International Journal of Engineering Inventions
e-ISSN: 2278-7461, p-ISSN: 2319-6491
Volume 9, Issue 9 [September. 2020] PP: 16-22

Material and Design Optimization of Artificial Hip Joint

Jainam Shah, Dr. Jeremy (Zheng) Li
Department of Mechanical Engineering\University of Bridgeport

Abstract: *Almost 2.5 million people in the United States have had hip replacement according to research by Mayo clinic [3]. Artificial hip joint is a device used to replace damaged part of hip joint with special material components designed by considering different aspects such as weigh of person, bone structure, daily activities of a patient. This treatment required due to Severe damage in Acetabular component which causes pain while doing day-to-day activities. Artificial hip joint is one of the toughest surgeries of human body. The main focus of this study was on simulating AHJ for a static and dynamic loading conditions. Also, different materials were tested using cross-sections and compositions of material using different designs.*

Finite element method was used to simulate stem and femoral components of Artificial hip joint. Results of the simulation were compared with standard data taken from one of the studies performed. Also, different design studies were performed to find out the best solution for the same design having less stress and deformation. There are so many different types of AHJ available in the market, as this study tried to combine them and create some newer design to make it simple to manufacture and also less in weight and cost.

In this study main focus was on two different kinds of loading criteria to find out the difference between static and dynamic loading conditions. The difference between them is negligible and that is why with the accurate condition all other simulations will be tested using the same boundary conditions and material properties. Multistep simulations are used to accommodate the iterative solver to get convergence. Both simulations were performed in static structural. Remote boundary conditions were used to make it as realistic as possible. AHJ used in this simulation has trapezoidal cross-section which is different from past studies.

Two materials are selected for femoral stem while applying static loading, Titanium (Ti-6Al-4V) and Chromium alloy (Co-Cr). CoCr found to be a better material choice for femoral stem in order to get better results for stress and deformations. As Trapezium cross-section has lesser amount of sliding and debonding at both interfaces, as compared to circular cross-section [1]. Bone conditions is also very important part to considered while selecting and designing AHJ because wet bone is less brittle compared to dry bones.

Keywords:*Artificial Hip joint, Material optimization, Design optimization, FEM*

Date of Submission: 05-12-2020 Date of Acceptance: 20-12-2020

I. INTRODUCTION

Hip joint is one of the most important joints in human Bones structure. As it is involved in human activities such as jumping, walking and running [4]. The artificial hip joint is required in order to replace the damaged components of Hip joint which mainly includes femoral bone head, Acetabular cup and Femoral stem which goes inside femoral bone. Before implanting such thing in human body, some simulations and design optimizations needs to be done in order to get maximum efficiency out of it.

To ensure prosthetic design safety relative to its mechanical behavior, different loading conditions shouldbe tested by performing FEM simulation. The average length of adult hip bone is almost about ¼th of total height. The healthy bone can withstand 10 times to its body weight, beyond which it can lead to fracture [6]. However, the effect of weight and sudden movement can increase the load to which the prosthesis is subjected by up to 10-20%, and in some cases even more significantly and this must be taken into consideration while designing AHJ, whether it will fail under this maximum load or not [5]. While designing AHJ normal static loading conditions has to be tested and also real-life loading condition while doing heavy activities such as jumping and walking takes place has to be tested in order to verify design.

The head prosthesis is loaded at an angle of 20̊ with a load of 3000N which approximates the peak gait load for a 70kg person [7]. Muscle load of 1250 N was applied at an angle of 20̊ to the vertical over the approximated area of the trochanter [2]. An ilio tibial-tract load of 250 N is applied parallel to the shaft of femur. Restraints were applied using remote boundary conditions in order to create an environment as realistic as possible.

II. GEOMETRY

CAD geometry is a must require and very first step in order to perform FEM simulation. CAD geometry was mainly created using PTC CREO student version using advanced modeling commands and then imported into Space claim, a modern design modeler of an Ansys. The geometry creation process was kept as simplified as possible and very close to realistic. But, at the same time ignorance of unwanted fillets, holes and other features which increases number of elements in the process of discretization.

The process of creating prototype and testing becomes expensive and time consuming, while creating 3D modeling using modern techniques and tools available we can easily simulate newly designed geometries with ease and at cost of very less time. CAD can easily perform 2D and 3D objects that can be useful before and after design for manufacturing reference.

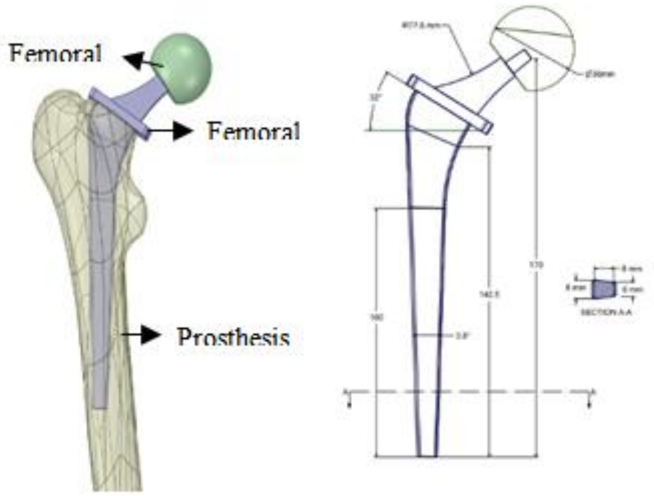

Fig 1 : CAD geometry of AHJ

Trapezium cross-section is used in present study to apply dynamic and static loading conditions.Total lengh of stem is around 170 mm and taper angle is 3 ̊, Cross section at distal location A-A is trapezium as shown in fig.1.As cross-section varies with change in length of stem it is important to create a geometry as realistic as possible for accurate results. Femoral head is a sphear in shape with hole inside with taper in order to provide support to stem head. Angle also plays an important role in specification of AHJ as it defines total angular movement of joint.

III. METHOD OF ANALYSIS

3.1 Material Properties

Four different materials were considered while selecting material for AHJ.

Table 1: Material Properties for AHJ assembly

Material	Young's modulus (Gpa)	Poisson's ratio	Fatigue Limit (Mpa)	Reference
Particulate reinforced Composites	25	0.35	40	[9]
Titanium alloys	100	0.30	550	[9]
Chromium	195	0.29	270-670	[10]
Ceramic	400	0.22	300	[11]

For static loading, Titanium and Chromium are selected for femoral stem and Ceramic for Femoral head. While, for Dynamic study, best from these two will be used to do further design optimization. Later for design optimization combination of composites and Chromium will be used for stem component.

3.2 Finite Element Method

FEM is mainly divided into 3 parts.

- Pre-processing
- Solving
- Post Processing

FEM method mainly consist of 5 steps to complete a simulation, Which starts with Engineering data: in which creation of material takes place according to requirement, Geometry import: which is created in other CAD softeware, Meshing: which is called descritization of geometry in small elements in order to integrate euqations over boundary conditions, Boundary conditions: defines the environment of eometry under which it is going to be used and Post processsing which gives us the results.

Descritization of geometry takes place for each components and element quality plays an important role in meshing, as it gives beter results for stresses. Finer mesh required at point of contact between femoral head and stem, as main force is being transmitted from head to stem. Total number of nodes and elements are kept within software limit.

Geometry	Type of element	Nodes	Elements
Femoral Stem	Solid 187	25088	14930
Femoral Head	Solid 187	6611	4182

Table 2: Meshing details

Solid 187 elements are used,which are 10 nodal Tetrahederal elements with mid-side node. These elements are higher order 3D elements which exhibits quadratic behaviour.

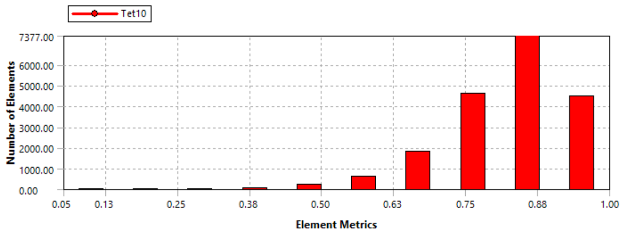

Fig 2: Mesh Matrix

Mesh matrix shows that mesh quality near "1" are good quality mesh while away from "1" are bad. For meshing Mechanical mesh were related using global element size. Aggressive mechanical approach is used with medium smoothing. Mesh defeaturing is captured and structured mesh created using above settings.

3.3 Static Loading

Static loading is applied on the Femoral head and collar of Femoral head as a part of resultant components. Loading is calculated and taken from standard data which was originally for 70kg person in steady state condition by ignoring activities such as Jumping, standing on one leg etc.

The head load is applied in downward direction at an angle of 20° with vertical magnitude of 3000 N. The surface of Femoral head is patched to apply load on only part of body where Acetabular cup is connected. Co-ordinate systems are created for each and every loading component to make it easier.

Trochanter load is applied on collar of femoral head in upward direction as a part of reaction forces from leg. A force component of 1250 N was applied on collar head in upward direction.

Traction force is applied on collar in downward direction as a part of resultant traction force. The traction force has a magnitude of 250 N.

Boundary conditions are used for restricting degrees of motion were remote boundary conditions. As according to original geometry, we have Prosthesis bone around stem and to simplify geometry and simulation we are not considering bone. But to accommodate effect of bone surrounding Stem remote boundary conditions are applied.

3.4 Dynamic Loading

As in static loading conditions several activities are ignored and constant value of force are considered, while in Dynamic loading we will apply forces in X, Y and Z direction on top of Femoral Head which will be transient load, means it will change with respect to time. Load was divided into 5 seconds and time step used were 0.5 seconds.

Load applied on top of femoral head is considered while imbalance of personnel takes place. That is the reason why maximum force is applied in X direction by considering fall in that direction. Chromium was selected as a material for Femoral head and ceramic for Femoral stem and boundary conditions also selected same as static loading conditions.

Simulation takes longer time than usual because of longer time run. Also, more computational power required to run this kind of simulations. Connectivity between parts were defined using bonded connection as using any other contact makes it nonlinear. Also used sub steps in order to get convergence and apply load gradually.

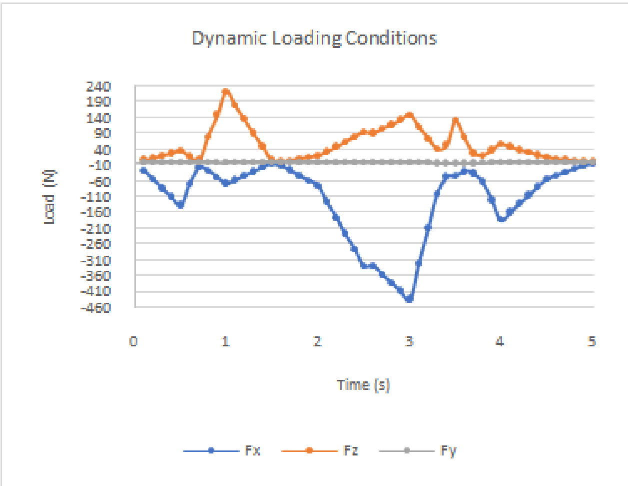

Fig: 3 Dynamic loading

3.5 Results

From Stress concentration results we can see that the maximum stress concentration is at contact point of collar and Femoral stem. The main reason for maximum concentration at that point is Stress Singularities. Which is called infinite stress value at specific location and also there is no way Femoral stem will fail from that location. So, that part can be neglected and according to the results plot on medical length of Femoral stem shows allowable stress values. Maximum stresses in Titanium are 80 Mpa at same location for Chromium is 63 Mpa. Also, deformation in Titanium is 20 microns while in Chromium is 10 microns. From this for further studies we will choose Chromium for dynamic loading and design optimization. Dynamic loading was applied on femoral head by creating surface patch in order to get surface contact area between femoral head and Acetabulum.

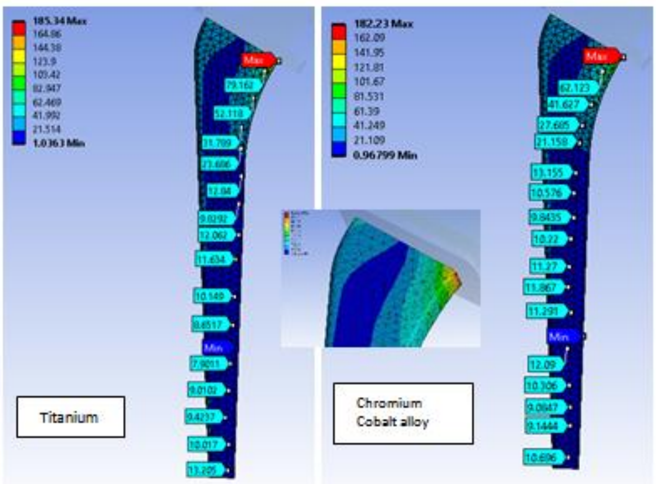

Fig 4: Stress contours for static loading in Femoral stem

	Material	young's modulus (Gpa)	Max Tensile Stress (Mpa)	Fatigue Limit (Mpa)	FOS
Static Loading	Titanium	100	185.34	550	3
	Cobalt	195	182.23	270-670	1.5-3.7
Dynamic loading	Titanium	100	280	270-670	1-4.8

Table 3: Factor of safety in static and dynamic loading

4 Design Optimization

For design optimization total 6 designs are chosen [2]. Designs are mainly dependent on material combination, where two different materials were chosen from given table 1. Particulate reinforces composites and Chromium were chose having 25 Mpa and 195 Mpa young's modulus respectively.

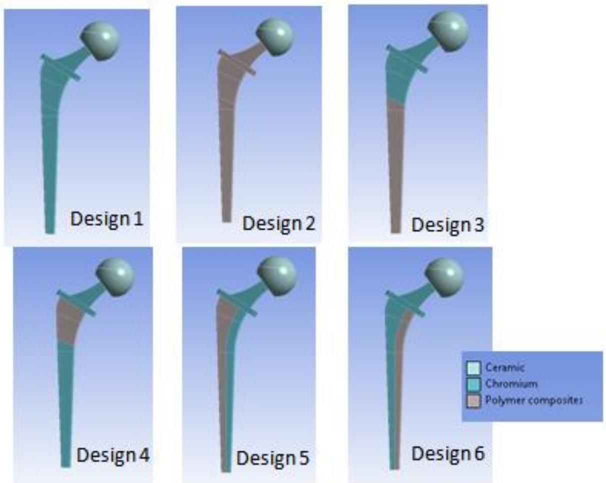

Fig 5: Combination of material

4.2 Results and comparison of Designs

Different designs are compared performance wise by applying static loading on each design. Materials assigned are according to color coding shown. Composition of Composites and Chromium are used in order to get lower stress value and lower deformation within geometry.

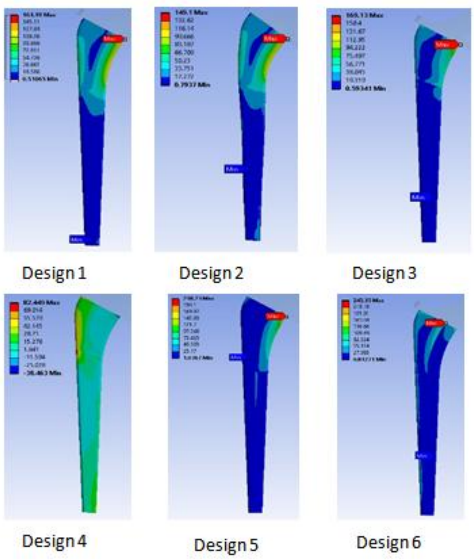

Fig 6: Stress concentration in different designs

Stress concentration in different designs is as shown in fig 6. Maximum stress concentration is again at the point of contact between collar and stem, which is stress singularity point. Stress concentration table is shown in table 4, which gives maximum stresses induced in designs and also factor of safety for same.

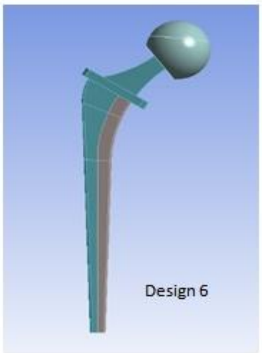

Fig 7 shows the comparison between 6 designs and gives maximum value of stress with respect to length in bottom half by creating construction geometry. From this result it is clearly seen hat Design 6 has less stresses compared to any other design and also stresses were calculated in top half in order to compare them, which are shown in fig 8. Also, in top half design 6 gives the smallest value for stresses. For that reason, we can choose design 6 over all other designs.

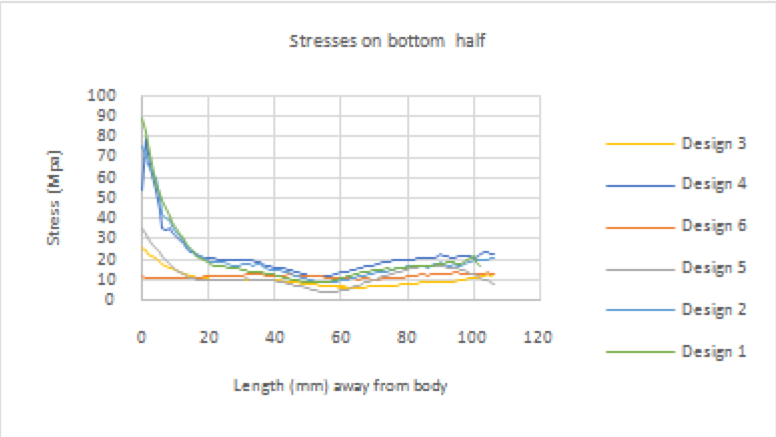

Fig 7: Comparison of stresses in bottom half

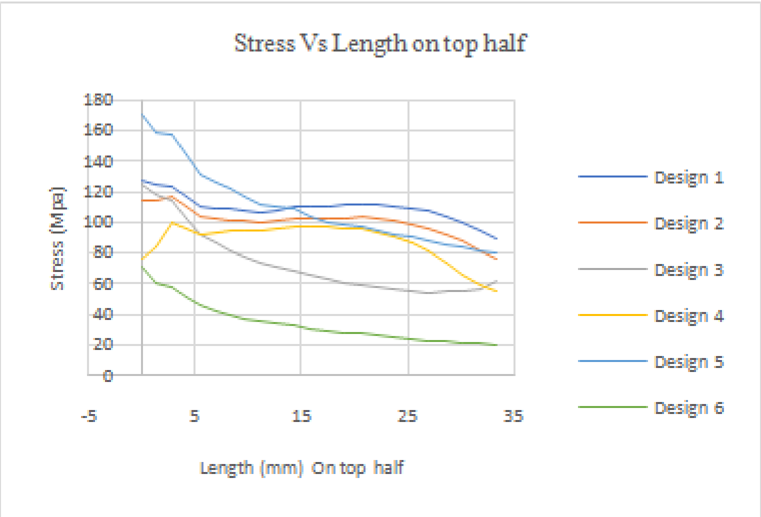

Fig 8: Comparison of stresses in top half

IV. CONCLUSIONS

Finite element analysis results showed that femoral stem with Trapezium cross section is better with design 6, with combination of Composites and Chromium material. It is observed that Chromium is better material than Titanium in static loading. Dynamic loading is tested on femoral head and which also shows the accountability of AHJ in dynamic loading conditions, means complex gait conditions. In dynamic conditions it is observed that the femoral head might get dislocate if more unusual force acts on joint. For future work topology optimization can be performed on same design for lattice structure which welcomes 3D printing for AHJ.

Reference

[1]. Shantanu Singh, A.P Harsha,"Analysis of Femoral Components of Cemented Total Hip- Arthroplasty Shantanu Singha "
[2]. Hussam El-Din FathiEL'Sheikh, "Finite element simulation of hip joint replacement under static and dynamic loading".
[3]. Mayo Clinic. http://www.mayoclinic.org/
[4]. D. J. Cleather, J. E. Goodwin, and A. M. J. Bull, "Hip and knee joint loading during vertical jumping and push jerking".
[5]. K. Colic, A. Sedmak, A. Grbovic, U. Tatic, and S. Sedmak, "Finite Element Modeling of Hip Implant Static Loading,"
[6]. A.Z. Senalp, O. Kayabasi, H. Kurtaran, "Static, dynamic and fatigue behavior of newly designed stem shapes for hip prosthesis using finite element analysis, Materials and Design".
[7]. Paul, J. P., "Loading on normal hip and knee joints and on replacements, " in Advances in artificial hip and knee joint technology (Edited by Holmann, D. and Schaldach, M)
[8]. Paul, J. P., "Approaches to design. Force actions transmitted by joints in the human body".
[9]. Henn, G., Prendergast, P. J., and Taylor, D., "Assessment of particulate composite material for use in hip joint prostheses,"
[10]. Fagan, M. J., and Lee, A. J. C., "Material selection in the design of the femoral component of cemented total hip replacements,"
[11]. Heimke, G., Jentschura, G., and Werner, E., "Direct anchorage of A1203 ceramic hip components: Three years of clinical experience and results of further animal studies,"
[12]. Kuiper, J.H. and Huiskes, R., "Friction and stem stiffness affect dynamic interface motion in total hip replacement,"