FATIGUE FAILURE ANALYSIS OF A GENERIC IMPLANT: A FINITE ELEMENT APPROACH

[1]Shivkumar Umadi*, [1]Vyjayanthi Murthy, [2]Dr. Suresh Nagesh, [3]Dr. Venkateswaran Perumal
[1]Research Associate, CORI, PES University, Bangalore, India
[2]Chair Professor and Head CARE, CORI, PES University, Bangalore, India
[3]Principal Engineer,Stryker Global Tech Center Pvt. Ltd, Gurgaon, India

KEYWORDS – Finite Element Analysis, Titanium implant, Fatigue Life.

ABSTRACT
The purpose of this research is to perform Stress and Fatigue Analysis of an Implant-based on the load cases and parameters typical for such implants. A 3D Finite Element Analysis (FEA) isperformed on a generic orthopaedic implant to estimate stress and displacement fields on each component of the implant. A titanium alloy implant is then evaluated for worst-case loading conditions. The pre-processing of the analysis is carried out on ANSA. The finite element resultsare then further post-processed in a FE-Fatigue software under uniform cyclic loading to evaluate the life and damage. In this study, the Fatigue life and Damage contours of the generic implant subjected to the load is also calculated considering additional variables including the size and orientation w.r.t. loading and boundary conditions. Close to a million cycles have been recorded with the structure showing minimal damage.This isconsidered safe for the loading conditions with requirements claimed by prior simulation and experimentation.

TECHNICAL PAPER -

1. INTRODUCTION

The need for biomedical materials for implants is increasing with the advancement in approach to clinical cases and medical technology needing artificial joints or bone plates.The major usage of metallic biomaterials includes dental, screws for fixation and artificial joint applications.Tremendous development has occurred in the field of biotechnology in the recent past with the advancement in technology and automation playing a pivot role in this perspective[3] Various materials including alloys are being developed and tested for tensionand fatiguecriteria for such implants. Since the human body space is limited, the biomedical materials should have enough torsional and tensile strength and the required mechanical properties as the artificial implants undergo high loads whenin use. Clinical failure of such generic Arthroplasty is categorized as a soft tissue, with change and alteration in loosening, periprosthetic infection, wear, or deltoid muscle dysfunction; however, in many cases, the origins of failure are multifactorial.[5]
Ti-6Al-4V(TC4) alloy is widely used alloy in medical applications as it possesses high corrosion resistance, which forms an important aspect for medical applications. In hip and knee prosthesis, the fatigue-wear interaction is considered as an important parameter in the ultimate failure of the medical devices.[6] Design oforthopaedic implant also plays a vital role as the notches and curvatures affect failure criteria to an extent as well. In the case of aprosthesis, fatigue failure is caused by repeatedly applied loads with progressive and

localized structural damage which occurs when a material is subjected to repeated loading and unloading. The graph of the magnitude of cyclic stress (S) against the logarithmic scale of cycles to failure (N) is considered for each material data for analysis. S-N curves are derived from tests on samples of the material to be characterized where regular sinusoidal stress is applied by a testing machine which also counts the number of cycles to failure. The outcome of a fatigue analysis is life and damage estimates. These contours give insights of failure criteria and the cycles the body has undergone before instincts of failure. The fatigue life prediction is one of the important factors in the design of medical devices. The purpose of the study is to calculate the fatigue life of the implant with a combination of biomaterial alloys subjected to loading.

2. DESIGN CONSIDERATIONS AND PREPROCESSING

Based on the design aspects of usage and Glenoid rotation parameters, an initial Design of the Humeral implant with addition of bone fixtures was formulated. Considering the stress plots, the design has been varied to suit the curvatures to withstand the applied load. The structure of the load parameters used has been discussed further. The low strength and small volume bone available cause such designing constraints. Also, there are various prosthetic options which exist in the surgical management of conditions affecting the humeral joint. The design and dimensions vary due to age category primarily. The design of the implant is carried out on Computer Aided Design software- SolidWorks 2015 as shown in *Figure 1*and the pre-processing on BETA-CAE ANSA for fine mesh to be considered for FE Analysis. The finite element mesh of the intact generic implant including bone fixtures in shown below in *Figure 2*.

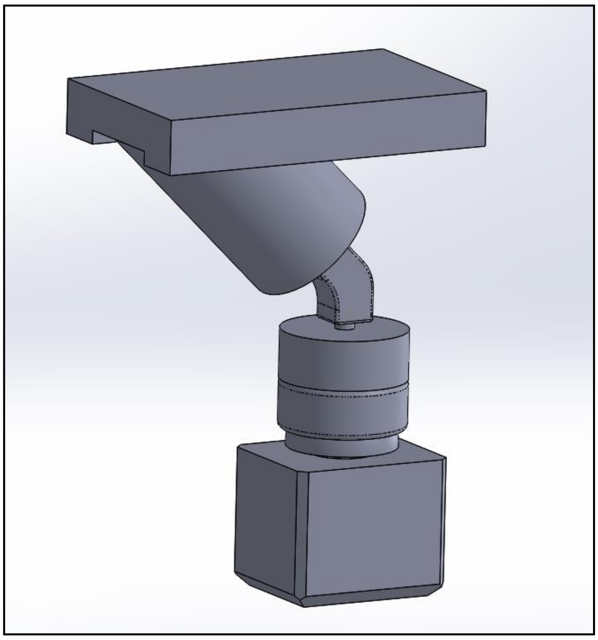

Figure 1: CAD Data

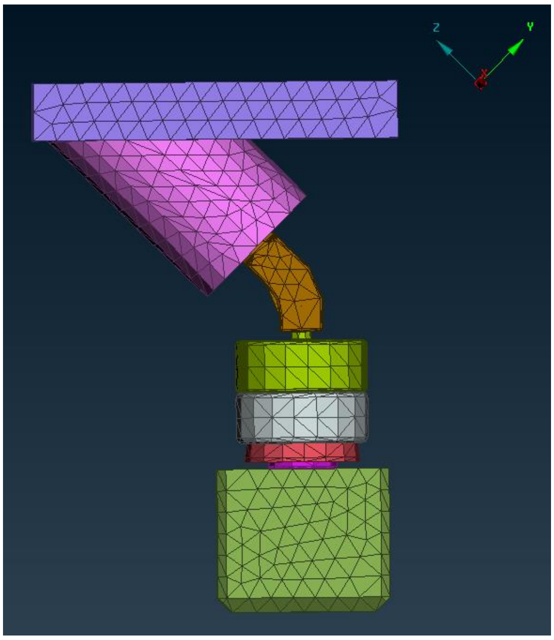

Figure 2: Meshed Model on ANSA

3. MATERIAL AND METHODOLOGY

In addition to the titanium alloy at the humeral body, Cobalt composite is usedfor the Glenoid sphere with Polyethylene going in the mix as well. Titanium screws are present on the head of the implant which attaches to the scapula bone. Bone block fixtures with same properties are used for experimental and simulation analysis. The polyethylene forms the cushion for the cobalt alloy as the Glenoid sphere gets in contact with the plastic spacer during movement, hence due to a lower strength of polyethylene, erosion of the material is very less.. The von-Mises stress contours are observedand further alteration in the design process is monitored and final design of the implant is made with a few iterations. Structural analysis is followed by Fatigue testing through numerical calculations and further on FE-Safe to get accurate results. This is important to understand the damage and fatigue life of the implant for safe and long lasting usage by the patient. Since the joint is a highly complicated setup which involves three anatomic joints, the scapula forms a key linkage and accomplishes a complex biomechanical function. The Glenoid cavity made out of Cobalt alloy is loaded with a spherical ball for rotation in the humeral head. Hence, modeling of the intact Glenoid, humeral component with FE Analysis is carried out for precise results.

4. DISCUSSION

Static structural FE Analysis is carried out to understand the stress plots and thereby changing the design for best-suited strength of the implant. Further, fatigue analysis is consideredfor combined shear and compressive stress to account for all the load that applies to this implant. Initially, the study of the effects of lateralization is carried out based on the parameters decided on the experimental setup. With a number of iterations, the load values used are 700N on the vertical side with 70N shear force. These same forces are used in all further analysis of the implant as they form the benchmark for the whole implant. The threads of the titanium screws show higher stress values due to its complex structure which allows more load action. Apart from the screws, the humeral stem with titanium alloy also have higher stress. However, these values lie below the yield stress values of the individual materials.

Conducting a finite element analysis requires knowledge of load vectors and moments with the mechanical properties considered and specifically used for a typical an upper extremity implant.[10] In our case, cobalt alloys have a wide range of yield strength and fatigue curves based on their forming and processing histories. Likewise, the titanium alloy forms a close counterpart for the determination of highest fatigue properties due to its properties of high tensile strength. Hence analyzing each of the materials is a task in itself asa combination of materials adds more stiffness with higher complications when considered individually. The titanium alloy component possesses α-β structures typical ofTi-6Al-4V.The metallurgical processing is not likely the cause of implant fractures since there are no apparent microstructural difference between the fractured and intact devices.No cracking was observed in the cobalt chrome components as well. These initial observations provided evidence that the titanium components with the smaller fillet radius may have been more prone to implant failurethan their cobalt chrome counterparts.

To determine the expected life and failure time point of the componentsaccording to the maximum von Mises stresses from the static finite element analyses,it was assumed that the loading condition had minimum stress of zero (i.e. not fully reversed) because it was expected that the patient returned to a neutral position after the loading condition.This cyclic condition corresponds to a stress ratio of 0. S-N curves for each material is determined and collected for a better comparison of the failure criteria. Thus, fatigue failure lifetime envelopes were determined for the design of bothcompressive and shear loading conditions and the bounds of the fatigue curves.

SIMULIA Fe-Safe is used as a tool for the Fatigue analysis of the generic implant. The software performs fatigue calculations on the stress history obtained from the finite element stress analysis using the total life approach. Contour plots illustrating the fatigue life of every part of the structure can be generated. Further CAD Model of the implant is updated with the addition of fixtures of bone and epoxy for analysis purpose. The implant is subjected to unit load in compression and shear with the forces considered individually as the other part is fixed. That is for compressive unit load, the bottom bone fixture is fixed inall DOF as shown in *Figure 3,* and vice versa for shear load shown in *Figure 4.* Then, the ODB files are transferred into the Fatigue software, the material properties are assigned and the fatigue analysis is carried out for cyclic loading. A harmonic amplitude of 750 units is employed for the compressive case and 70 units for the shear. The results show that the first signs of failure occur at the taper junction of the male humeral stem connecting the titanium alloy of the humeral component which is discussed in the further section.

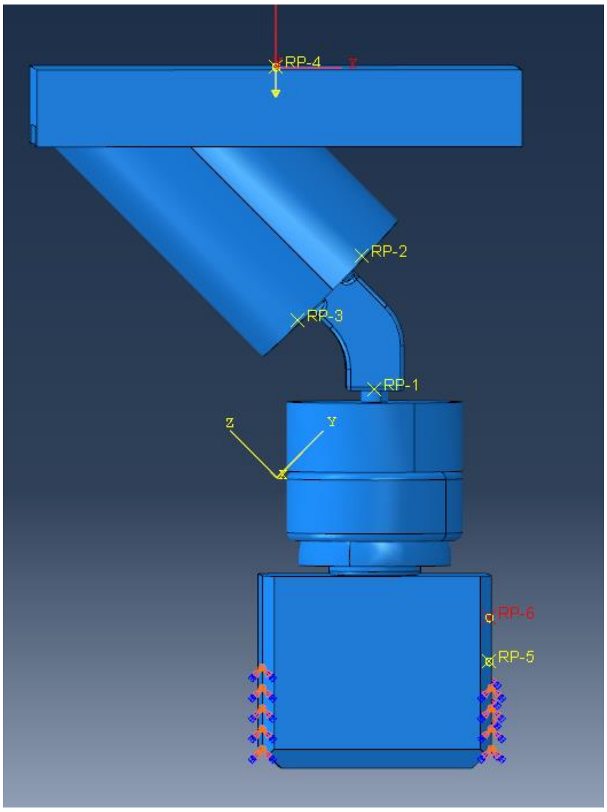

Figure 3: 1N Compressive Load

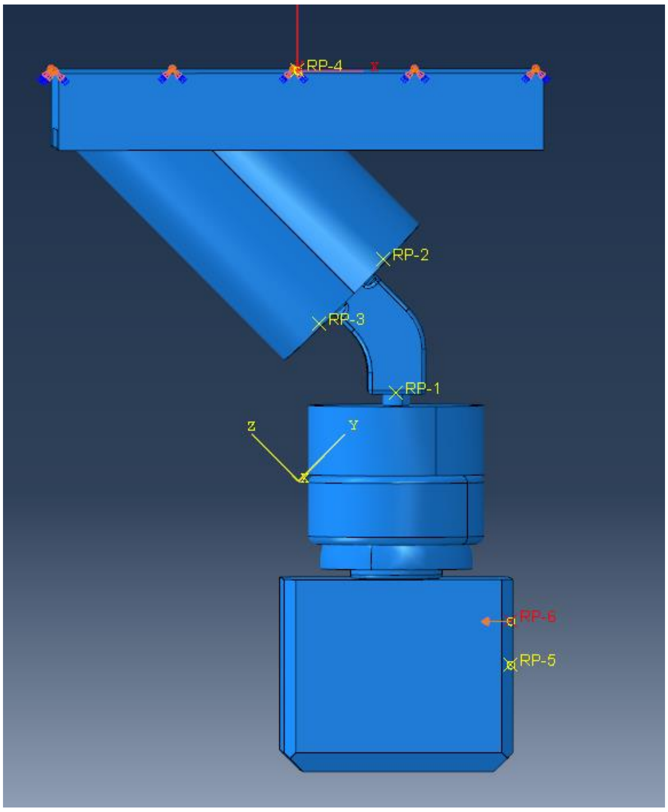

Figure 4: 1N Shear Load

5. RESULTS

The fatigue tests for the Implant was evaluated for understanding the relative effects caused by the change in geometry and loading. The Titanium alloy is most widely used biomedical material due to its high corrosive and high fatigue life properties. The fatigue tests were conducted on the implant for the applied loads. A combined compressive and shear loading is considered for maximum and minimum cases as well to create an envelope of life estimates for these components. The equivalent values were considered with the generic implant undergoing high cycles of loading. The S-N relation for the Ti-6Al-4V alloy is shown in *Figure 5*. The stress fatigue is depicted and corresponds to the fracture surface.

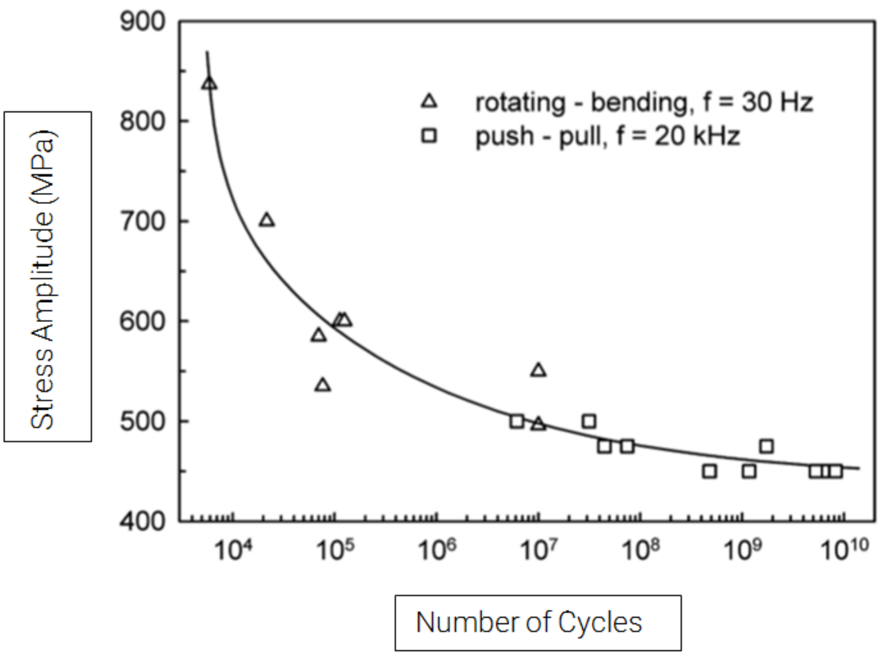

Figure 5: S-N curve of Ti-6Al-4V

Generally, fatigue life consists of both crack initiation and crack propagation lives. In some cases, the structures are designed for only crack initiation life. Here, the Log life depicts the number of cycles taken for failure of any component. The implant is designed toresist all the maximum stressed locations with increase in the fatigue life as well. The
Figure 6 below represents the Log10 of the Life expectancy of the entire structure subjected to the high cycle of loading as mentioned earlier for consideration of worst-case scenario This is measured in the number of cycles to failure on FE-Safe. The titanium alloy stem shows failure at a certain limit. The overall estimated life is 10^7 cycles which is acceptable. Typicallya million cycles is a benchmark for theoretically infinite life. Hence, the component is safe for the loading considered. No cracking was perceptible in the Titanium or the cobalt alloys used on the implant.

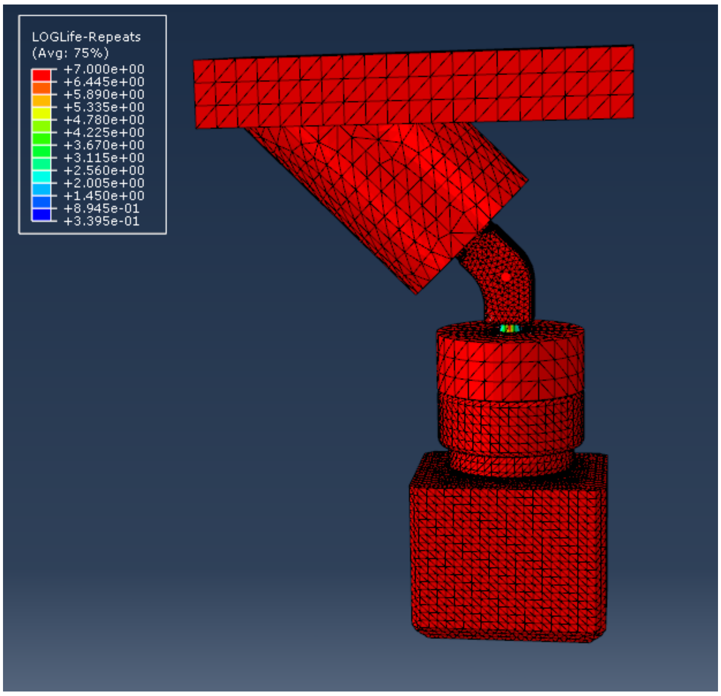

Figure 6: Log Life contour of the Implant

The fatigue life may be expressed in terms of the damage that is done to the structure by a prescribed loading sequence or as the number of repeats of the sequence that will cause the failure of the structure. *Figure 7* below shows the amount of damage that a material sustains under fatigue loading for a Fixation device.

A value of unity represents failure. The values secured here are close to zero hence there occurs no damage at all hence the implant is safe for the load applied.

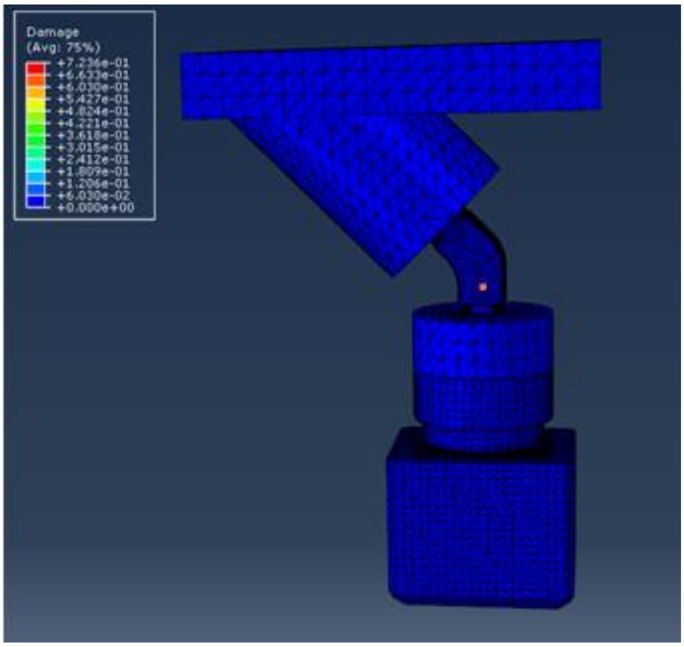

Figure 7: Damage contour

6. CONCLUSION

1. The importance of Titanium Alloy composite is quite high because of its mechanical properties suiting the human body hence widely used in Biomedical applications.
2. For the given loading determined by experimentation, the stresses fall well inside the individual component's yield strength based on the finite element analysis.
3. High cycle Fatigue analysis is carried to determine the life of the entire implant for the prior loading considered.
4. Close to a million cycles with no damage observed in the simulation software, hence the implant is safe undergoing cyclic loading.
5. ANSA provided all flexibilities in terms of modeling complex bio-medical implants.

REFERENCES

1. Drew D Moore, James E Moravek, Erin A Baker, Meagan R Salisbury, Kevin C Baker and J Michael Wiater, "Exploring Failure of Total Shoulder Arthroplasty Systems Through Implant Retrieval, Radiographic, and Clinical Data" Journal of Shoulder and Elbow Arthroplasty, Volume 1, 2017
2. Johannes A. Eckert, Ulrike Mueller, Sebastian Jaeger, Benjamin Panzram, and J. Philippe Kretzer, "Fretting and Corrosion in Modular Shoulder Arthroplasty: A Retrieval Analysis" Hindawi Publishing Corporation, BioMed Research International, Volume 2016, Article ID 1695906, 01 June 2016
3. Patrick J. Denard, Evan Lederman, Bradford O Parsons, Anthony A. Romeo, "Finite element analysis of glenoid-sided lateralization in reverse shoulder arthroplasty" Journal of Orthopaedic Research, 9 Aug 2016.
4. Nadeem Ali Bhatti, Kyvia Pereira, Magd Abdel Wahab, "A continuum damage mechanics approach for fretting fatigue under out of phase loading", Tribology International 117, 39–51, 2018
5. Teng Zhang, Noel M Harrison, Pat F McDonnell, Peter E McHugh and Sean B Leen, "Micro–macro wear–fatigue of modular hip implant taper-lock coupling", http://sdj.sagepub.com/content/49/1/2, The Journal of Strain Analysis for Engineering Design, 2014
6. María Prados-Privado, José Antonio Bea, Rosa Rojo, Sérgio A. Gehrke, José Luis Calvo-Guirado, and Juan Carlos Prados-Frutos, "A New Model to Study Fatigue in Dental Implants Based on probabilistic Finite Elements and Cumulative Damage Model", https://doi.org/10.1155/2017/3726361, Applied Bionics and Biomechanics, Volume 2017, Article ID 3726361, 2017
7. Korkmaz Y M, Coker D, "Finite element analysis of fretting contact for non homogenous materials", IOP Conf. Series: Materials Science and Engineering 295 Volume 012006, 2018
8. Michael Christopher Hogg, "Fretting wear and fatigue in taper junctions of modular orthopaedic implants", Doctor of Philosophy, The University of Sydney, June 2016.
9. E. Kobayashi, H. Mochizuki, H. Doi, T. Yoneyama, M. Otsuka and H. Hamanaka: Structural Biomaterials for the 21st Century, eds. M. Niinomi et al. (TMS, 2001) pp. 121–126.
10. ASTM F2028-14, Standard Test Methods for Dynamic Evaluation of Glenoid Loosening or Disassociation, ASTM International, West Conshohocken, PA, 2014, www.astm.org