ISSN 2395-1621

"Mechanical Evaluation of Hip Joint Implant Device by Using FEA"

[#1]Pawar A. J., [#2]Biradar N. S.

[1]pawar.anup.j@gmail.com
[2]nsbiradar123@gmail.com

[#1&2]Department of Mechanical Engineering, JSPM's Imperial College of Engineering & Research, Wagholi, Pune,India.
Savitribai Phule Pune University

ABSTRACT

The science of biomechanics is deals with human body skeleton and its body implants. The human body skeleton system has the number of joints. In case of fractures, these joints are fixed with different fixation methods by using metallic angled devices. Biomechanical point of view human hip joints are complicated in structures and also capable of functioning under the critical conditions. When a metaphysis or epiphysis area of the long bones is fractured, it is a challenge to develop specific angle devices that can be used in hip joint.

The aim of study to find mechanical properties of internal fixation angled devices, which is used in the human hip implant. Mechanical behavior of angle devices is carried out by applying compressive loads and boundary conditions using FEA technique.

The results of the angled devices obtained from FEA analysis are helpful for surgeon and orthopedic physician to choose specific angle device for particular hip implant to serve the specific life.

Keywords— **Metallic Angled devices, Metaphysis, Epiphysis, FEA technique.**

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

I. Introduction

Science of biomechanics is concerned with the action of internal or external forces, on the living body. It also includes bioengineering, research and analysis of the mechanism of living organisms and the application of engineering principles to from a biological system.

Biomechanics is also applied in studying human muscular skeletal systems. In few years, research applied force to study human joint reaction forces. Biomechanics is widely used in orthopedic industry to design orthopedic implants for human hip joints, external fixations and medical purposes. The biomechanical factors determine where a bone will fracture. It includes the loads applied and the mechanical properties of bone and bone tissue. The loads on a normal bone result in fracture and are typically extremes.

Human bones are strong and can resist tremendous bending and compression forces without breaking. Muscles pull on bones make them to move, but movement would not be possible without joints between the bones. Humans can resemble tissues, but it not for the joints between bones that allow bones to move once the muscles are provided the pull in nature. Machine parts most likely to wear out are those that rubbing together and they require the maintenance. Movable joints are places in the body, the bones slide on each other. We can little attention on them. Human joints are self- maintaining, but damage to a joint can make movement very difficult.

Hip joints are complex in structures also capable of functioning under critical conditions, and it is a challenge for doctors, scientists and engineer to develop specific implants that can be used in a hip joint to serve a specific purpose for orthopedic applications.

II. OBJECTIVE AND METHODOLOGY OF PRESENT STUDY

The aim of present study focuses on the roles of mechanical bone strength, stress conditions of hip joint and strength of angled devices to be implant on the life span of hip implants.

1) To import and edit the geometry

2) Meshing will be done in ANSYS 15.0 itself to generate elements around 1 lac.
3) Meshing of the model by Tetrahedral Solid187 elements in ANSYS 15.0.
4) Static structural and linear buckling analysis is to be carried out on a given Hip Joint Implant Device.

III. FINITE ELEMENT ANALYSIS

The FEA is numerical analysis technique for getting approximate solutions to differing kinds of engineering issues. In engineering things we are able to notice that it's necessary to get approximate resolutions to drawback instead of actual closed kind solution.

It is impractical to get analytical mathematical solutions for several engineering issues. An analytical resolution may be a mathematical expression that provides the values of the required unknown amount at any location within the body, as consequence it's valid for infinite range of location within the body. For issues involving complicated material properties and boundary conditions engineer resorts to numerical ways that offer approximate, however acceptable solutions.

The finite component technique has become a strong tool for the numerical solutions of a large vary of engineering issues. It has been developed at the same time with the increasing use of the high-speed electronic digital computers and with the growing stress on numerical strategies for engineering analysis. This technique started as a generalization of the structural arrange to some problems with elastic time disadvantage, started in terms of varied equations. A static structural analysis determines the displacements, stresses, strains, and forces in elements caused by forces that don't induce important inertia and damping effects. Steady loading and response conditions i.e. the loads and the structural response is assumed. A static structural load can be performed using the ANSYS. The types of loading that can be applied in a static analysis is related with ASTM F384 standard and test methods describes methods for single cycle bend testing for determining intrinsic, structural properties of hip joint implant device. The test method measures compression bending stiffness and compression bending strength of implant device for hip joint. This test method is providing a means to mechanically characterize different angled device designs. It is not the intention of this test method to define levels of performance for hip joint implant device, as these characteristics are driven by patient-specific clinical requirements. This test method is designed to provide flexibility in the testing configuration so that a range of clinical failure modes for the hip joint implant device (for example, side plate, lag screw, and barrel fractures) can be evaluated. The values stated in SI units are to be regarded as standard.

Linear buckling analysis predicts the theoretical buckling strength of a perfect elastic structure. Linear buckling analysis often yields quick but non-conservative results. A more accurate approach to predicting instability is to perform a nonlinear buckling analysis. A step by step increasing load is applied during this analysis to seek the load level at that structure becomes unstable. Using the nonlinear technique, model will show features like initial imperfections, gaps, plastic behavior and large-deflection response. Results calculated by the linear buckling analysis and buckling load factors that scale the loads applied in the static structural analysis. The buckling load factor is to be applied to all the loads used in the static analysis. A structure can have infinitely many buckling load factors. Each load factor is associated with a different instability pattern. Non zero constraint apply in the static analysis also the load factors calculated in the buckling analysis. However, the buckling mode shape associated with this load will show the constraint to have zero value. Buckling mode shape displays are helpful in understanding how a part or an assembly deforms when buckling, but do not represent actual displacements.

A) Finite Element Analysis Steps

1. Modeling -

Given solid DHS with breadth of 19 mm file in x_t format has imported in ANSYS workbench. Geometry has threaded portions internally and externally, and threaded portions are not feasible in ANSYS environment. So, we had removed these threaded portions and extruded the given geometry. Also there is contact gap between lag screw and barrel. And we have removed this gap by increasing the diameter of lag screw at the contact between the lag screw and barrel. For the purpose of giving support and applying load we have imprinted the faces of lag screw and side plate.

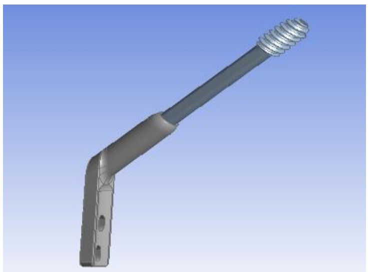

Fig 1 Original geometry

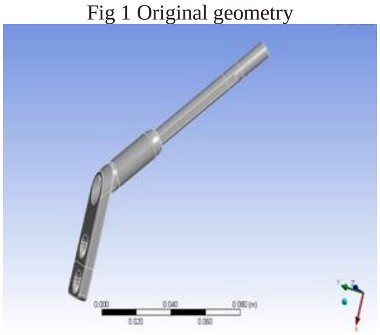

Fig 2 Edited geometry

2. Meshing –

After the modeling next step is generation of Finite Element Mesh. SOLID elements are used for the hip implant meshing. After generation of coarse mesh, it is refined as per the geometry and critical sections of the model. Mesh transition happens once refined mesh interfaces with coarse mesh. It connects different types of elements.

SOLID187 part could be a higher order 3-D, 10-node part. SOLID187 features a quadratic displacement behavior and is well matched to modeling irregular meshes.

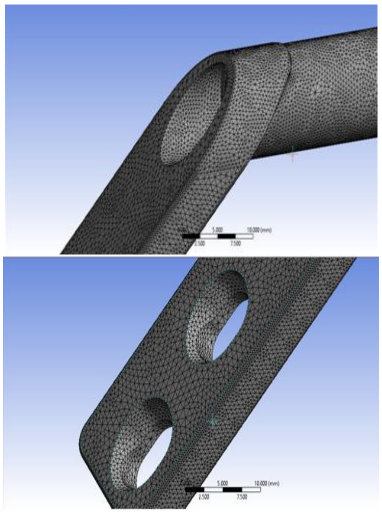

Fig 3 Meshed geometry

3. Boundary Conditions-

Load applied at lag screw at 0.8 percent offset criteria, in the fig 4 point A shows load. The fixed support at side plate is applied as shown in fig at point B. The elastic support applied inside the holes shown in point C and following load steps are applied-

Table I
LOAD CASE TABLE

Load Case	Multiplier	Force
LC1	X	800N
LC2	1.5X	1200N
LC3	2X	1600N
LC4	2.5X	2000N
LC5	3X	2400N
LC6	3.5X	2800N

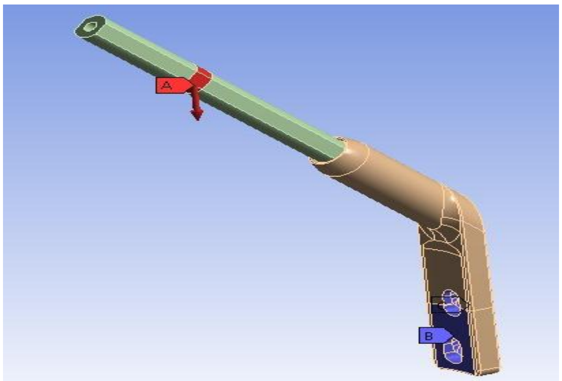

Fig 4 Boundary conditions

The above geometry is solved for static structural analysis and linear buckling analysis.

IV. RESULT

1. Results of static structural analysis (DHS barrel plate 19 mm width). The Fig. 5 and Fig. 6 give the local stress and deformation plot for load step of 800N.

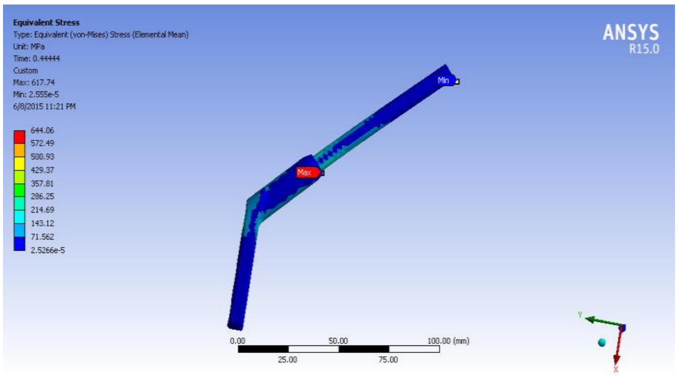

Fig 5 Local Stress Plot

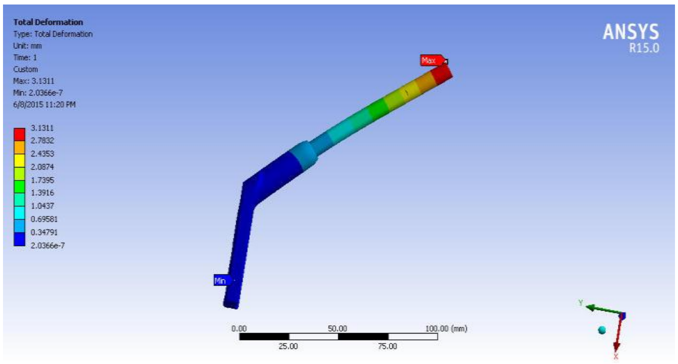

Fig 6 Local Deformation Plot

2. Results of linear buckling analysis (DHS plate with 19 mm width). The Fig.7 and Fig. 8 gives the equivalent stress and total deformation plot for load step of 800N.

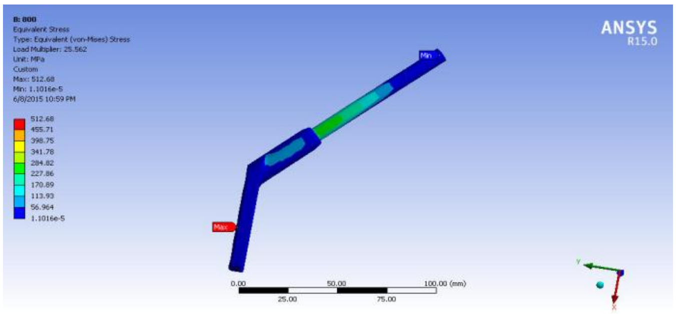

Fig 7 Equivalent Stress Plot

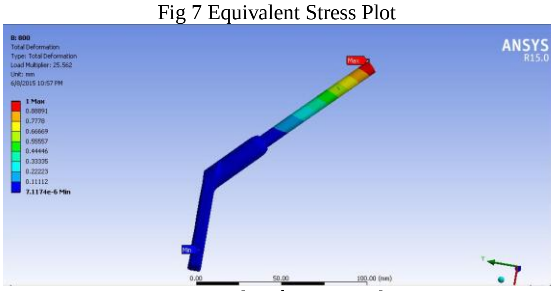

Fig 8 Total Deformation Plot

Similarly, we get all results which are tabulated in the Table II

Table II
RESULT FOR DHS PLATE WITH 19 MM WIDTH

Load Case	Load multiplier	Local Deformation (mm)	Local Stress (MPa)
B1 (F= 800N)	25.56	3.13	644.06
	32.29		
	89.04		
	144.83		
	149.45		
	186.18		
B2 (F=1200N)	17.04	4.69	1016.2
	21.53		
	59.35		
	96.53		
	99.56		
	124.08		
B3 (F=1600N)	12.77	6.261	1349.1
	16.14		
	44.5		
	72.34		
	74.63		
	92.89		
B4 (F=2000N)	10.22	7.831	1674.8
	12.91		
	35.63		
	57.94		
	59.65		
	74.47		
B5 (F=2400N)	8.52	9.394	2066.9
	10.76		
	29.71		
	48.31		
	49.62		
	62.06		
B6 (F=2800N)	7.30	10.954	2370.3
	9.22		
	25.43		
	41.37		
	42.66		
	53.17		

V. CONCLUSION

- As per the need of project, given geometry was edited in ANSYS Design Modeler and Meshing is done, Using Solid 187 Element which is suitable for static structural and linear buckling Analysis of 3D geometry.
- In the Analysis lag screw is totally extended for 115mm and the analysis was meant for the worst case.
- Finite Element model was built according to rules and regulations ASTM F384 Standards.
- Around 65000 elements were generated.
- Load steps considered were 800N, 1200N, 1600N, 2000N, 2400N, 2800N.
- Static Structural and Linear Buckling analysis were done in order to find out maximum equivalent stresses generated and deformation under certain loading conditions.
- In buckling analysis, buckling modes and Load Multipliers are calculated.

These results are helpful for orthopedic designer for optimize the material and for surgeon or orthopedic physician to selection of specific implant device for particular hip implant to serve the specific life.

ACKNOWLEDGEMENT

This study is an outcome of assorted efforts by us in collating and distinctive the sources of information and knowledge. fruits. We find no words to specific our feeling to our guide academic. Biradar N. S. who not solely suggested and guided us throughout study however additionally answered all our queries regarding gathering of knowledge, correct structuring of the report, and its improvement.. I additionally specific my sincere thanks to Mr.. Rout Sundaram R. , Mr. farzin B., Mr. Bandgar Vivekanand B. of Extencore solutions Pvt. Ltd. Pune for uplifting, Guiding and there Unconditional support.

REFRENCES

[1] Philip Tate,Seeley's Principal of Anotomy & Physilogy.pp.181-184
[2] Hamid Reza Seyyed Hosseinzadeh,Mehrnoush Hassas Yeganeh,Hamed Reza Seyyed Hosseinzadeh, Plate Fixation in Orthopedics (Basic Concepts and Clinical Applications),pp.1-6
[3] Chandrapatala, Belgunda; "Introduction to Finite Element in Engineering";third edition PHI; 2012;pp276-286
[4] Peishan Hu, Rong Liu, Jie Liu, Glenn McRae, Matthew X. Yao, Rachel Collier, "Advanced Stellite alloys with improved metal-on-metal bearing for hip implants", Materials and Design 60 (2014),pp.424–432
[5] Twin Hook, W.H. Roerdink, M.M. Aalsmab, G. Nijenbanningb, D.P. van Walsum ," The dynamic locking blade plate, a new implant for intracapsular hip fractures: Biomechanical comparison with the sliding hip screw and Twin Hook", Injury, Int. J. Care Injured 40 (2009),pp.283–287
[6] Kasem Siamnuai, Supakit Rooppakhun, "Influence of plate length on the Mechanical performance of dynamic hip screw", IPCSIT vol. 23 (2012)
[7] Standard Specifications and Test Methods for Metallic Angled Orthopedic Fracture fixation Devices Designation: F384 – 12
[8] W. Kajzer, A. Kajzer, J. Marciniak, "FEM analysis of compression screws used for small bone treatment", AMME, volume33, issue 2, Aprial 2009.
[9] Hendra Hermawan, Dadan Ramdan and Joy R. P. Djuansjah, Metals for Biomedical Applications.

