CYCLIC FINITE ELEMENT ANALYSIS OF TI-6AI-4V TITANIUM SPECIMEN

Omar Lee
Alexandra Schonning
Jutima Simsiriwong

University of North Florida

omarlee148@gmail.com; aschonni@unf.edu; j.simsiriwong@unf.edu

Abstract: In this paper, finite element simulations of TI-6AI-4V titanium cylindrical dog-bone specimens subject to sinusoidal loading mimicking specimens loaded in an MTS axial fatigue testing machine are presented. The geometry was modeled according to ASTM E466 standards, and the specimens were modeled using second order tetrahedral elements. The load was applied axially, and symmetry constrains were imposed on three surfaces. A static load was first applied axially, and the results were used in the durability fatigue and life analysis application of Siemens NX. A strength-life plot was created for the specimen. The simulations are performed to help determine how best the finite element methodology (FEM) can be used to guide experimental cyclic testing for surface treated specimens.

Key Words: *Ti-6AI-4V Titanium, durability, cyclic analysis, finite element analysis, FEM, FE*

1. INTRODUCTION

Medical implants are subject to highly variable and cyclic loads and need to last a long time as replacement requires surgeries. As such, implants are commonly made from titanium Ti-6Al-4V because of its high strength and corrosion resistance. Still, Ti-6Al-4V is not perfect and to optimize the implants, further understanding of the material is needed. While Ti-6Al-4V is corrosion resistant, it still contributes to implant failure (Antunes and Oliveira, 2015). Because the surface roughness is a contributing factor to corrosion fatigue behavior (Zhang and wang, 2022), one way to improve the material is to change its surface characteristics though coating or shot peening (blasting the surface with a stream of objects (Campbell,1971)). In determining if and how this improves the cyclic characteristics of the material both experimental and computational studies are being conducted. The computational study is performed using finite element analysis (FEA) and the technique will be verified by the experimental analysis. The FEA will allow researchers and implant designers the tools to analyze the cyclic behavior of the material subjected to specific shapes and loads, which is difficult and expensive to do experimentally. This paper presents the first stages of developing such a FE model. A dog-bone shaped Ti-6Al-4V specimen is subjected to cyclic load and the cyclic behavior of the material is presented though an S-N curve. Next stages of the research will include surface treatment and comparisons to experimental data.

2. METHODOLOGY

The methodology applied consisted of developing a Computer-Aided Design (CAD) model of a dog-bone specimen, reducing it to an 1/8 volume to allow for symmetry, and then preparing the model for finite element analysis by applying a mesh, material, boundary conditions, and loads. The finite element model was then solved first using static analysis and then cyclic analysis.

3. CAD MODEL PREPARATION

A dog-bone specimen was modeled using the CAD application of Siemens NX. The dimensions followed recommendations of ASTM specification E466 (ASTM,2021), as shown in Figure 1. To allow for symmetry conditions during finite element analysis, the CAD model was cut along three symmetrical planes resulting in the 1/8 model geometry shown in Figure 2.

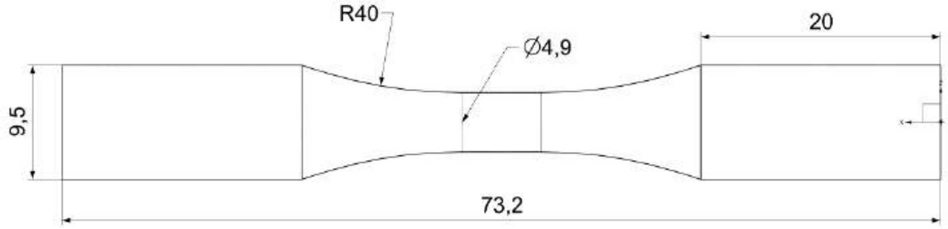


Figure 1 Dog-bone shaped specimen with dimensions (mm)

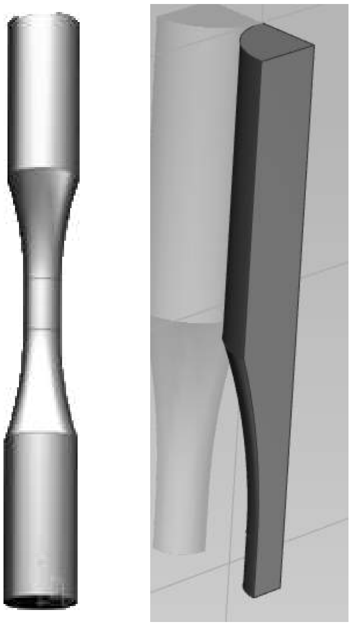


Figure 2 CAD model before and after modifying for symmetry boundary conditions

4. MATERIAL AND MESH

NX Siemens was used in conducting the finite element modeling and analysis of the specimen. The Ti-6AI-4V titanium was modeled using the following properties (Antunes and Oliveira, 2015): Young's Modulus of Elasticity of 12.1GPa, Poisson's ratio of 0.34, a Yield Strength of 805kPa and an Ultimate Tensile Strength of 845MPa.

Second order tetrahedral elements (CTETRA (10)) were used in modeling 1/8th of the dog-bone specimen. The mesh consisted of 3165 elements and 5735 total nodes. An h-refinement method was applied to the mesh to ensure convergence. The number of elements vs. von Mises Stress are reported in Table 1 which shows that the stress changed less than 2% as the mesh was refined from 223 to 3165 elements. While it can be argued that the coarser mesh could have been successfully used, the finer mesh of 3165 elements was used as the time to solve the model was insignificant.

Table 1 H- Refinement method

Number of Elements	223	373	567	709	827	870	1471	1666	3165
Von Mises Stress (MPa)	218.62	217.75	216.62	216.19	215.73	215.68	215.66	215.32	215.22

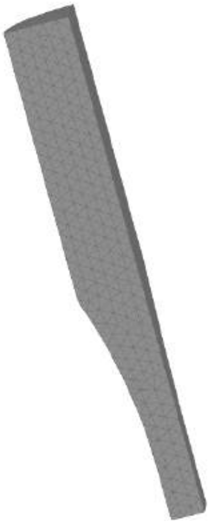


Figure 3 Meshed Model

5. BOUNDARY CONDITIONS AND LOAD APPLICATION

To better control the symmetrical behavior of the model during finite element analysis symmetry conditions were applied to three planes and resulted in a 1/8th of the original object being modeled and analyzed. Boundary conditions were applied to restrict motion across the planes of symmetry. The CTETRA (10) element only has three linear degrees of freedom, and as such the constraints were applied to all nodes on the planes of symmetry and hindered motion perpendicular to the symmetry planes.

The load was applied to simulate the load that would be applied to a physical specimen in a Material Test System (MTS 858 MiniBionix). The FE load application was slightly modified from the physical application in that it was applied to the flat circular surface rather than though a grip on the cylindrical surface. Because the area of interest (waist of the specimen) is far from the location of load application, this simplification is not expected to significantly alter the results at the location of interest. The load was applied in the axial direction of the specimen.

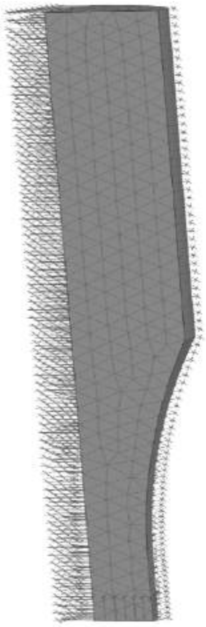


Figure 4 Applied constraints on one of the symmetry planes

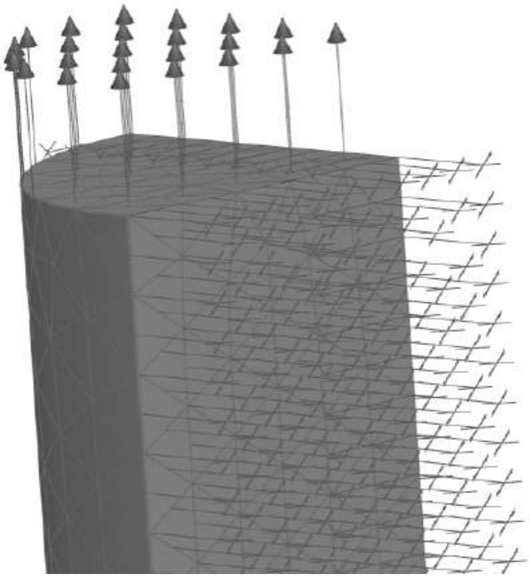


Figure 5 Applied loading conditions

An arbitrary load magnitude of 1000 N was first applied to the specimen. Because the analysis was linear in nature, the maximum allowable load for static analysis could be calculated by linearly scaling the results from the arbitrary load using Equation 1.

Maximum allowable load=Applied load x (Ultimate Tensile Strength /Maximum von Mises Stress from arbitrary load) Equation 1

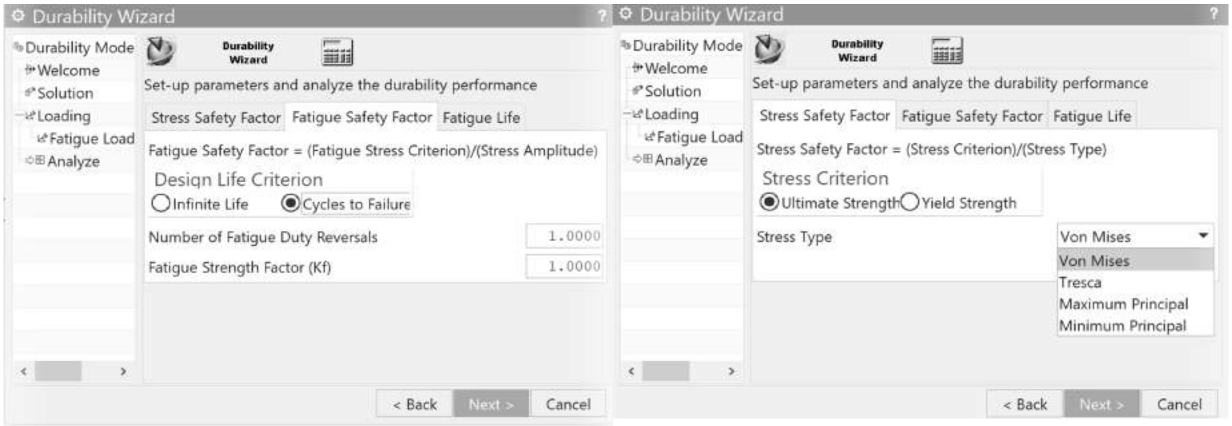
6. STATIC ANALYSIS

The static analysis was performed to determine the maximum allowable load for one cycle so that those results could be used for cyclic, or durability analysis. From the linear scaling in Equation 1, the maximum load was calculated to be 3983 N.

7. DURABILITY ANALYSIS

NX Siemens Durability application was used in determining the number of cycles that can be applied to the specimen for various loads, allowing the creation of a strength-life (S-N) diagram. For this analysis, the loading pattern was set to Full Unit Cycles to best replicate how the specimen will be tested in the MTS machine.

As seen in Figure 7, the stress criterion was set to Ultimate Strength, the stress type to von Mises stress, cycles to failure were used for the design life criterion, and stress life (high cycle fatigue) was used for the fatigue life criterion.



Set-up parameters and analyze the durability performance

Fatigue Safety Factor = (Fatigue Stress Criterion)/(Stress Amplitude)

Design Life Criterion

○ Infinite Life ● Cycles to Failure

Number of Fatigue Duty Reversals 1.0000

Fatigue Strength Factor (Kf) 1.0000

Set-up parameters and analyze the durability performance

Stress Safety Factor = (Stress Criterion)/(Stress Type)

Stress Criterion

● Ultimate Strength ○ Yield Strength

Stress Type Von Mises

Figure 6 Durability parameters selection

8. RESULTS AND CONCLUSIONS

For the static analysis, the material behaves linearly and the maximum allowable force that could be applied was calculated to be 3983 N based on results from an arbitrary load application and the material's ultimate tensile strength of 845MPa. Figure 6 shows the von Mises stresses resulting from the arbitrary and maximum allowable load. As expected, the results have the same distribution.

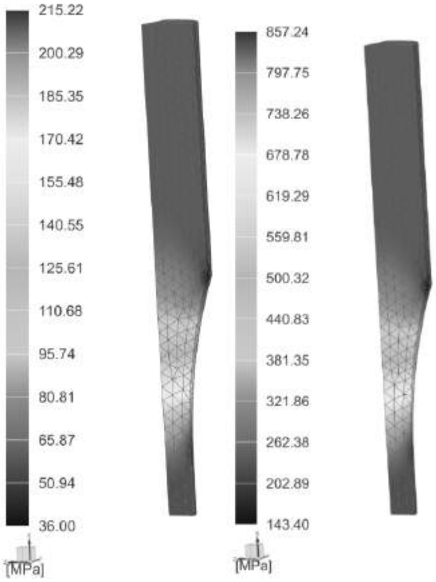


Figure 7 Von Mises stress results from arbitrary load (left) and maximum allowable load (right)

For the durability analysis, the number of cycles to failure were computed for various loads ranging from 100% of the maximum static load applied (3983N) to 25% of the load applied (996N), which was determined to be the infinite life of the model at 100 million cycles. The load was reduced in 3% increments from the 100% load. The resulting data is plotted on the strength-life (S-N) curve shown in Figure 8.

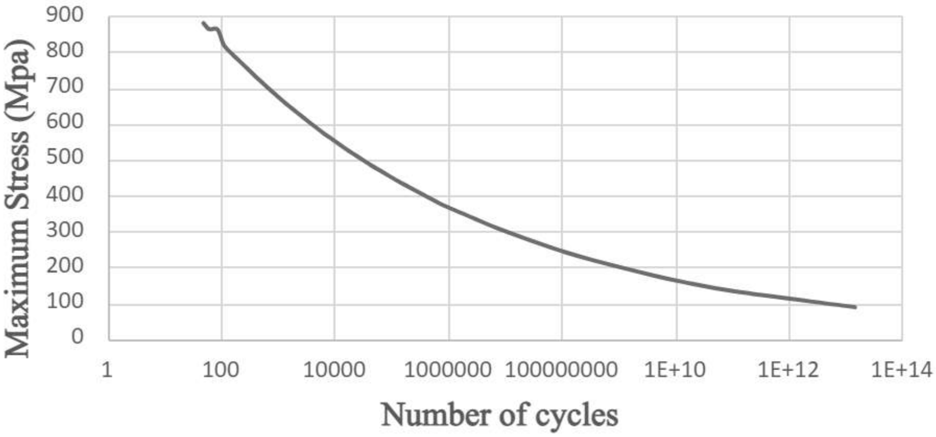
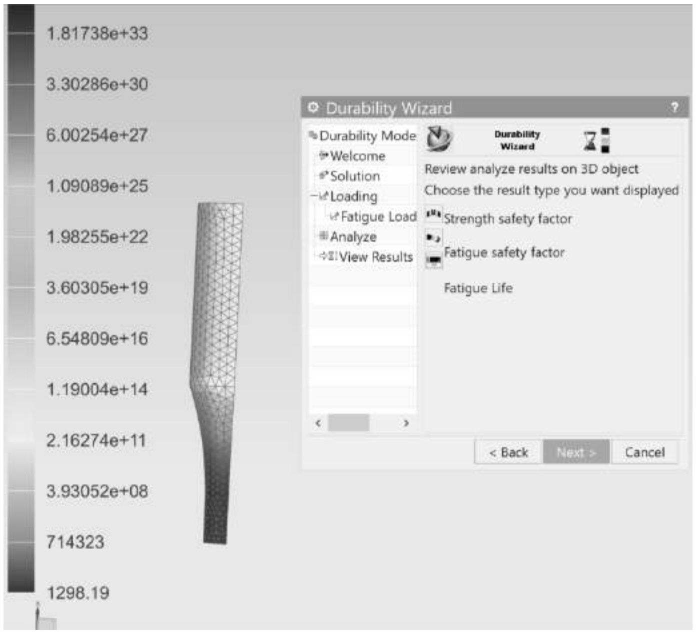


Figure 8 S-N curve made from Durability Simulation

The number of cycles computed for one of the cyclic analysis is shown in Figure 9. The gage length of the specimen fails at significantly fewer cycles than the grip section, supporting the validity of the simplification of load application.



Review analyze results on 3D object

Choose the result type you want displayed

Figure 9 Cycles to failure during durability analysis

To continue to improve implants it is vital that the behavior of the material is well understood and that it continues to be optimized for the human body environment and load applications. The presented FE analysis is the first step in developing a FE model that can be used for cyclic analysis of various implants. Improvements of the model will be to model a different surface finish and then later to apply the material to geometries of implants and various loading scenarios.

9. ACKNOWLEDGMENTS

We are grateful to the Auld and White Foundation for their support of this project.

10. REFERENCES

[1] Antunes, R. A., & M. C. L. de Oliveira. (n.d.). vol. 8, no. 3, pp. Acta Biomaterialia, 8(3), 937–962.

[2] Campbell, J. E. (1971). Metals and Ceramics Information Center, Columbus, Ohio, United States.

[3] NX Siemens finite element library. (Accessed April 2022).

[4] Standard Practice for Conducting Force Controlled Constant Amplitude Axial Fatigue Tests of Metallic Materials (ASTM E466). (2021). ASTM International(DOI: 10.1520/E0466-15).

[5] Zhang, Q., S. Xu, J. Wang, X. Zhang, J. Wang, & C. Si. (2022). Surface topography metrology and, 10(1), doi: 10.1088/2051-672X/ac4c83.