Fatigue Life Prediction for Hip Stem using Finite Element Analysis

Sagar Karande, PG student, Department of Mechanical Engineering, VPCOE, SPPU, Pune.
S. V. Shelge, Assistant Professor, Department of Mechanical Engineering, VPCOE, SPPU, Pune

Abstract— **Hip joint replacement surgeries have increasing trend due to failure of femur bone in older as well as younger patients. Age, bone diseases and accidents are the roots for the failure of bone which leads to hip replacement surgery. Artificial hip implants made from different materials are used in the joint replacement surgery. Many literatures reported premature failure of artificial implant due to fatigue failure as load experienced by the implant is of fluctuating in nature. Hence, prediction of fatigue life of implant has greater importance and will create a choice for patient in selection of proper implant according to his physical need. In this study, fatigue life of implant is predicted using FEA and its validation through physical testing is completed. Orientation and loading of implant is done as per guidelines given in ISO 7206-4 norms for both FEA and physical testing. It has been observed that experimental and numerical results are in the acceptable limit hence fatigue behavior prediction using FEA can be used for design alteration.**

Key words— **Fatigue life, FEA, Hip implant, Static analysis.**

I. Introduction

HIP joint is most important joint in human skeleton and it is one of the prime weight bearing joints. Human bones and joints are constantly working under varying forces. The variation in boundary and human body conditions further increases the magnitude of the problem due to which, the joint replacement surgeries are increasingly carried out. In these surgeries, human joint is replaced by an artificial implant, made up of bio-materials like stainless steel, titanium, composites and others.

It was alleged that joint replacement surgery is to be performed only on older patients with very limited physical movements, but in modern era, even younger patients are being treated with the joint replacement surgeries. It will be wrong to expect from younger patients to restrict their movements to merely walking for a few times in a day. This paves the way for articulating a validated finite element model of implant, with which, its life can be predicted for various activities of patient and loading conditions arising from those activities. If such practice of predicting fatigue life data is amended to all types of implants available in the market, it would be easier for the surgeons, and the patients to select a suitable implant as per the patient's physical needs. The surgeons will also come to know about the credentials of the product they are going to use, with the availability of the fatigue life data.

Total hip arthroplasty (THA) is currently the international standard of care for progressive hip joint disease [1]. The worldwide demand for THA has risen substantially over the past decade. In 2010, it was observed that 3, 26,100 total hip replacements were performed among inpatients of all ages in United States [2]. Newsletter of the Indian Society of Hip and Knee Surgeons shows that there is increase in number hip replacement from 194 in 2006 to 3604 up to 2012 in India. The current registry of Indian Society of Hip and Knee Surgeons shows that the number of hip replacement surgeries to 8266 [3].

There are many causes why a total hip replacement may fail. Breaking of implants inside the bone is one of the worst case failures [4]. Recent premature failures of total hip replacement have raised concerns about their load bearing capacity, safety, reliability, and survival rates. Specifically, femoral stem fracture is one of the most serious problem of total hip replacement (THR) and effects in greater morbidity and higher cost of revision hip surgery. Hip stem fracture incidence varies from 0.23% to 10.7% depending on geometric design and prosthetic materials. The late failure of implant illustrates the importance of proper long-term testing of implants with regards to cyclic loading, as patients undergoing total joint replacement are increasingly younger and more active, putting implants at greater risk of long-term fatigue failure [5]. A study was conducted on 127 patients who had hip replacement revision surgery to determine the impact of each cause of revision surgery in total hip arthroplasty. The results shows that most common cause of rescue was aseptic loosening in 38 (30%) and instability in 30 (24%) followed by fracture in 26 (20%) [6].

This paper is based on the analysis of a hip stem under static load and fatigue life prediction of the same. In the present study, collared bipolar hip stems purchased from two companies are used for analysis. The hip stem is made up with a typical composition of '18Cr10Ni2.5MO'. The loading and orientation of the implant is done as per the ISO 7206-4 norms.

II. Literature Review

Several researchers find out the fatigue life for different types of hip implants with different materials using finite element method and also through experimental testing. In the literature Fatigue life of partial hip prosthesis has been predicted by using FEA for an activity of brisk walking. The comparison of results from static analysis for FEA and physical testing shows that results are fairly matching to each other [7]. A comparative study on the fatigue behavior of eight different hip stem demonstrate that analyzing hip stems with the finite element method (FEM) can be applied with confidence to support standard fatigue testing and used as an alternative method [8]. Bergmann et al, [9] find out mechanical loading of the hip. The average patients loaded their hip joints with 238% of body weight (BW) when walking at speed about 4 km/h and with slightly less when standing on one leg. When going upstairs joint contact force is 251% BW and when going downstairs it is 260% BW. On average it is 23% larger when going upstairs than during normal level walking. They suggested that implants should mainly be tested with loading conditions that mimic walking and stair climbing. Syed zammer et al, [10] carried out analysis of hip joint model using finite element software

ANSYS. Stress distribution obtained from result of static analysis and material properties of fabricated UHMWPE polymer matrix composite specimens were used to estimate fatigue life of hip joint model. They use linear palmgren damage rule to calculate factor of safety, which shows that the component is safe under design. C.M. Styles et al, [11] as the new designs and material combinations including coatings may introduce fatigue problems, makes an initial attempt to develop accelerated fatigue testing procedures to enhance the methodology of hip implant lifetime prediction. They conducted a test on a test piece in physiological solution such as Ringers at 37 degree Celsius using superimposed block overloads. David Bennett et al. [12] performed a finite element analysis (FEA) on six hip stem designs. The hip implant designs were then analyzed at forces ranging from 2.5 to 7 KN. These forces were selected because a typical gait cycle generates forces up to 6–7 times the body weight in the hip joint. The FEA results were compared for various stem designs assuming a rectangular cross-section. The design objective for a hip stem is to have a low stress, displacement, and wear at a very high fatigue life. The analysis performed in this paper may aid also in the fatigue testing of implants and/or base the designs for tested materials at a suitable fatigue life. Subsequently, the stems that had the highest stress and displacement models were then optimized for a lower stress and displacement combination. The cross-section that comprised a circle in the medial end and a square at the lateral end was found to provide suitable design characteristics.

III. CAD Model And Finite Element Analysis

The shape and size of the hip stem has significant influence on the performance of the stem. Hip stem with smooth surfaces generally reduces stress concentrations and improves the fatigue life. Stem shapes with non-smooth surfaces deliver good bonding at the interface and prevent possible sliding at the interface. The level of stress concentration and tendency for fatigue depends on the acuity of the stem surfaces. A three dimensional CAD model of the implant is prepared by using 'CREO parametric-3.0' software. The model is prepared by using actual dimensions measured from the implant. Fig. 1 shows the orientation of the hip stem as per the ISO Norms. The ISO 7206-4 is guide to test endurance properties and performance of stemmed femoral components [13].

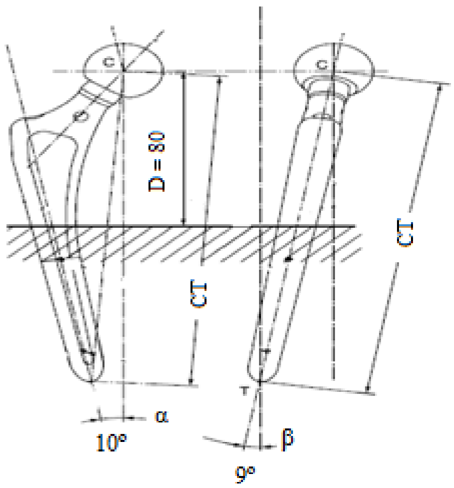

Fig. 1. Orientation of the stem [13]

A. *Material properties*

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

TABLE I
MECHANICAL PROPERTIES OF MATERIALS

Material	Stainless steel (316L)	Epoxy Resin
Tensile Strength (MPa)	552	-
Yield Strength (MPa)	207	-
Modulus of Elasticity (GPa)	193	6
Poisson's Ratio	0.31	0.35

B. *Finite element analysis*

The STEP file of three dimensional CAD model is imported to ANSYS Workbench 14.5 software to prepare a finite element model. The model is prepared to find out stresses in the implant when oriented and loaded as per ISO 7206-4 test protocol. Fig. 2 shows the FEA model of the hip implant with loading mechanism. The block number one acts as a plunger of the machine, imparting the force to the top of the stem head. The modulus of elasticity of the plunger is similar to that of the stem material to avoid dispersion of harder material into softer material. The plunger has only vertically downward motion with respect to the frame. The vertical load is applied on the top of the stem head. The block number two, which is a stem holding frame, characterizes the solidified epoxy resin. It surrounds the stem, and grips it from outside. The modulus of elasticity of the hip stem holding frame at the stem surface is considered same as that of epoxy resin, holding the stem. The reference value of 6000 N/mm2, stated in ISO 7206 is considered for this determination.

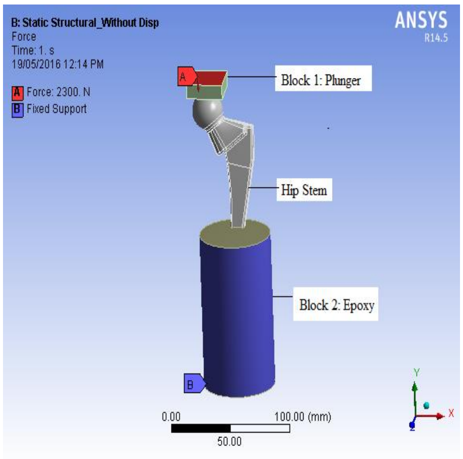

Fig. 2. FEA model of the hip implant

The ISO 7206 gives test procedure to find out whether the implant being tested satisfies the norms to have sufficient life under constant amplitude loads. The FEA model of implant needs to be tilted in frontal and lateral planes to orient it as per the ISO 7206 norms. For meshing of this model, auto mesh is adopted. Solid tetrahedron elements with mid side nodes are selected, as this type is found to be more suitable for irregular and curved nature of the model. Meshing element size is selected as 2 mm. The Fig. 3 shows the meshing of implant. A static vertically downward load of 2300 N is applied on the top surface of the femur head, after the meshing is completed. The solution is run in FEA solver after application of the load.

C. *Results of finite element analysis*

In the finite element analysis of implant model, two locations are found to be critical in terms of stress developed. The inner and outer surfaces of the stem where it touches the surface of holding epoxy resin are two such heavily stressed areas since the bending moment produced by the applied load is more at these points. These stresses are not considered for the calculation of fatigue life as the implant is completely inserted in human bone. The stresses 10 mm above epoxy surface is considered for fatigue life calculation. The inner and outer surfaces of neck of the stem are the other two areas where failure is reported in literature. These locations are named as 'Surface In', 'Surface Out', 'Neck In', and 'Neck Out' as shown in Fig. 3.

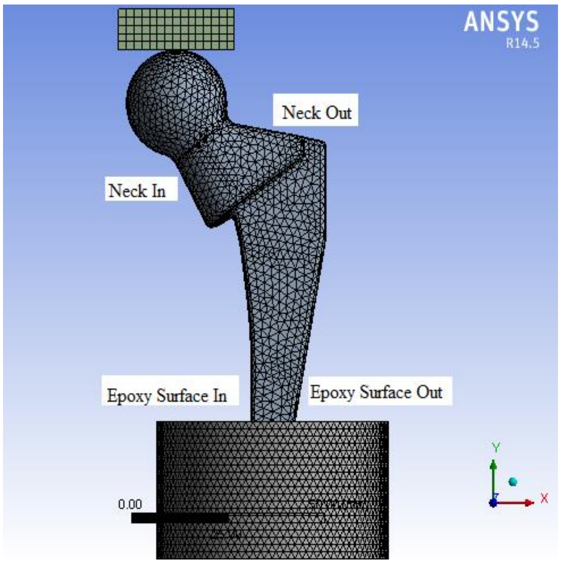

Fig.3. Mesh model in ANSYS

The above Fig.3 shows the meshing of model here tetrahedral solid 187 element type is used with average quality of element as 0.844. The stresses developed under a static load of 2300 at four different random locations are also shown in fig. 4. Here Surface-In portions is experiencing compression while the surface-out portions of stem experiencing tension.

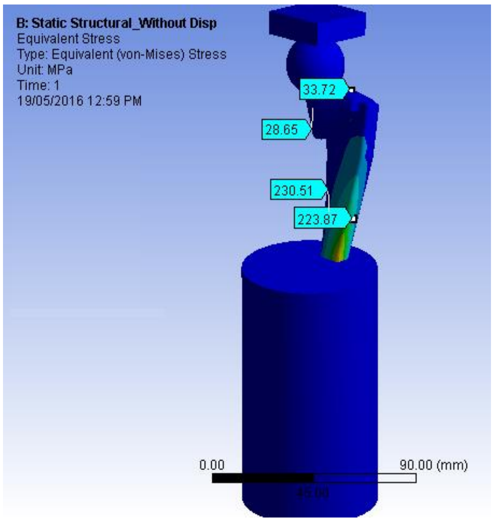

Fig. 4. Stress at four different locations.

The fatigue strength S_e' of the implant material is 276 N/mm2. However, the endurance strength of actual implant is less than that of standard test specimen due to the effect of size, surface finish, reliability, temperature, and load. The maximum stress developed at the four random locations obtained from FEA is 230.51 MPa. Hence the stress generated at four different locations is less than materials endurance limit, the implant has infinite fatigue life at that particular locations. The stress developed in the implant under static loading will be same as those developed under repeated loading.

IV. Physical Testing

The testing of the implant is done under static conditions, by orienting the implant as per ISO 7206 protocol. The stem of implant is held in epoxy resin with stem axis making 10º angles with vertical in frontal plane and 9º in the lateral plane. A fixture is manufactured using wooden block to hold the implant as per the orientation given in ISO 7206 norms. The test set up used for physical testing of the hip implant consists of three parts namely strain gauge set up, data acquisition system and loading frame. The strain gauges work as a sensor to capture the data of strain developed at marked locations of the implant during the loading process. The strain gauges are glued to the surface at the marked location of the specimen. Four uniaxial foil type strain gauges are used to form quarter strain gauge bridges. A high speed multi-channel data acquisition system with simultaneous sampling and hold capability to record the strains during the tests is used. This data acquisition system reads and represents the data generated by strain gauges in terms of strain. A Universal Testing Machine (UTM), with a capacity of 40kN is used as loading frame, to apply the load as per the ISO 7206-4 standards. Fig. 4 shows the data acquisition system and the implant with strain gauges loaded under the loading frame.

A. *Results of physical testing*

The compressive stress is developed at Neck In and Surface In locations of the implant. The tensile stress is developed at Neck Out and Surface Out locations of the implant. Fig. 5 shows a comparative chart of stresses developed at four different marked locations with respect to time.

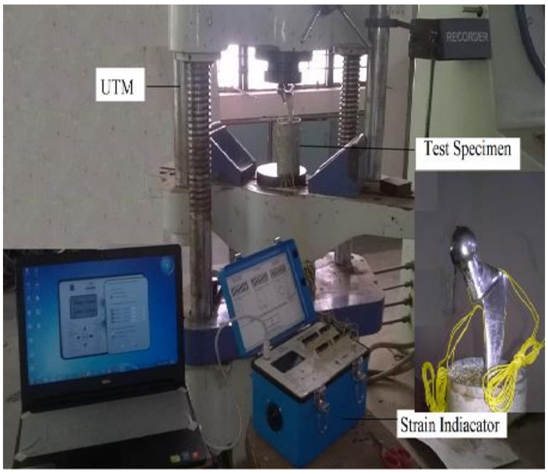

Fig. 5. Data acquisition system

B. *Comparison of results*

The comparison of results of FEA and physical testing is shown in Table II. It can be concluded from the comparison, that the results are fairly matching to each other. The stresses developed at all four random locations of implant during physical testing are within 10 percent variation from the values obtained in FEA. This is an acceptable limit of variation for FEA. It can be safely concluded that the FEA model is validated, because the results of FEA and physical testing are matching with each other within the acceptable limits. The comparison of the measured strain at four different locations during physical testing at particular loads with strain obtained in FEA at the same locations is shown in Table III

TABLE II
COMPARISON OF RESULTS (Stress)

Sr. no	Position	Magnitude of stress (MPa)		Nature of stress
		FEA	Physical testing	
1	Neck in	28.65	31.59	Compressive
2	Neck out	33.72	39.75	Tensile
3	Epoxy surface in	230.51	239.127	Compressive
4	Epoxy surface out	223.87	222.915	Tensile

TABLE III
COMPARISON OF RESULTS (Strain)

Applied force (N)	Measuring method	Strain (με) values at 4 different points			
		1	2	3	4
2100	Strain gauge	154	194	1189	1110
	FEA	139	177	1172	1081
2200	Strain gauge	157	198	1217	1126
	FEA	145	186	1189	1107
2300	Strain gauge	162	205	1239	1155
	FEA	148	174	1194	1130
2400	Strain gauge	170	214	1261	1202
	FEA	153	179	1234	1167
2500	Strain gauge	175	220	1287	1233
	FEA	161	192	1257	1195

V. FATIGUE LIFE CALCULATION

Fatigue is the progressive and localized structural damage that occurs when a material is subjected to cyclic loading. Fatigue life is the number of applied repeated stress cycles a material can endure before failure. The load considered for prediction of fatigue life is equal to 2300 N as specified in the ISO 7206-4 norms. Here fatigue life of implant is predicted from the induced stress in the implant when static load of 2300 N is applied on its head. For determining alternating stress from the induced stress, equation (1) is used from the ASME section viii, division 2.

$$S_{alt,k} = \frac{K_f . K_{e,k} . \Delta S_{p,k}}{2} \quad (1)$$

Where, $S_{alt,\,k}$ is alternating stress, $\Delta S_{p,\,k}$ is induced stress. K_f is fatigue strength reduction factor, $K_{e,\,k}$ is fatigue penalty factor. From above equation alternating stress is calculated by assuming fatigue strength reduction factor and fatigue penalty factor is equal to 1. Now the fatigue life of the component is calculated by using equation (2),

$$N = 10^X \quad (2)$$

Where,

$$X = \frac{C_1+C_3Y+C_5Y^2+C_7Y^3+C_9Y^4+C_{11}Y^5}{1+C_2Y+C_4Y^2+C_6Y^3+C_8Y^4+C_{10}Y^5}$$

$$Y = \left(\frac{S_a}{C_{us}}\right) \cdot \left(\frac{E_{PC}}{E_T}\right)$$

The induced stress in the implant 10mm above embedding surface is 230.51 MPa. Here fatigue strength reduction factor and fatigue penalty factor is considered as unit. From equation (1) & (2) alternating stress and fatigue life of implant is calculated respectively.

VI. RESULTS

The von misses stresses developed at the four random locations are evaluated through FEA. The locations chosen are same as those selected during validation phase of the model. The comparison of results is shown in the fig. 5. This shows that results are fairly matching with each other.

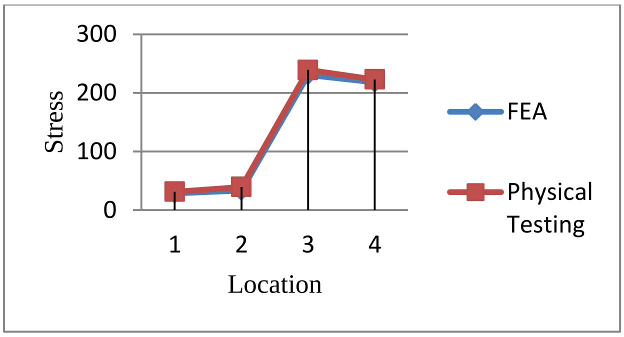

Fig. 6. Comparison of FEA and physical testing results

The pattern of the load and stresses shows that stresses are mapping the pattern of load. The stresses developed vary in the same pattern with which the load varies. If the load is increased or decreased with a certain percent, the stress pattern will also exhibit the same change in its magnitudes. The maximum induced stress under 2300N is 230.51 MPa at the surface In location. Hence for the calculation of fatigue life of the implant only maximum stress is considered. The predicted fatigue life of the implant from the equation mentioned is more than $5x10^6$ cycles.

VII. CONCLUSION

This paper presents a procedure for predicting the fatigue life of hip implant under static loading. The FEA modeling and physical testing is carried out for this purpose as per ISO 7206-4 norms. Here the stresses developed at four random location is less than the fatigue strength of the material hence the implant has infinite life at that locations. The comparative results are within the acceptable limits. The current study has also demonstrated that analyzing hip stems with the finite element method (FEM) can be applied with confidence to support standard fatigue testing and used as an alternative.

ACKNOWLEDGEMENT

Author would like to thank Royal Medimed Pvt. Ltd. Company for providing the implant. Author also would like to thank Prof. A. H. Kolekar, Dr. Ravi Sarangapani and Dr. B. Ravi for their valuable guidance.

REFERENCES

[1] Kurtz et al, “International Survey of Primary and Revision Total Hip Replacement” 56th Annual Meeting of the Orthopaedic Research Society.
[2] Monica L. Wolford et al, “Hospitalization for Total Hip Replacement among Inpatients Aged 45 and Over: United States, 2000–2010”, NCHS Data, No. 186, February 2015.

[3] Dr. Ashok Shyam, “Newsletter of the Indian Society of Hip & Knee Surgeons”, ISHKS, Nov-Dec 2012, Vol.1.

[4] Monika Oravcova et al, “Premature Failure of A Femoral Hip Replacement”, Materials Today: Proceedings 3 (2016) 1114 – 1117

[5] Michel Nganbe et al, “Unusual Fatigue Failure of Cobalt Chrome Alloy Cementless Femoral Stem: Implant Retrieval and Biomechanical Analysis”, Journal of Medical and Biological Engineering, 426 34(5): 426-430.

[6] D. Capon-Garcia et al, “Causes for Revision Surgery in Total Hip Replacement: A Retrospective Epidemiological Analysis”, Rev Esp. Cir Ortop Traumatol. 2016;60(3):160-166

[7] K.V. Datar et al. “Fatigue Life Analysis of Partial Hip Endoprosthesis for an Activity of Brisk Walking”, IJERA, Vol. 2, Issue 6, November-December 2012, pp.994-1001.

[8] Mahmut pekedis et al. “Comparison of Fatigue Behavior of Eight Different Hip Stems: a Numerical and Experimental Study”, J. Biomedical Science and Engineering, 2011, 4, 643-650.

[9] Bergmann et al. “Hip Contact Forces and Gait Patterns from Routine Activities”, Journal of Biomechanics 34 (2001) 859–871.

[10] Syed zammer et al. “Fatigue Life Estimation of Artificial Hip Joint Model Using Finite Element Method”, Materials Today: Proceedings 2 (2015) 2137 – 2145.

[11] C.M. Styles et al. “Development of fatigue lifetime predictive test methods for hip implants”, Biomaterials 19 (1998) 1057-1065.

[12] David Bennett et al. “Finite Element Analysis of Hip Stem Designs”, Materials and Design 29 (2008) 45–60

[13] ISO 7206-4, Implants for Surgery-Partial and Total Hip Joint Prostheses-Part 4: Determination of Endurance Properties and Performance of Stemmed Femoral Components