

Fatigue Analysis of Endoskeletal Prosthesis using FEA

Joy H Jones*, B.Anil**

Abstract—Total replacement of affected part is successfully applied to patients affected by bone tumour, cancer etc. This avoids a high above knee amputation, and patient satisfaction is high. Nearly 14% of the locally made Indian Prostheses using 316L Stainless steel have structurally failed within short period of usage. It is estimated that the main cause of failure in implants is due to fatigue. Forces applied to the prosthesis due to human activity generate dynamic stresses and it is essential to ensure the prosthesis against fatigue failure. The rainflow cyclic counting method which is used to convert the irregular load histories to blocks of constant amplitude cycles have been now widely used in computational methods. In this study, a three dimensional model of the failed prosthesis is developed and Fatigue analysis have been carried out using commercial finite element analysis code ANSYS10 which utilizes rainflow method. Fatigue analyses have been performed under reduced gait cycle on the prosthesis model along with the distally replaced part of the femur bone model. The gait cycle was applied as one event and analysis were carried out for infinite life criteria. It has been observed that failure due to alternating stress occurs at critical location after 235 Cycles. Fatigue behavior prediction using FEA can be utilized for design modification.

I. INTRODUCTION

The distal femur is the most common site of osteosarcoma. Patients with osteosarcoma of the distal femur have traditionally been treated with a high above knee amputation. Developments in prosthetic devices made prosthetic reconstruction of the distal femur. The functional results are excellent and patient satisfaction is high. But nearly 14% of the total implantation of the newly designed Indian Prostheses were structurally failed within a short period of Usage [1-2]. Performance and success of long-term survival of cemented prosthetic replacement is related to the attachment of the prosthesis to the bone. Cement Metal interface failures, separation of the stem-cement interface and fractures in the cement may initiate the initial loss of the fixation of the implant [3]

In Fixation, the design of the prosthesis is very important to keep the long term attachment of the prosthesis to the bone. If the prosthesis shape design leads to high stresses in fixation areas, fatigue failure is quite likely to occur. Most of the

researches on prosthesis are devoted to static analysis [1-2, 4-5]. In the literature, in order to avoid revision of prosthesis, implants should be designed with enough strength to fatigue [6]. In this work the fatigue analysis of the failed prosthesis has been carried out. For the analysis three-dimensional model of the femur and implant were constructed. The analyses have been performed using ANSYS10.

II. CAD AND FINITE ELEMENT MODEL

A. CAD Model

The shape of the prosthesis has significant influence on the performance of the prosthesis. Prosthesis with smooth surfaces generally reduces stress concentrations and lead to high fatigue life. Stem shapes with sharp or non-smooth surfaces provide good bonding capability at the interface and prevent possible sliding at the interface. The level of stress concentration and tendency for fatigue depends on the sharpness of the stem surfaces

A cadaver bone was used for making femur geometry. Key points are used for constructing models in bottom-up approach. For obtaining co-ordinates of the key points in a 3-D working space, a fixture was arranged using V Blocks and sine bar on a surface plate to orient the bone in the required position. By taking these points along with the state of art modeling techniques in ANSYS, modeling the femur bone has been done. The failed knee prosthesis is shown in fig 1. The three main parts of the prosthesis are head, body and tail. The body is cylindrical in shape and it has length 120 mm and diameter 12 mm. The tail part is used to connect the prosthesis with femur bone by fixing it inside the bone cavity. The geometry of the prosthesis is also modeled using ANSYS which is shown in fig 2.

B. Finite Element Model

The three dimensional Finite element model required for analysis is created by discretizing the geometric model. The discretization was performed in ANSYS environment. The three dimensional solid geometry of the femur, modeled separately, which is shown in fig.3 exported as IGES file. This file is imported into the environment where the prosthesis model is opened. After the replacement of distal part of the femur bone model, the combined model of prosthesis with proximal femur part which is shown in fig.4 has been prepared for the analysis, assuming perfect bonding between bone and prosthesis.

*M Tech Student, College of Engineering Trivandrum, Kerala, India (e-mail: joyhjones@gmail.com)

**Professor, College of Engineering Trivandrum, Kerala, India.

To build the finite element model, femur, bone cement and implant were meshed using a higher order three dimensional element: SOLID187 which has quadratic displacement behavior and is well suited to modeling irregular meshes. The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The contact is considered as perfectly bonded. The finite element model consists of 32061 elements and 48660 nodes. The femur consisted of 13677 elements and the prosthesis consists of 18384 elements. The finite element model of the femur and femur along with prosthesis are shown in fig.3 and Fig.4

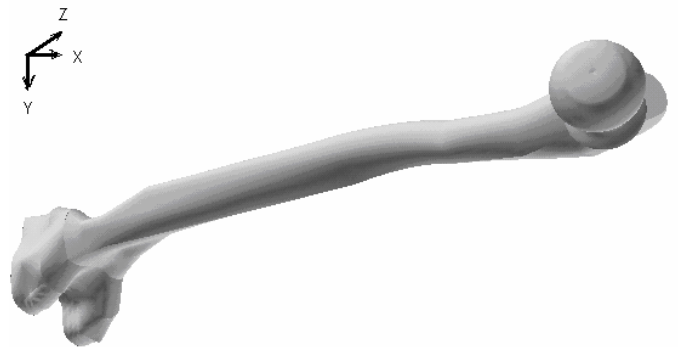


Fig.3. CAD model of femur bone



Fig.4. CAD model of prosthesis along with femur bone model



Fig.1. Prosthesis (a) Before failure, (b) After failure

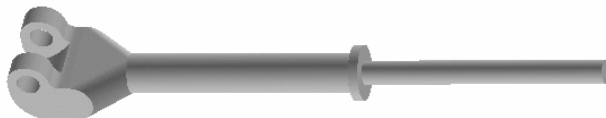


Fig.2. CAD model of the prosthesis

C. Material Properties

The materials used for the manufacturing the implant is 316L steel. Mechanical properties the materials used for analysis are shown in Table 1 and Table. 2.

D. Loading Conditions

The loading applied was taken from the work of Bergmann[8]; It is assumed that the person with prosthesis produce less stress compared to the normal person so a one by third of the gait cycle load is applied. The time history of the dynamic load is shown in fig.5 for a man weighing 70 kg .The load components during one cycle is considered as an event in the fatigue analysis

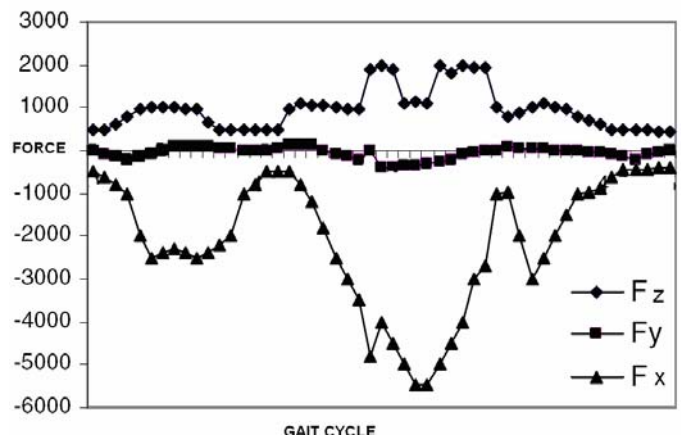


Fig.5. Loading conditions applied to hip

Table-1 Mechanical Properties of materials

Material	Tensile strength (N/mm ²)	Yield strength (N/mm ²)	Modulus of elasticity (N/mm ²)	Poisons ratio
Stainless steel (316L)	579	290	193000	0.3
PMMA (cement)		110	2000	0.33

Table-2 Mechanical Properties of Bone

Material	Modulus of Elasticity	Poisson's Ratio
Cortical Bone	17,000	.3
Trabecular Bone	1,000	.32

III. FATIGUE ANALYSIS

A good implant design should satisfy maximum or an infinite fatigue life. This can be ensured by physical testing or a fatigue analysis. Fatigue calculations for the implant are conducted for 316L stainless steel biocompatible material. The fatigue resistance of 316L Biocompatible material is very low compared to other biocompatible materials. The chart of fatigue properties of biomaterials is shown in fig. 6.

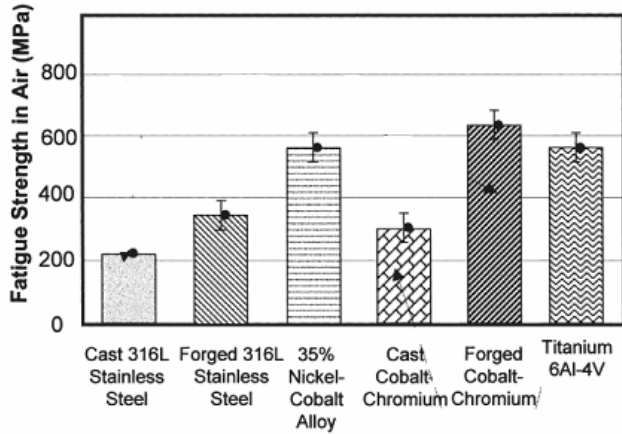


Fig. 6. Fatigue strength of some common implant alloys

Fatigue analyses can be performed according to Goodman Soderberg, Gerber theories.

Mean stress is defined as

$$\sigma_m = \frac{(\sigma_{\max} + \sigma_{\min})}{2}$$

and Alternating stress as

$$\sigma_a = \frac{(\sigma_{\max} - \sigma_{\min})}{2}$$

According to Goodman theory any combination of mean and alternating stress that lies below the Goodman line which is defined by

$$\left(\frac{\sigma_a}{S_e}\right) + \left(\frac{\sigma_m}{S_u}\right) = \frac{1}{N}$$

will have infinite life. Here N stands for safety factor, S_e for endurance limit and S_u for ultimate tensile strength of the material. Similarly for Soderberg approach

$$\left(\frac{\sigma_a}{S_e}\right) + \left(\frac{\sigma_m}{S_y}\right) = \frac{1}{N}$$

where S_y is the yield strength of the material. Unlike Goodman and Soderberg, Gerber approach is not linear in form and Gerber line is defined by the equation

$$\left(\frac{N\sigma_a}{S_e}\right) + \left(\frac{N\sigma_m}{S_u}\right)^2 = \frac{1}{N}$$

In ANSYS, fatigue results are given using cumulative fatigue usage.

$$\text{Cumulative fatigue usage} = \frac{N_u}{N_a}$$

Where N_u is the Number of cycles used and N_a is the number of cycles allowed. The number of cycles used is taken as the number of cycles for infinite life which is taken as 10^6 in this work

All Fatigue analyses are performed by utilizing the Von Mises stress obtained from the Static analyses performed. The stress life (S/N) approach is used for determining the fatigue life of the implant materials which is taken in this work. This approach is useful for the initial process of materials selection of implant materials that will be subjected to high cyclic loading conditions. The advantage of this approach is that it represents both initiation and propagation of cracks in the aggressive environment [10]. The S-N data given for analysis is given in fig 7.

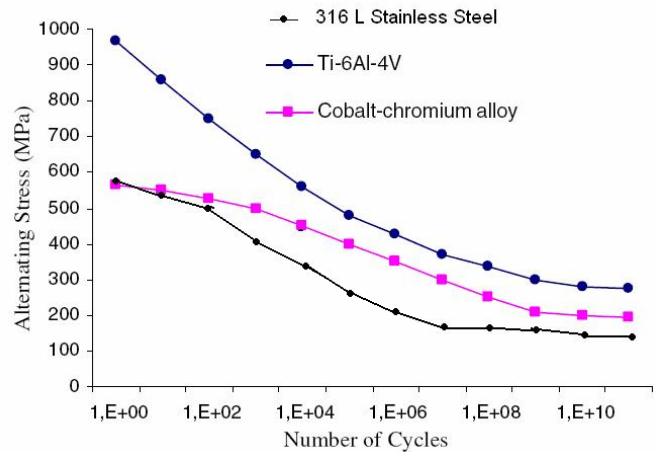


Fig. 7. S-N Curve of different biomaterials

In this work static analysis is performed before doing fatigue analysis. Using the result of static analysis few critical locations were selected and fatigue analyses are performed for those locations. The S-N Curve (fig. 8.) and stress concentration values should be given as input before performing analysis. Ten fatigue locations; one event and 21 loadings are selected and given to memory storage for performing fatigue analysis. From the gait cycle loading, 21 loading conditions were applied for the single event. The stresses for each node are stored from the result file after doing static analysis for 21 load conditions applied at node 23456 at the hip. The stresses stored in one of the critical locations; node 77 is shown in the fig. 7. The FEA program calculates all possible stress ranges and keeps track of their number of occurrences, using a technique commonly known as the “rainflow” range counting method which is used to convert the irregular load histories to blocks of constant amplitude cycles.

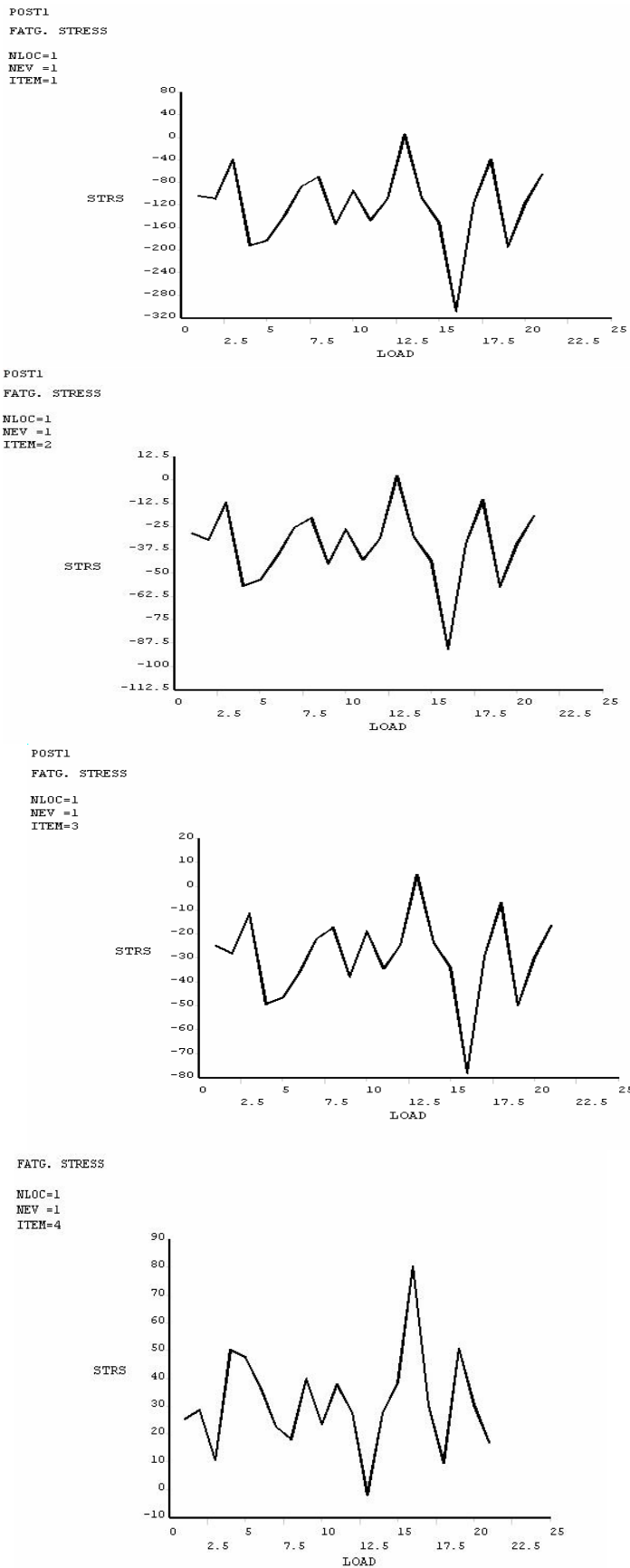


Fig. 8. Total Stress Plots (S_x, S_y, S_z, S_{xz}) in node 77

At a selected nodal location, a search is made throughout all of the events for the pair of loadings (stress vectors) that produces the most severe intensity range. The number of repetitions possible for this range is recorded and the remaining number of repetitions for the events containing these loadings is decreased accordingly. The stress ranges having significance will be stored and the remaining occurrences of smaller stress ranges belonging to that event will subsequently be ignored. This process continues until all ranges and numbers of occurrences have been considered. The sequence of loading has a major influence on the fatigue life, especially in the case of notched components

IV. RESULTS AND DISCUSSIONS

Of the ten critical locations (fig. 9 and fig. 10), the alternating stresses and cumulative fatigue usage factor are given in Table.2 and it is found maximum at third location that is at node 1565. The alternating stress produced at this node is 449.75 N/mm^2 . Similarly at locations 1, 2, 4, 5, 6, 7 the alternating stress exceeds the safety limit before reaching endurance limit. The allowed cycle at third location is only 235 which mean the failure would happen within short period of usage. It is found that the failure of newly designed Indian Prosthesis was also happened at tail-body interface. Fatigue analysis predicts failure within short period of usage at this location.

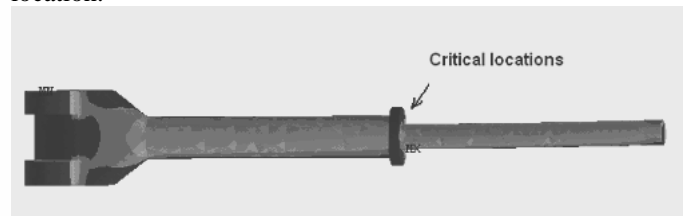


Fig. 9. Critical locations in the FE results.

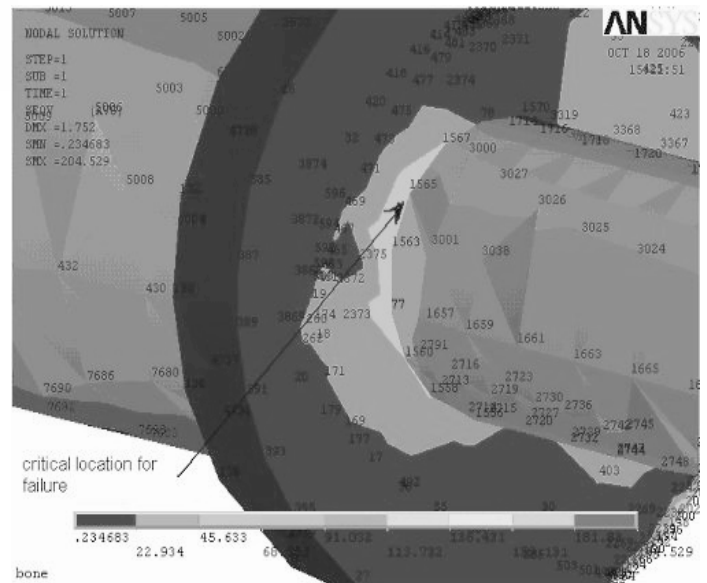


Fig. 10. Critical location of failure

Table 3. Cumulative Fatigue Usage at critical locations

Location number	Node number	Alternating Stress	Cumulative fatigue usage for infinite life cycle
1	77	349.92	64.15
2	1563	394.92	78.88
3	1565	449.75	4251.427
4	3001	219.23	6.85
5	79	277.73	20.33
6	1574	260.71	15.93
7	1577	352.23	50.77
8	3387	67.547	0.01
9	3722	124.39	0.01
10	1771	91.231	0.01

[10] Teoh SH, "Fatigue of Biomaterials : a review.Int. journal of fatigue 2000;22;825-37.

The location of maximum alternating stress due to fluctuating load is different from maximum stress obtained after a static analysis. This shows that the point which is predicted to be safe against static conditions may fail under dynamic repetitive loading conditions.

V. CONCLUSION

Fatigue analysis using FEA has been performed. It has been found that the fatigue damage happened at tail-body interface during analysis which substantiates the failure of the prosthesis in actual usage. The Values obtained for Cumulative fatigue usage for infinite life cycle predicts that few critical locations are not able to withstand against alternating stresses within endurance limit. This predicts that the prosthesis is unsafe against fatigue for normal gait cycle loading conditions for infinite life. Further modification of prosthesis can be performed accurately incorporating the present analysis.

REFERENCES

- [1] RameshChand "Stress Analysis of Human Femur Bone prosthesis and design modification" ,Thesis,College of Engineering University of Kerala, 2005.
- [2] RameshChand, B Anil, "Stress Analysis of Human Femur Bone prosthesis and design modification" 6thNational Conference on Technological trends NCTT, 2005 pp. 62-66.
- [3] A Zafer Zenalp, Oguz Kayabasi,Hasan Kurtaran "Static, dynamic and fatigue behaviour of newly designed stem shapes for hip prosthesis using finite element analysis," Jl.of Materials and Design, 2006; Article in press
- [4] S.K Senapathi, S Pal, "Finite Element Analysis:An Effective tool for prostheses design",Trends Biomater. Artif. Organs.; 2004 ;17; pp141-148
- [5] Chaodi Li, Cristopher Granger, "Failure analysis of composite femoral components for hip arthroplasty",Journal of Rehabilitation research and development,2003; 40, pp131-146.
- [6] M Dolare and J.M.Garcia, "New Trends in computational methods applied to design of prosthesis and implants" 2003, Aaragon Institute of Engineering research, Spain.
- [7] Harrigan T P,Harris WH "A Three dimensional non-linear finite element study of the effect of cement-prosthesis debonding in cemented femoral hip components," Jl.of Biomechanics, 1991;24; pp. 1047-1058.
- [8] Bergman.G.,Graichen.F and Rohlman A,"Hip loading during walking and running, measured in two patients",Jl. of Biomechanics.,26;8;pp.969-990.
- [9] ANSYS, Ansys Theory Ref. Manual, Release 10 Ansys Inc.2005