$$S_n = S_e \quad at \ N = 10^6$$

$$axial\_loading : .... S_m = 0.75 S_{ut}$$

$$S_e' = 0.5 S_{ut}$$

$$S_a = \frac{S_{maximum} - S_{minimum}}{2}$$

Figure 2: The governing equations for calculating fatigue life N. S_a is the amplitude stress calculated from the maximum and minimum stresses on the meniscus plate in the model as seen in the fifth equation. S_a is equal to S_n at a certain fatigue life and must be determined by finding the coefficients a and b in the first equation. Coefficients a and b can be found using the ultimate tensile stress of the meniscus plate material to find S_m and S_e using the third and fourth equations. Once S_m and S_e are found, the first equation can be solved using the second equation and developing a system of equations.

Section 2: Development and Description of the CAD Geometry

CAD geometry for an artificial knee implant is available from GrabCAD, an engineering community that allows engineers to share files[1].

Description of CAD Model

A knee implant consists of three components to replace the patient's knee. The geometry used for this project is shown below in Figure 3. Part 1 is metal, attaches to the end of the femur, and is thus called the femoral implant. Part 2 is plastic, acting as the meniscus, providing a cushion and surface for the knee to rotate upon. This part is called the meniscus plate. Part 3 is also metal and replaces the top surface of the tibia, thus called the tibial implant. This particular knee implant does not replace the patella, however surgeons commonly use a portion of the patient's scrap bone and craft a new patella for cosmetic purposes.

Figure 3: Geometry of knee implant that will be used in FE analysis. Exploded view on the left, assembled view on the right.

Loading Conditions:

The human knee joint is subjected to different forces for all normal activities. For the purpose of this project, we will be simulating the force subjected to a knee joint during walking. On average, a male weighs 195 pounds and a female weighs 140 pounds. The recommended number of steps per day that an individual should take is around 10,000. While walking, the human knee joint is subjected to a force equivalent of 2.5-2.8 times the individual's body weight. Knee implants typically last for around 10 years and then require replacement or repair. Therefore, the project will consist of cyclically loading the knee with approximately 487.5 lbs of force (2168.5 N) to determine the stress amplitude of the knee. The stress amplitude will then be used to calculate the model's fatigue life.

Material:

The materials used in a total knee replacement implant must be biocompatible, thus limiting the amount of materials available to use for the implant. Typically, a titanium or cobalt-chromium based alloy is used for the femoral and tibial implants and a high molecular weight polyethylene is used for the meniscus plate. For further analysis, the team will need to conduct research to determine alternative materials. For the goal of comparing the analytical model to the experimental data on artificial knees, our team used a titanium alloy (Ti-6Al-4V) for the femoral and tibial implants and a high molecular weight polyethylene for the meniscus plate. The material properties for these are shown in the table below.

Table 1: Material Properties

Material Properties	Ti-6Al-4V	High Molecular Weight Polyethylene
Density (kg/m^3)	4430	931-949
Ultimate Tensile Strength (MPa)	950	40
Yield Strength (MPa)	880	2.5
Poisson's Ratio	0.35	0.46
Young's Modulus (GPa)	113.8	0.90

Section 3: Development of Finite Element Meshes

Femoral Implant

First, we opened the SolidWorks assembly and saved the parts as STEP files. Issues importing the part through the Hammer cluster occurred, possibly due to improper importation of files into the interface. The team downloaded the parts files through Mozilla Firefox on the Hammer system and then attempted to import the parts with no success. However, the team was successfully able to import the STEP files into the student edition of Abaqus on a personal computer. The imported geometry of the femoral implant is shown below:

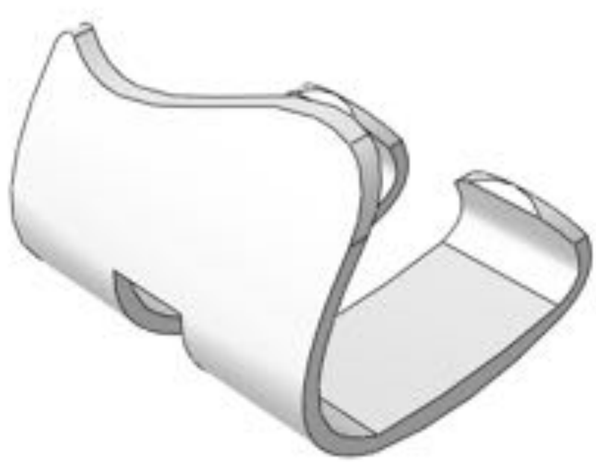

Figure 4: Femoral Implant CAD Geometry

Next, material properties of the part were defined. This part of the artificial knee attaches to the patients femur and is made of a Ti-6Al-4V alloy. The elastic material properties of Ti-6Al-4V were input into Abaqus and the entire part was defined with the material properties.

Next, a solid, homogeneous section was created and the section was assigned to the entire part. Next, an independent instance was created as the group was unable to mesh the part with a dependent instance. A mesh was created for the part using a tetrahedral mesh. The meshed part is shown below:

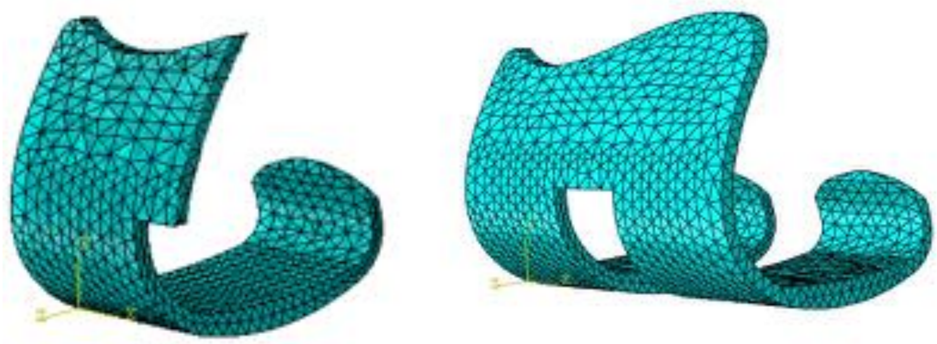

Figure 5: Mesh of the femoral component. (Right) cross section of femoral mesh.

Meniscus Plate (Wear Resistant Insert)

A tet element type was used in meshing, more specifically a C3D10. This was the only mesh type that produced no errors when meshing. The material assigned to the mesh was to vary, but the group used Ultra High Molecular Weight PolyEthylene (UHMWPE) for the model accuracy study.

To create the mesh, the group first imported the part into Abaqus as 1 part. Then a material was created for the part, UHMWPE, using researched values. The material was assigned to the part using an independent instance. An element type was then assigned, selecting many different types, until settling on a C3D10 tet. The mesh of the meniscus plate can be seen below. The meshed meniscus contained 62,228 nodes and 41,957 elements.

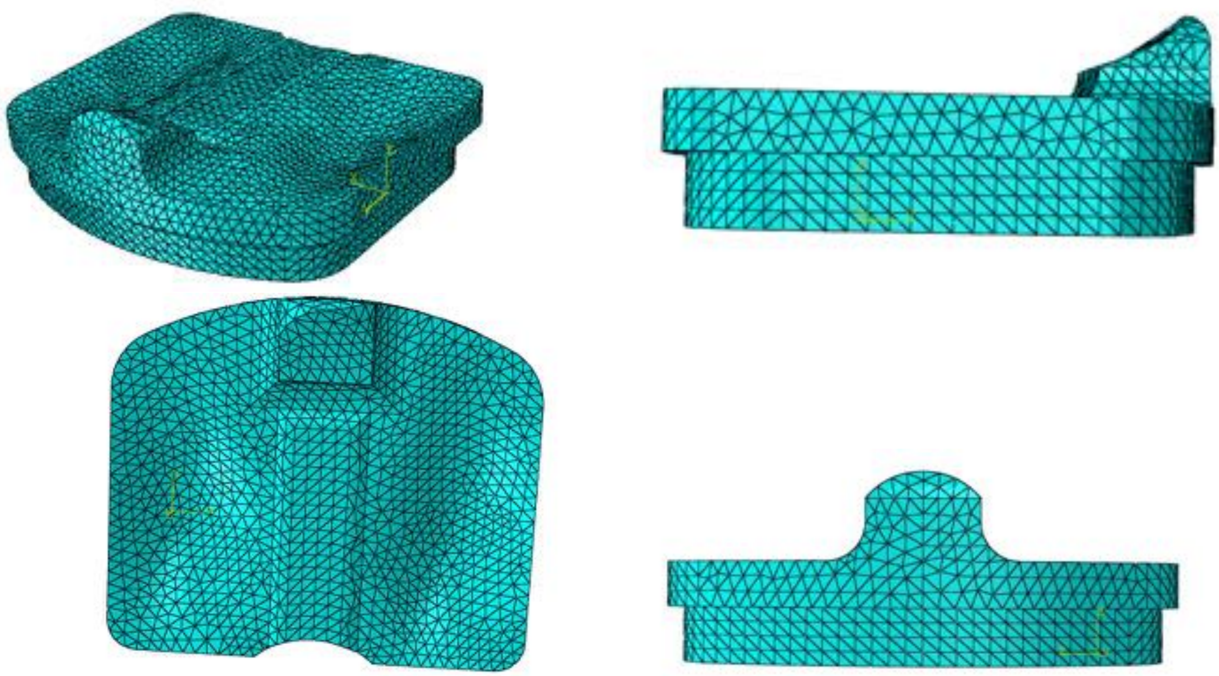

Figure 6: Mesh of the meniscus plate shown at multiple viewpoints.

Tibial Implant

Figure 7: Imported CAD geometry of the tibial implant.

First, the model of the tibial implant was imported into Abaqus through the Hammer Cluster. Similarly to the femoral implant, the material properties for Titanium alloy Ti-6Al-4V were applied to the part. However, the part contained instances that were not fully defined and therefore the area to be meshed could not be defined. As such, the team began to shift the project direction to modeling just the meniscus plate.

Mesh Quality

Each part of the Artificial Knee was combined in an assembly by importing all 3 parts into Abaqus and creating an instance with all 3 put together. The model was remeshed using tets (C3D10). The mesh was verified, specifically the shape factor, and the result can be seen below. The yellow elements shown are in danger of breaking the shape factor limit, but are not. Overall, the mesh is of a high quality.

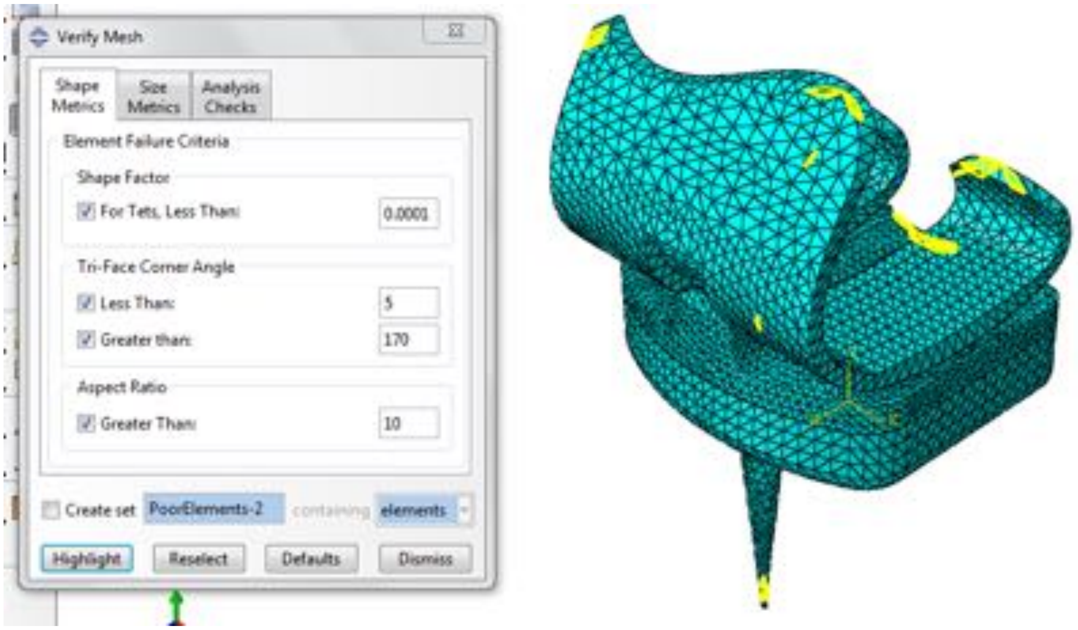

Figure 8: Mesh quality highlighting elements with a shape factor less than 0.0001, corner angle less than 5 and greater than 170 degrees, and an aspect ratio greater than 10. The yellow regions indicate lower quality elements, but not poor quality.

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

Total Knee Implant

Assembly

Each part of the artificial knee was combined by importing all 3 parts into Abaqus and creating an instance with all 3 parts. The assembly was remeshed using tets and a mesh quality diagnostic was run as shown previously. The result of the assembly is shown below.

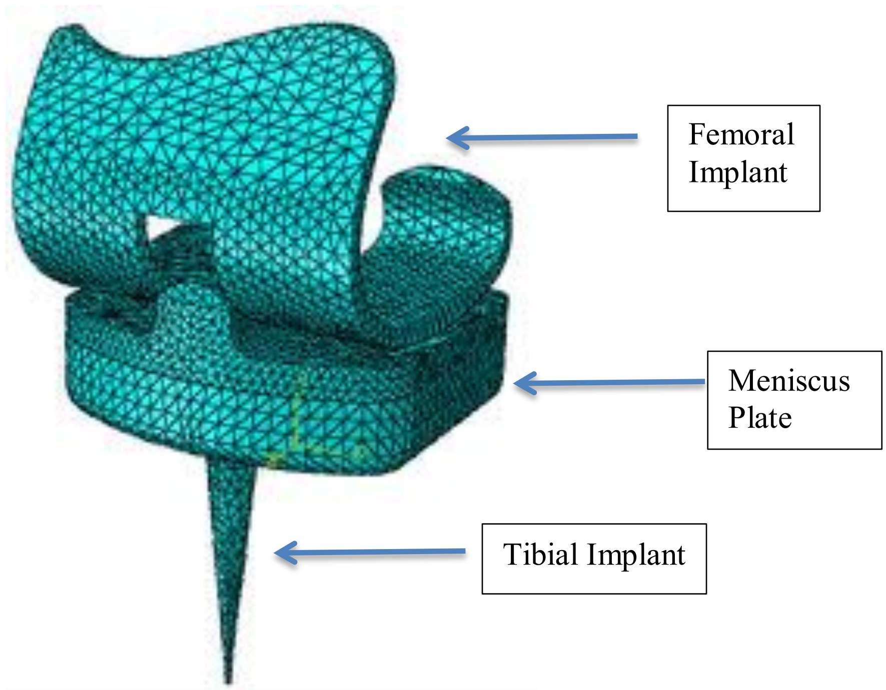

Figure 9: Parts of the CAD geometry with mesh

Boundary Conditions

The boundary conditions for the model were to set the bottom (tibial) piece in place without displacement, to allow the middle plastic piece to deform, and to allow the top (femoral) piece to move freely. The only boundary condition needed to do this was a zero displacement condition on the tibial implant coupled with the interactions of the other two pieces. A rigid body constraint was added on the tibial piece by creating a Displacement/Rotation Boundary Condition on the entire tibial part and entering in zero displacements in the x,y, and z directions for step 1. Under the initial step, the x, y, and z-axis were selected for the condition.

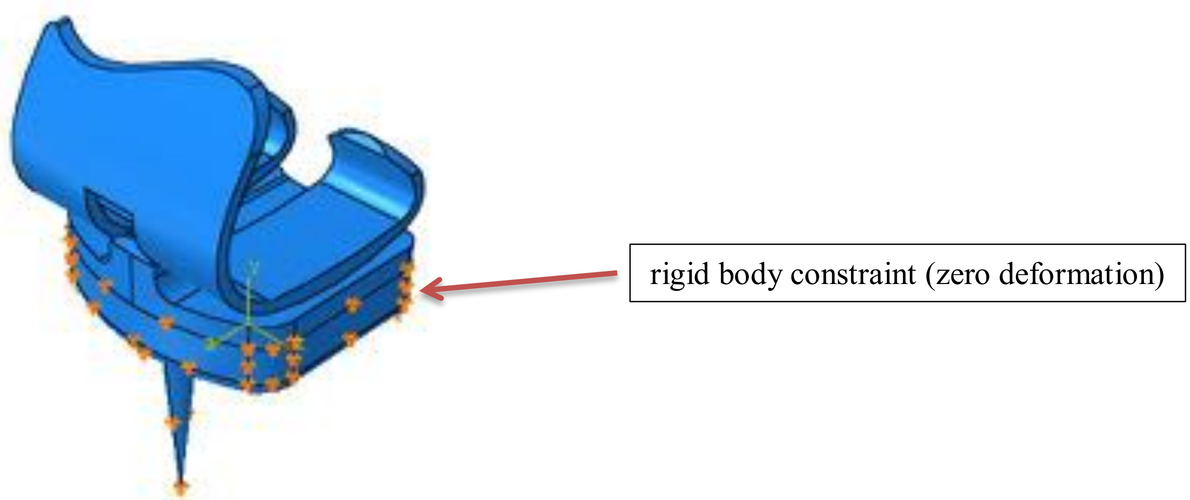

Figure 10: Rigid body constraint

Meniscus Plate Model

After consideration and issues with the tibial implant, the team decided that only the meniscus plate would be necessary to evaluate for fatigue life. The decision was made because the meniscus part will wear down and fail first before the femoral and tibial parts due to the softer material used in the meniscus section.

Assembly

Assembly was not required in this simulation as only one part was used for the analysis.

Boundary Conditions

When the knee implant is assembled, a portion of the meniscus plate is fixed to the tibial implant. Therefore, all surfaces of the meniscus plate that are in contact with the tibial part are fixed. A traction force was applied to the top of the meniscus plate to simulate the load applied by the femoral implant. The magnitude of the traction force was 2168.5, as indicated before in this report. The traction force was set to follow an amplitude, therefore applying a cyclic load to the part.

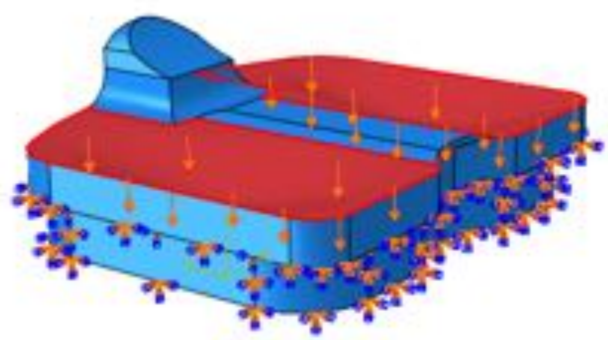

Figure 11: Meniscus plate boundary conditions. The traction force is shown in the red with the arrows indicating the direction of the force. The bottom of the part is fixed.

Section 5: Development and Description of Model Interactions

Total Knee Implant

Only two contact interactions were necessary for the model; at the junction between the tibial and meniscus plate and between the meniscus and femoral implant. A contact interaction was created between the femoral implant and the meniscus plate. The interaction was created to be a tangential/normal behavior with hard contact/friction applied to the meniscus plate's top surface and the femoral piece's bottom surface. The coefficient of friction between the surfaces was 0.2. The hard contact property had a checkbox unchecked so that the parts could not separate after contact. The second interaction was created between the meniscus plate and the tibial implant. The interaction property had tangential and normal behavior having a hard contact and a frictionless aspect. The hard contact property did not allow the parts to separate afterwards. The images below show the contact interactions between the three pieces of the model.

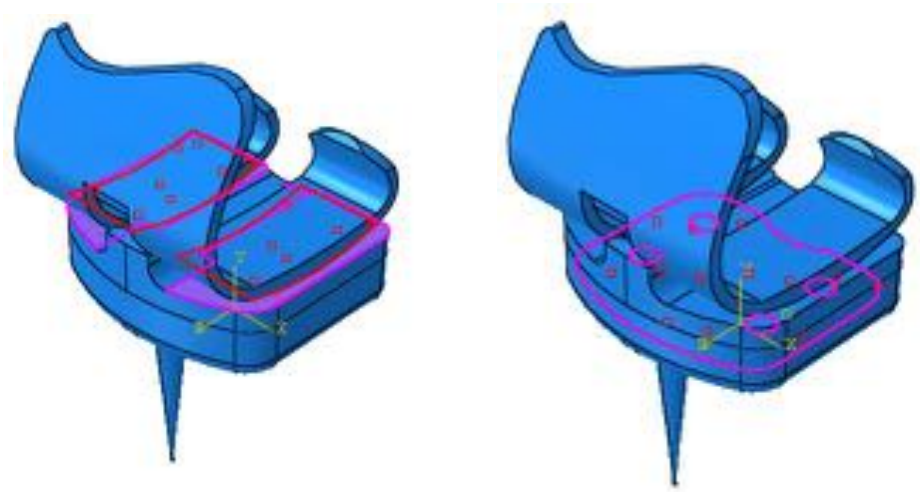

Figure 12: Total knee implant interactions. (Left) shows the frictional/hard contact interaction between the femoral and meniscus parts. (Right) shows the frictionless/hard contact interaction between the tibial and meniscus parts.

The load used in the simulation was a cyclic load. The group was confused on how to create a proper cyclic analysis, and so used a concentrated load with amplitude. The load was made to be -1 in the CF2 box (y-axis), have an amplitude with circular frequency of 1.7 Hz (average male walking frequency), a starting time of 0, an initial amplitude of 2168.5 N, and A & B coefficients of 1. This load was applied to the entire femoral implant as shown below. However, the model could not run and had several errors. This ultimately contributed to the decision to analyze just the meniscus plate.

Figure 13: Cyclic load is applied to the femoral implant.

Meniscus Plate Model

The meniscus plate model is a simplified version of the total knee implant. A general static step was applied to the model in which the total time was set to 5.1 seconds. Since the average gait cycle for a human is 1.7 seconds, the simulation ran for an equivalent of 3 steps. The increment was set to 0.85 seconds since the model only considers one knee joint. A periodic amplitude was also defined. The circular frequency was defined as 1.85 seconds. Since the model only considers one knee joint, only half of the gait cycle is used, therefore the 1.7 seconds can be divided by 2. The cycle rate of 0.85 cycles/second converted to rad/s is equal to 1.85 seconds. While walking, only a compressive load is applied to the knee. The initial amplitude was set to negative 1 so only a compressive force is applied to the knee. The group only wanted the amplitude to model a cosine wave for simplicity; therefore, the coefficients A and B for the amplitude were set equal to 1 and 0, respectively.

Section 6: Analysis of Finite Element Model

Simplified simulation

In order to understand the process of applying cyclic loading to a problem better, a simple, one element problem was created. The element was a 1 m x 1 m cube with the base constrained. The material used for the simulation was not relevant as only the cyclic loading was being examined. However, steel was used as the material for the simple analysis. Two steps were used in the problem, an initial step and a direct cyclic step. Within the direct cyclic step, the cycle time period was set to 10 seconds which is the total run time. In the incrementation tab, the parameters were selected as fixed and the increment size was set to 1. All other parameters were left to the default settings.

An amplitude was created where the time span, circular frequency, starting time, and initial amplitude were defined. Figure 14 displays the values that were used in the simulation. Next the boundary conditions were defined. The base of the cube was constrained and a displacement of -0.1 m was defined for the top of the cube as shown in Figure 14. Within the displacement boundary condition, the amplitude was defined as Amp-1.

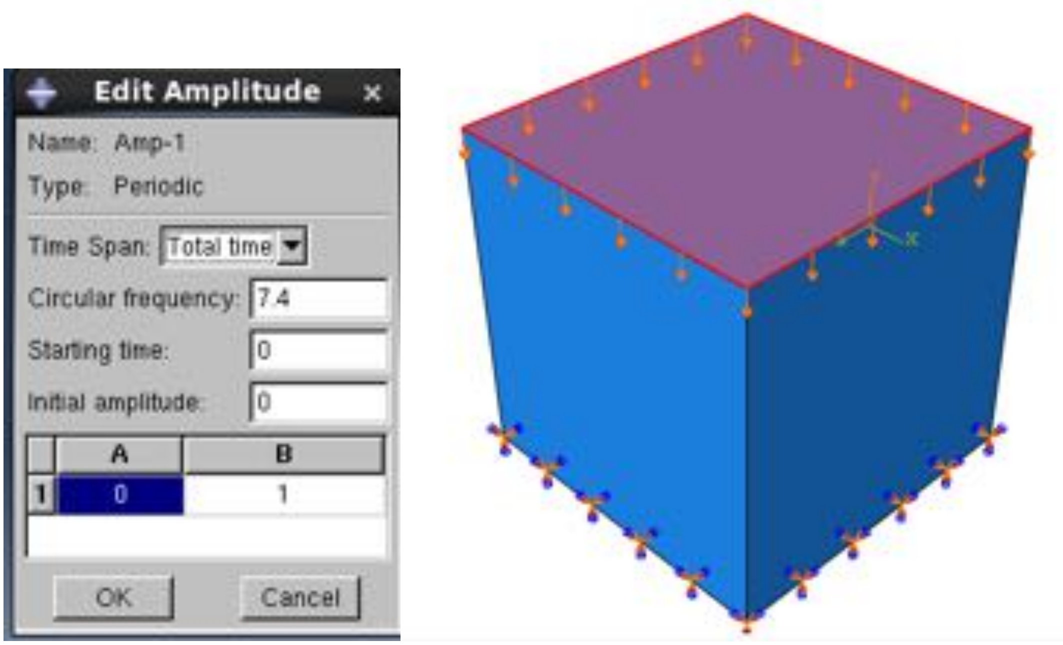

Figure 14: Amplitude and boundary conditions of simplified model.

The cube element was then meshed using default settings and the job was run through the ICS system. In the results, the model can be toggled between the 10 time steps and observed for displacement and stress. The cyclic step function did work, however, difficulty occurred when trying to understand how to use the periodic function of the step. With the simplistic problem, the cyclic step function was better understood. In the figures below, the difference in displacement can be seen from time step 0 to time step 20. The total time in the images is different than originally stated, but the same results occur for the different cycle times.

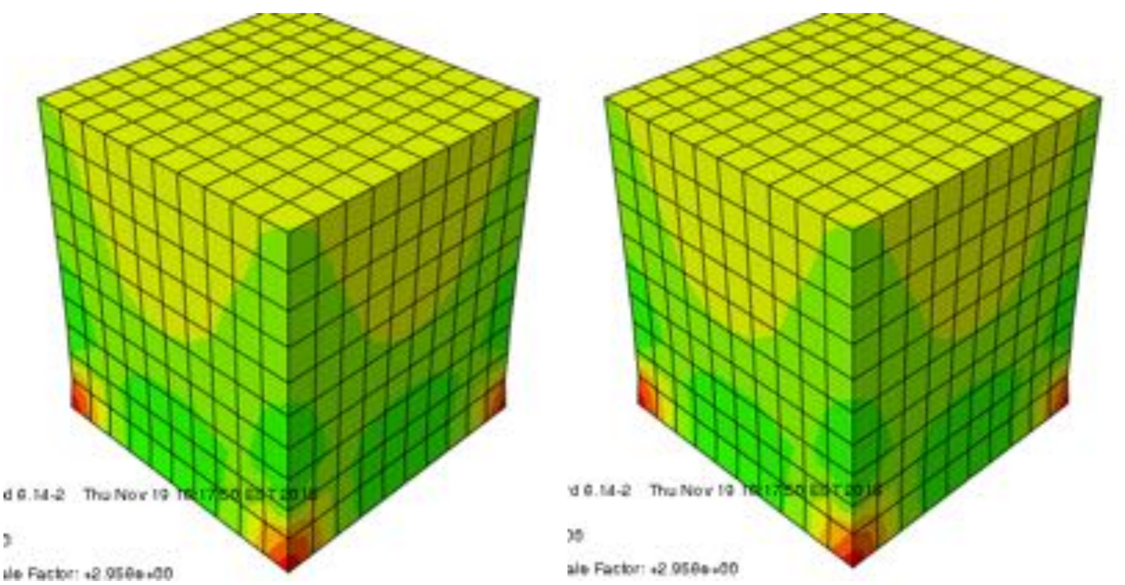

Figure 15: (left) Stress and deformation of the cube at time step 0. (Right) Stress and deformation of the cube at time step 20. The contour plot of each cube appears to be the same, however the magnitude of the stress represented by each color differs.

Meniscus Plate Model

Using what the team learned from the simple model, the concepts were applied to the meniscus plate model. The model characteristics were essentially the same, only changing the geometry from the simple block to the meniscus plate.

Section 7: Summary of Major Findings

Using the meniscus plate model, the team was able to determine the stress applied to the meniscus for the cycle of 3 steps. XY data was generated to calculate the stress at the centroid of one element over the time period. The element where XY data was generated from is highlighted in Figure 19 of Appendix 1. Figure 16 displays the stress over 5.1 seconds. The plot form makes sense as if an individual were walking for 5.1 seconds, one knee joint would be loaded twice. The triangle wave displays how the knee gradually takes a load until the maximum loading force, then the load gradually decreases. The peak stress in this plot is 6.3 kPa.

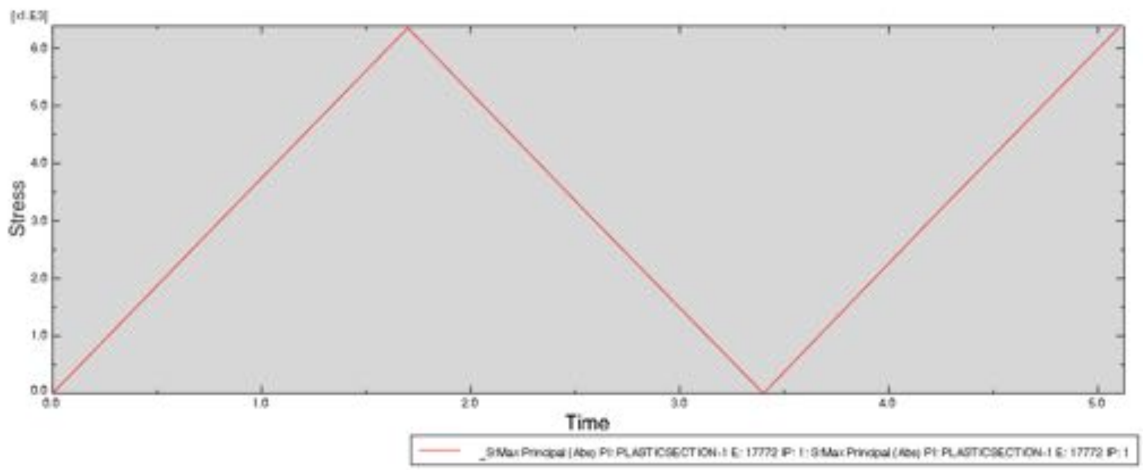

Figure 16: Cyclic stress from element centroid of meniscus.

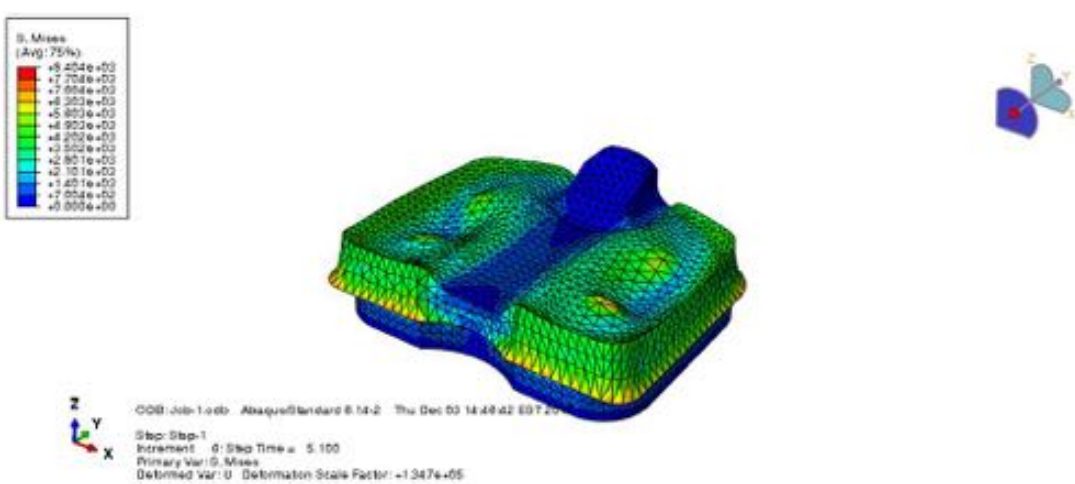

Figure17: Contour plot of meniscus at 5.1 seconds. The deformation appears to have an abnormal shape.

Figure 17 above displays the contour stress plot of the meniscus plate. The deformation scale factor is 1.247e5 which amplifies the displacement. The displacement takes an abnormal shape, something that will need to be evaluated further. Figure 20 in Appendix 1 displays how stress propagates through the part. Stress was also measured along a path to observe quantitatively how stress propagates throughout the material. As shown in Figure 21 of Appendix 1, the path was chosen through the area of material in the Z direction.

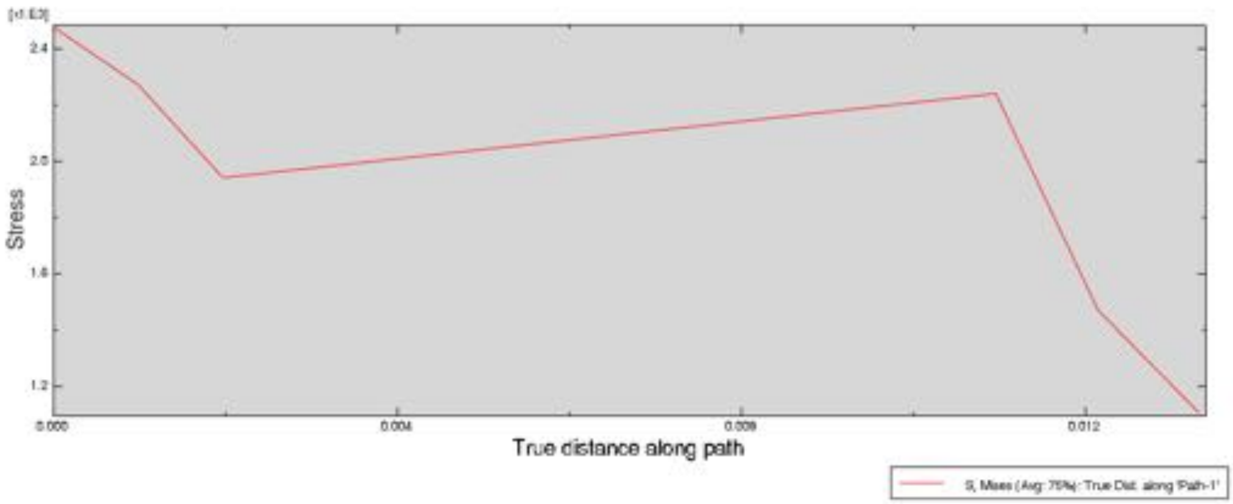

Figure 18: Stress along the Z axis path through the meniscus plate.

As shown in Figure 18, the stress along the path is greatest at the surface that interacts with the femoral part. The stress sharply declines and then steadily rises through the material then sharply declining again at the end of the part. The stress data gathered from the centroid peaked at 6.3 kPa and does not match the stresses collected along the path. Further analysis is necessary to understand the differences in stress magnitudes.

A modal analysis was also conducted for the meniscus part. The modal frequencies were determined for the first 3 eignevalues. The modal frequencies are shown in Figure 22 of Appendix 1, but are extremely high. An artifical knee will never be operated near this frequency, therefore there is no danger presented to the patient from vibration. Stress contour plots of the knee over the course of the three modal frequencies can be observed in Figure 23 of Appendix 1.

Fatigue Life Analysis

From the results of the meniscus plate model, the fatigue life of the artificial knee could be determined. The maximum and minimum stresses exerted on the meniscus plate were 6.3 kPa and 0 respectively. The minimum stress being zero is attributed to the ideal situation where the knee is completely unloaded, even though there is residual stress on the knee at all times in real life. From these values of maximum and minimum stress, as well as the material properties of UHMWPE, the fatigue life was determined as shown in Appendix 2. The fatigue life was determined to be 2.45×10^{70} cycles. At 0.85 cycles per second, the life of the meniscus plate is approximately 6.604×10^{62} years. Our group was initially shocked at this value, but upon further analysis, the number seems logical given the nature of the simulation and fatigue calculations. Our model subjected the meniscus plate to a maximum stress of 6.3 kPa, but UHMWPE's ultimate tensile strength is 40 MPa. The alternating stress of 3.15 kPa that occurs in the knee is well below this ultimate tensile strength, therefore a conclusion can be made that this low of a stress causes little fatigue on the part. With such little fatigue stress on the meniscus plate, it has an almost infinite fatigue life. Secondly, the nature of fatigue diagrams shows that certain materials "knee" at a certain point called the fatigue limit. At the fatigue limit, the material essential can have an infinite fatigue life. The fatigue limit for the meniscus plate was calculated to be 20 MPa, which is much greater than the alternating stress of 3.15 kPa. Therefore, the meniscus plate has an almost infinite fatigue life based on the FEA model conditions.

Conclusions

Our meniscus plate model showed that it has an infinite fatigue life for the modeling conditions of cyclic loading in one direction. However, the model did not account for any other forces that occur on knees in real life. Knees experience torsional forces in many directions as well as compression forces in more than one direction. Therefore, the actual fatigue life of the meniscus plate should be determined using a more accurate model that has compressive and rotational forces representative to those that are applied to an actual knee. Furthermore, the actual mode of failure for an artificial knee could be more complex than that of fatigue. Adhesive wear can cause chips of material to dislodge from the meniscus plate and femoral implants, thus causing abrasive wear to occur and speed up the failure of the knee joint. Other modes of wear and fracture could also occur due to effects inside the body and loading conditions of the knee.

In addition, several other factors were not included or considered in our model for the sake of simplification—but should be considered in more accurate model. For instance, biocompatibity, which is one of the most important safety factors of this device, was not explored. However, for future studies, biocompatibility would have to be factored in. This would be especially important if applying new material properties to parts. These new materials could pose, among many problems: poisoning of the individual, corrosion due to bodily fluids, and other harmful conditions. Also, similar to biocompatibility, other health factors would most likely have to be factored into the model in order to improve it. As an example, a person who suffers from arthritis of the hip joint, or even a hip replacement in addition to their knee replacement, would significantly change the user's gate and disturbed forces on the knee. This could significantly change the mode of failure of the device.

In summary, the model produced in this study, although simplified, provides a quality basis for the study of downward cyclic loading on the meniscus plate of a knee implant. The simulation found a relatively infinite life for the failure mode examined in this study. This would indicate that other failure modes are more likely causes of the knee implant failing. Real world examinations of failure of knee implants backs this assertion, as abrasive wear due to friction is the most common mode of failure. This simulation holds real value as many commercial companies are attempting to lengthen the life span of their knee implants from the relatively short 10-15 year lifespan they are currently at. In the future, many factors including biocompatibility and abrasive wear need to be added to the model; however, as it stands, the FE model produced is a quality basis to begin analyzing all of the factors which lead to the durability of a knee implant.

Section 8: Works Cited

1. https://grabcad.com/library/knee-implant
2. http://www.sciencedirect.com/science/article/pii/S1018363912000025
3. http://www.arthritisresearchuk.org/~/media/Images/Arthritis-information/Knee-replacement/12631-AR-Knee-Replacement.as
4. https://en.wikipedia.org/wiki/Fatigue_%28material%29
5. https://www.efatigue.com/training/Chapter_4.pdf
6. http://academic.uprm.edu/pcaceres/Courses/INME4011/MD-6B.pdf

Appendix 1: Figures

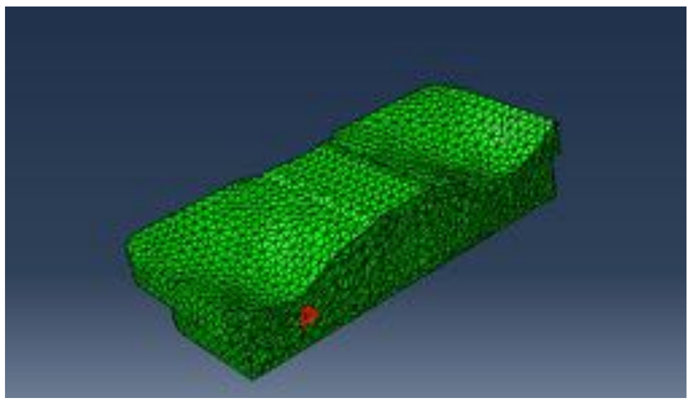

Figure 19: Cutaway view of the part highlighting the element that XY data was generated

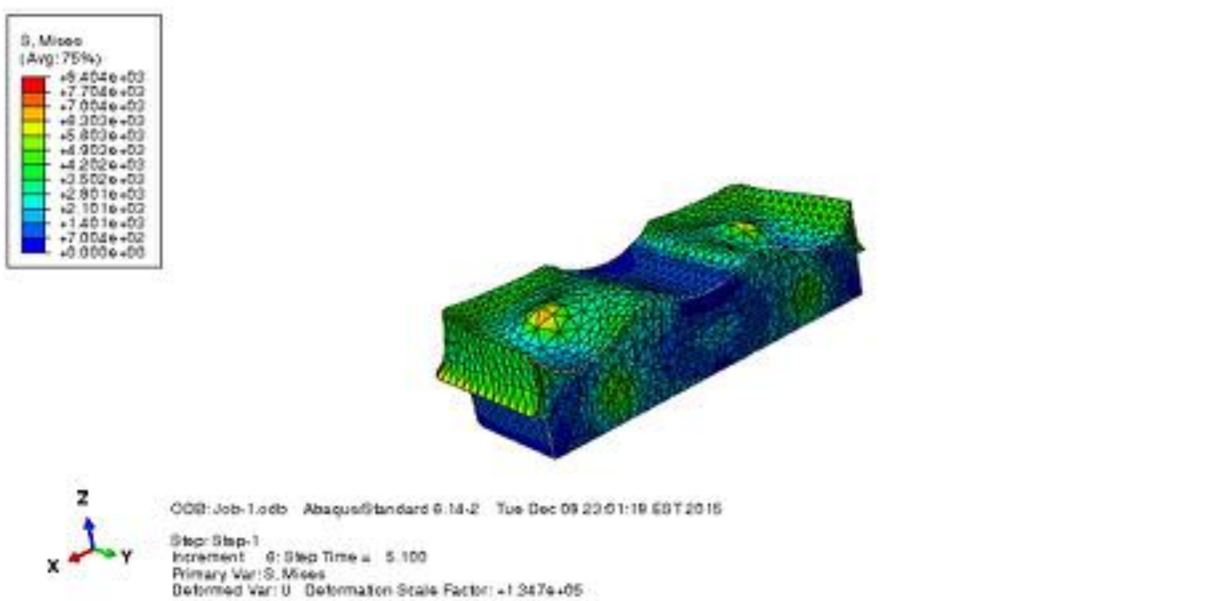

Figure 20: Contour stress plot of meniscus cutaway.

Figure 21: Cutaway view of meniscus highlighting the path that the stress XY data was measured at.

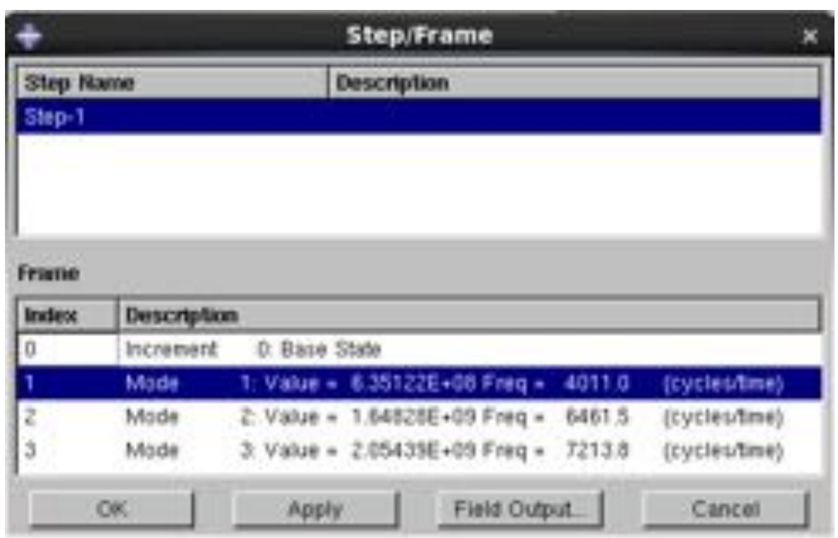

Figure 22: Modal frequencies for the first 3 eigenvalues of meniscus part.

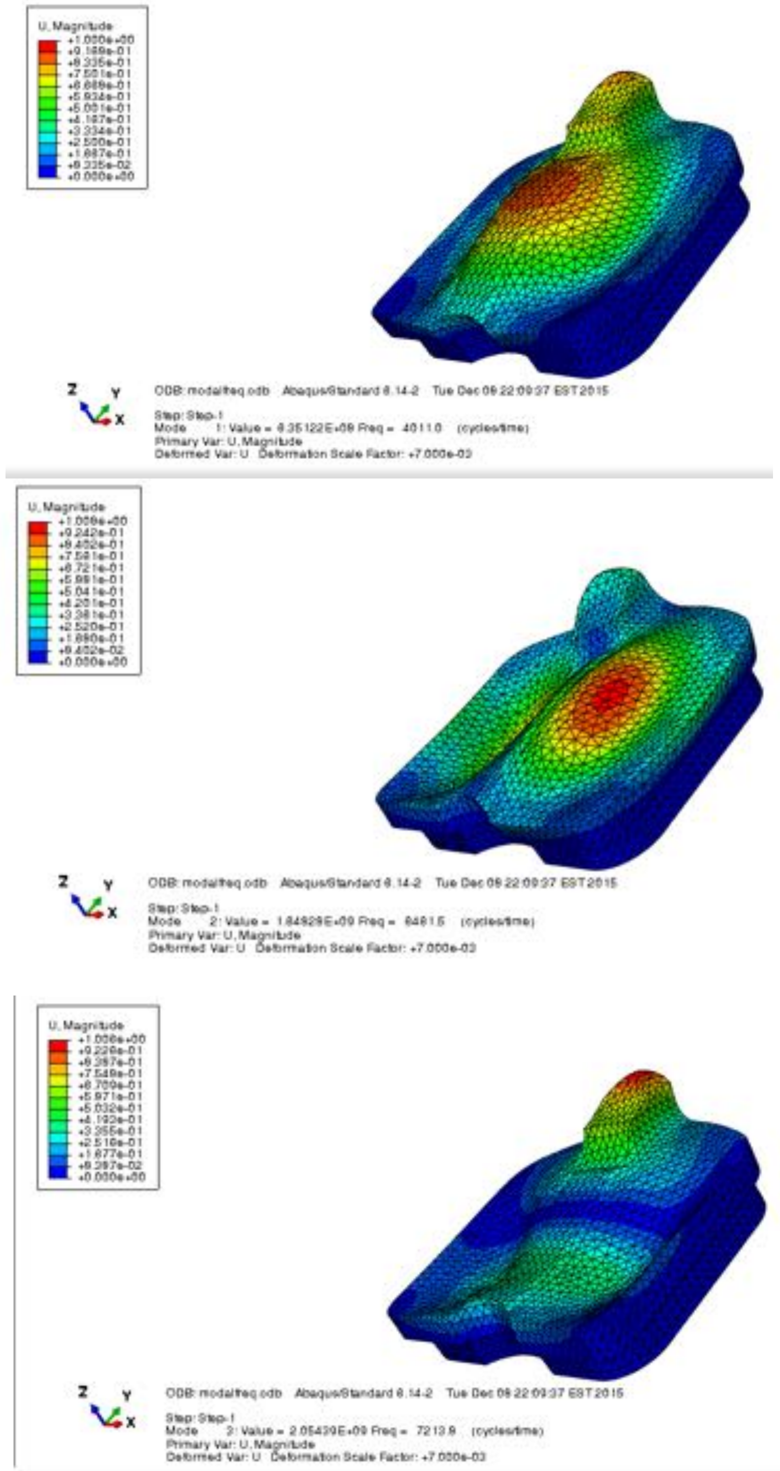

Figure 23: Contour plot at each modal frequency of the meniscus part.

Appendix 2: Fatigue Life Calculations

The image below shows the governing equations for the fatigue life calculations.

$$S_n = aN^b$$

$$S_n = S_m \quad at\ N = 10^3$$
$$S_n = S_e \quad at\ N = 10^6$$

$$axial\_loading : ....S_m = 0.75S_{ut}$$

$$S_e' = 0.5S_{ut}$$

$$S_a = \frac{S_{\max imum} - S_{\min imum}}{2}$$

1. Alternating Stress S_a

   $S_a = S_{max} - S_{min}/2$
   From model, $S_{max} = 6.3\text{x}10^3$ Pa and $S_{min}=0$
   $S_a = 3.15\text{x}10^3$

2. S_m

   $S_m = 0.75*S_{UT}$ for axial loading
   $S_m = 0.75*40$ MPa
   $S_m = 30$ MPa

3. S_e

   $S_e = 0.5*S_{UT}$
   $S_e = 0.5*40$ MPa
   $S_e = 20$ MPa

4. Coefficients a and b

   log(N) = log(a) – b*log(S_n)
   log(N) + b*log(S_n) = log(a)

   log(1E3) + b*log(30E6) = log(a)
   -log(1E6) + b*log(20E6) = log(a)
   log(1E3) – log(1E6) + b*[log(30E6) – log(20E6)] = 0

b = [log(1E6) – log(1E3)] / [log(30E6) – log(20E6)]

b = 17.04

log(1E3) + 17.04*(30E6) = log(a)

a = 10^{130}

5. N at $S_n = S_a$

log(N) = log(10^{130}) – 17.04*log(3.15E3)

N = 2.45×10^{70} cycles

N = 6.604×10^{62} years