Musculoskeletal Modeling of Hip Joint and Fracture Analysis for Surgical Planning Using FEA

Sathya Ganapathi[1], **Shilfa Thoppay Premkumar**[1], **Thenmozhi Malaikannu**[1], **Kavitha Anandan**[1]

[1] Department of Bio-medical Engineering, Sri Sivasubramaniya Nadar College of Engineering, Chennai, India

Abstract

Background: Hip fractures are a major cause for disability in patients. They require immediate attention as they could otherwise cause death. Hip fractures are almost always treated with surgery by implantation. Implants are of various types accounting for the many variations in hip fractures.

Objectives: This paper presents the design and analysis of a hip implant using Finite element analysis. Fracture conditions are determined and the optimal design of the implant is obtained for improving healthcare and patient safety.

Methods: Anthropometric parameters of the human femur bone are collected from a particular age group. These are then used to obtain a CAD model of the bone using CATIA. The standard Charnley hip implant, used in total hip replacement surgery is also modeled. The proposed models are analyzed using ANSYS software by assigning appropriate material properties to the bone and implant. The stress distribution is observed when loads corresponding to normal gait conditions are applied. The load at which fracture occurs is then determined experimentally.

Results: Based on the analysis results of the modeled bone, the implant is optimized by varying the base cross section, the bio-materials used, and the design parameters so that, its stress response mimics that of the actual bone. It is found that the model no 2 as in Table 6 with head diameter 28mm, neck diameter 10mm, neck angle 128 degrees has minimum strain at the neck region with a value of 0.65 and is found to be suitable for implant design. Results show that initiation of fracture in the implant occurs at 2000N and complete fracture occurs at 2400N.

Conclusions: The 3D models are very useful in simulation of bone fractures and internal fixations with implants. In this work, the hip joint and implant model, developed in CATIA software, help to understand how these structures adapt to external forces disturbances [15]. This will help the doctors to chose the optimal implant for a particular patient. This leads to greater accuracy and patient specificity.

Keywords

Finite element modeling, Fracture analysis, Gait cycle, Hip joint, Charnley hip implant model

Correspondence to:

Anandan Kavitha
Department of BME, SSN College of Engineering
Address: Kalavakkam, Chennai – 603110
E–mail: kavithaa@ssn.edu.in

EJBI 2013; 9(2):27–36
received: January 16, 2013
accepted: March 25, 2013
published: August 30, 2013

1 Introduction

The hip joint serves a very important biomechanical function. While supporting the majority of the human body ($\sim 2/3$ of total bodyweight) the joint must simultaneously facilitate smooth articulation of the lower limbs to enable bi-pedal gait. During routine daily activities, forces on the order of 5.5 times bodyweight are transferred between the femur and pelvis [11]. Fracture is the most important aspect of joints. Factors such as abnormal joint geometry, body weight, and prior injury are stated as major cause for hip fracture. Heavy loads on joint are implicated as the significant cause for the hip fracture. In recent years the number of hip fractures continues to increase in elderly population. Worldwide, the total number of hip fractures is expected to overstep 6 million by the year 2050 [1]. Approximately one-third of fracture patients went on to receive a hip replacement [2, 3, 4, 5]. There are four different types of fracture patterns (i) Femoral head fracture, involving the femoral head. (ii) Femoral neck fracture (iii) Subtrochanteric fracture involving the shaft of the femur immediately below the lesser trochanter (iv) due to diseases such as Osteoporosis. All of these fractures are treated by strenuous surgical procedures that involve implantation. But total hip replacement surgery is done predominantly in older patients with femoral neck fracture as they would be unable to withstand plates or nails.

The design of the hip prosthesis has been modified continuously to grapple with the advanced technology and patients postulates. Many new approaches have been evidenced in hip joint analysis with better material design

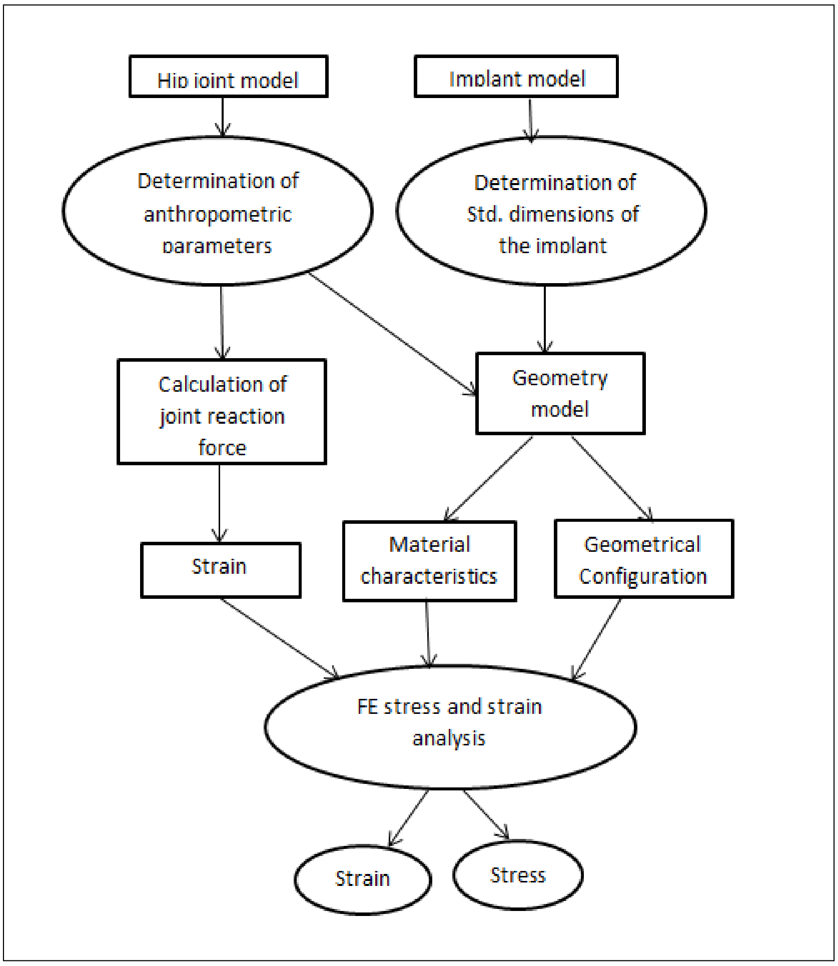

Figure 1: Flowchart for the approach.

and computer technology. Existing implants are not custom made for Indian anatomical specifications and hence cause problems in surgery. Also, choice of implant is a subject of debate. This proves the need for implant optimization.

In this study, three design parameters are chosen to investigate what further modifications can be made in the current implant design, that will increase its efficiency. This is done by observing the stress response of the implant using FEA. By analyzing through the computational protocol, distribution of stress can be depicted and the error and uncertainty for particular body gesture can be assessed [14]. In this work, it is proposed to model the acetabular-femur joint. In the development of modern computational techniques, attention of researchers has now turned toward using combined 3D reconstruction and virtual environment technologies to train clinicians and to help surgeons plan patient-specific, complex procedures like plastic surgery, surgery for trauma from accidents and reconstruction surgery. The 3D models are very useful in simulation of bone fractures and internal fixations with implants. These models are also important to understand how human musculoskeletal structures adapt to external forces disturbances [15, 16, 17].

Due to the usage of software the surgical procedures are avoided and it also helps to design a better implant. The subject-specific models are generated by using biomechanical modeling and analysis of a joint. In the recent scenario successful hip replacement surgery is done with the Charnley model implant in most of the cases.

2 Methods

Owing to the complexity of the work, it was divided into several phases as shown in Fig 1. The process can be broadly classified into Modelling and analysis. The implant and bone model are dealt with separately. The process is as explained below.

For the design of human hip joint three significant anthropometric parameters that vary with age, height, weight of a person are considered [16] and the biomaterial characteristics of the hip joint are also added to improve the accuracy of the study. For the load acting on the joint, theoretical calculation of joint reaction force is done with respect to the gravitational force acting on the hip joint and body weight, for a complete gait cycle. For determining the fracture condition the FEA method is used. The material characteristics of the bone and implant is summed up during FEA analysis. In the final phase FEA is accomplished and stress and strain distributions were obtained. Accuracy is assessed by comparison of theoretical value with that of experimental results.

Table 1: Measured anthropometric parameters.

Parameters	Obtained values	Mean	Standard deviation
Horizontal offset (in cm)	4.2 3.5 4.1 3.7 3.9	3.88	±0.03
Femoral head diameter (in cm)	4.7 3.9 4.5 3.8 4.3	4.24	±0.552
Neck shaft angle (in degrees)	133 129 130 128 131	131	±0.43

2.1 Modeling

Modeling involves using a set of measured anthropometric parameters to construct a CAD (Computer Aided Design) model of the femur bone. This will be a 3 dimensional computerized model of the actual bone and will serve as an input to the next stage i.e finite element analysis. The implant model can be constructed from standard dimensions.

There are many softwares that can be used for modeling a human joint like CAD, PRO-E, ABACUS, AUTOCAD, SOLIDWORKS etc.[7]. These softwares are programmed for designing perfect machinery and aerospace parts. CATIA provides a unique set of tools that provide benefits like faster design, reduced lead times, quality of surfaces, associative drawings and innovative solutions. Moreover, CATIA has been widely used for modeling biomechanical joints [12]. Thus, it has been chosen.

Selection of Anthropometric Parameters

Modeling the femur bone in CATIA requires that, some parameters that define the basic geometry of the bone be given as input. It has been proven in earlier publications that the following three parameters measured, will prove to be adequate in representing the bone's geometry. The parameters are femoral head diameter, neck shaft angle and horizontal offset. These parameters depend upon demography. Radiographic images (N=5) of the hip joint belonging to five male patients of age ranging from 45 to 55 are obtained from a hospital. The parameters are measured using digital caliper and micrometer. The values are tabulated and mean values are calculated and used for modeling. Patient demographics are as indicated in Table 1 and the measured parameters are as given in Table 2.

Table 2: Measured Demographic Parameters.

S. No	Age	Height (cm)	Weight (Kg)
1	55	151.25	58
2	43	150.5	55
3	49	156	60
4	45	153	56
5	47	154.5	59

Modeling in CATIA-V5

In CATIA V 5.0 2-D working plane is selected for initial modeling of hip joint. The measured anthropometric parameters are used as input various tools from Sketcher Work Bench, the sketch can be designed. The mean values used are as in Table 3. Various taskbars and menus help to draw a needed shape and dimension. Measured anthropometric parameters are given as input while designing the hip joint in the sketcher toolbox. The final 2-D design is converted into 3-D model by applying constraints to the sketch and creating a pad on exiting the workbench. The constraints mentioned here refer to the geometrical measurements such as length, breadth and angles. These values are implemented using the measured parameters. The hip joint model is as shown in Fig 2.

Table 3: Anthropometric parameters (average) used for modeling the hip joint.

Horizontal offset (in cm)	Femoral head diameter (in cm)	Neck shaft angle (in degrees)
3.88	4.3	131

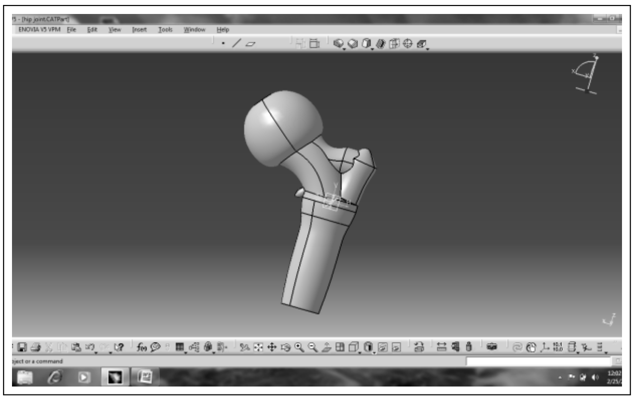

Figure 2: The designed hip joint model.

Charnley Hip Implant

The Charnley implant has 4 major parts- the acetabular cup made up of stainless steel 316L, the acetabular liner made up of Ultra High Molecular Weight Polyethylene (UHMWPE) that lubricates the joint and the femoral part with head, neck and tail portions which is also made up of stainless steel 316L. The liner fits inside the cup for replacing the function of cartilage in order to enable easy movement of the hip joint [18].

The standard dimensions of the implant are obtained from manufacturers, using which the implant is modeled in CATIA. The implant dimensions used have been tabulated in Table 4. The modeled implant is as shown in Fig 3.

Table 4: Standard Dimensions Of The Implant.

Parameters	**Values**
Neck shaft angle	128 degrees
Femoral length	162 mm
Head diameter	28 mm
Liner diameter	32 mm
Cup diameter	44 mm
Neck diameter	14 mm

This standard implant is further used for analysis and optimization. Optimization is done by varying the parameters like base cross-section, biomaterials and geometrical dimensions. These design parameters are varied and different models are done in CATIA and analyzed to obtain the most optimal values.

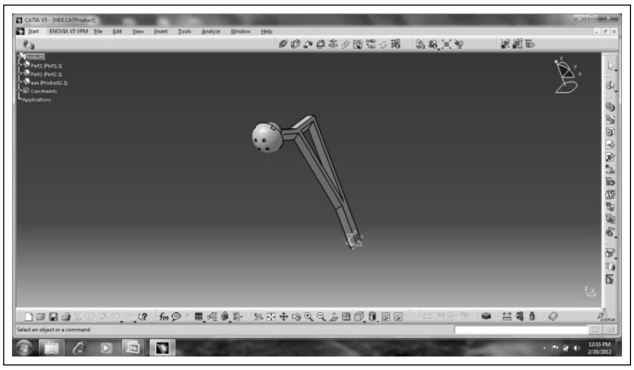

Figure 3: The designed implant model.

2.2 Finite Element Analysis (FEA)

FEA is a computational technique that is used to solve real world problems. Using FEA, it is possible to analyze and assess certain physical properties of objects. For example, FEA computes the strain developed in an object when subjected to a force or stress. The distribution of stress can also be viewed. Such computations are carried out based on the material properties of the specified object. Hence Finite Element Analysis will be of great use in observing the stress response of the hip joint.

ANSYS software is used to perform analysis. It is flexible, innovative, reliable, user friendly and compatible on complex structures like human bone joints.

Table 5: Phases of gait and load response on hip joint (BW = body weight) (1kg = 9.814 N).

Phases of gait	Planes of motion	Degrees of motion	Load applied (in BW)
Heel strike (initial contact)	Flexion	30	0.5BW
Single legged stance condition (loading response)	Flexion Abduction	5 4	2.7BW
Terminal stance	Extension	10	2.5BW
Swing phase (initial swing)	Flexion	20	1BW
Mid swing	Flexion	30	1BW
Terminal swing	Flexion	30	1BW

Assigning Material Properties

The bone is a composite material made up trabecular and cortical bone. These types not only differ in their structures but also in mechanical properties. The hard outer layer of bones is composed of compact bone tissue, so-called due to its minimal gaps and spaces. Its porosity is 5–30% in an adult skeleton. Compact bone may also be referred to as dense bone. Filling the interior of the bone is the trabecular bone tissue (an open cell porous network also called cancellous or spongy bone),which is composed of a network of rod- and plate-like elements that make the overall organ lighter and allow room for blood vessels and marrow. Trabecular bone accounts for the remaining 20% of total bone mass but has nearly ten times the surface area of compact bone [8]. Consideration of this difference during analysis yields better results. However this has not been considered in this work. The primary tissue of bone is made up of osseous tissue, is a relatively hard and lightweight composite material, formed mostly of calcium phosphate also called as calcium hydroxyl apatite which gives strength and rigidity to the bone. It has relatively high compressive strength, of about 170 MPa(1800 kgf/cm^2) [6] but poor tensile strength of 104–121 MPa [7] and very low shear stress strength (51.6 MPa), meaning it resists pushing forces well, but not pulling or torsional forces. For the surface of the bone with cortical structure, the appropriate properties are added and also for the region with trabecular bone together with the properties of calcium content [7, 8].

In the case of implants, four types of bio-materials are taken into consideration, depending upon their role as hip implants, such as Stainless steel 316L and UHMWPE, Tantalum, Ni-Ti alloy and Co-Cr alloy. The Metal alloys form the femoral stems, as they provide strength and endurance that allow the replacements, due to their solidity and resistance to wear and tear. Ceramic surfaces also provide a framework for Osseo integration. Polymers like ultra-high molecular weight polyethylene. The characteristic feature of the polymer is their elasticity and firmness

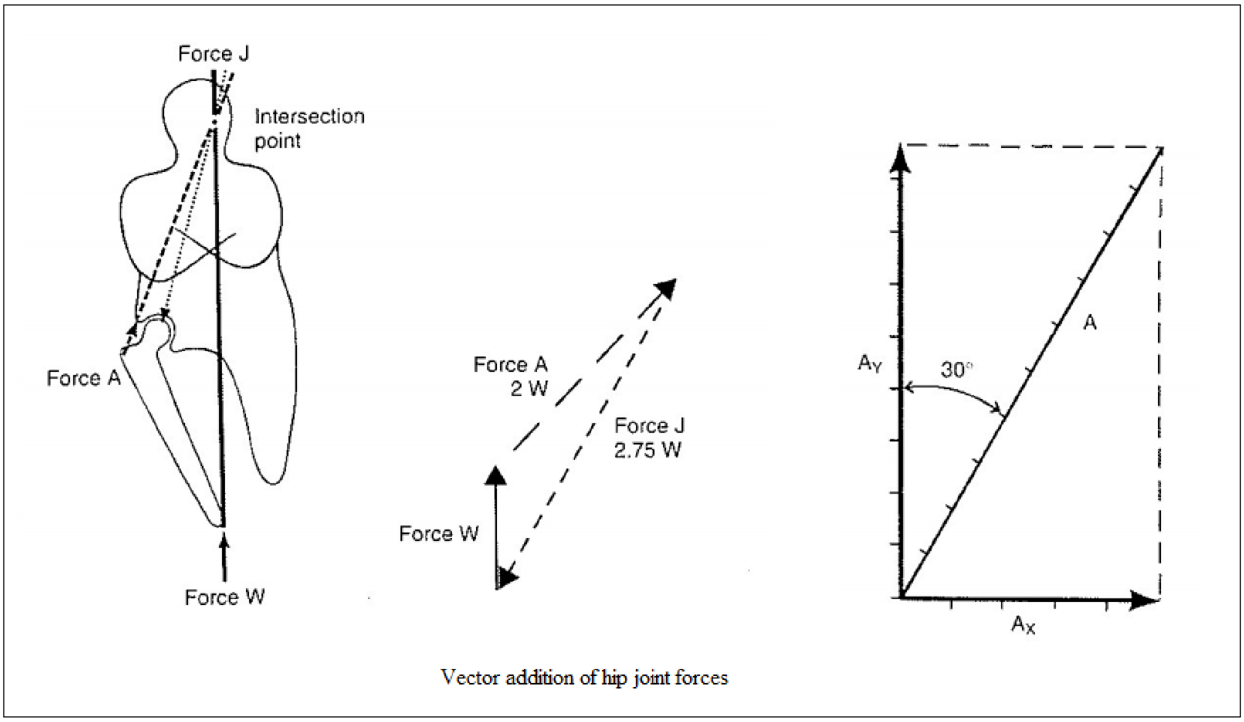

Figure 4: Free body diagram of hip joint during single legged stance condition.

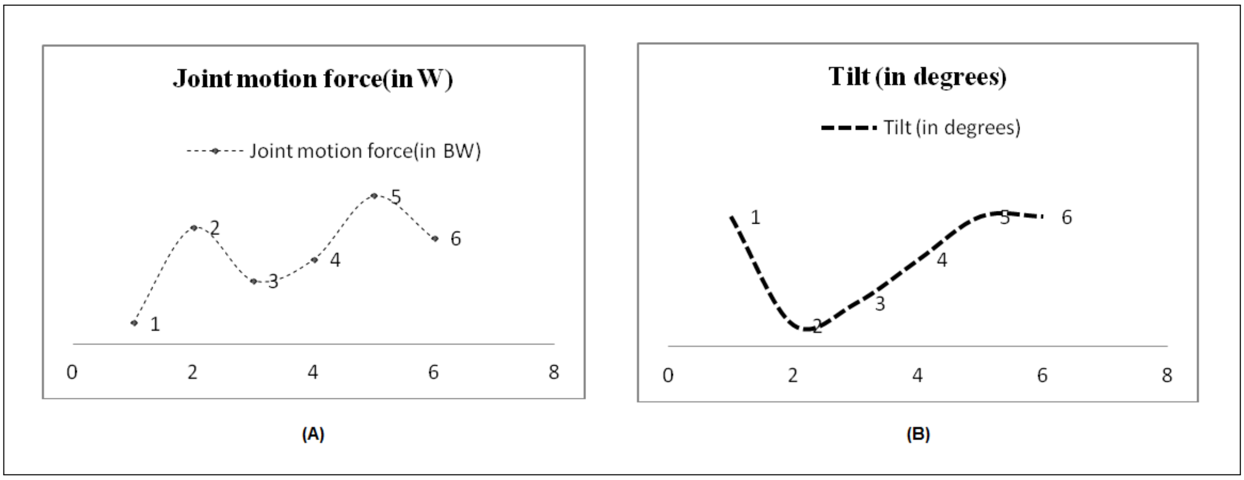

Figure 5: Graph A shows the joint motion force of the hip joint calculated using the free body diagram where BW is body weight and graph B showing the tilt of the joint (in degrees) during gait cycle 1. Heel strike, 2. Single Stance, 3. Terminal stance, 4. Swing phase, 5. Mid swing, 6. Terminal swing.

that offers frictionless joint mobility to hip replacement implants. Tantalum has a special feature and they are shape memory alloys [9]. A distinctive feature common to all these biomaterials is that all are biocompatible, that is they do not cause any inflammatory response and they seem to be natural [7, 8, 9, 10].

Loading Conditions

With every move our body makes, there is an internal force associated. This force is exerted on our limbs and thus cause movement. Hence, the bones in our body are naturally capable of bearing and reacting to these loads. It is possible to calculate these loads from free body diagrams that depict gait cycles.

Gait cycle is defined as the Series of rhythmical, alternating movements of the trunk and the limbs which results in the forward progression of the center of gravity. Gait cycle has two alternating phases 1) stance phase (60%) 2) swing phase (40%).

Stance phase of the gait begins with the initial heel contact and ends with toe-off. The tasks performed are - weight acceptance (i) initial contact (ii) loading response, - single limb support (i) mid stance (ii) terminal stance and (iii) pre-swing. Swing phase encompasses the entire time the foot is in the air for limb advancement. The task performed is – limb advancement (i) initial swing (ii) mid swing and (iii) terminal swing.

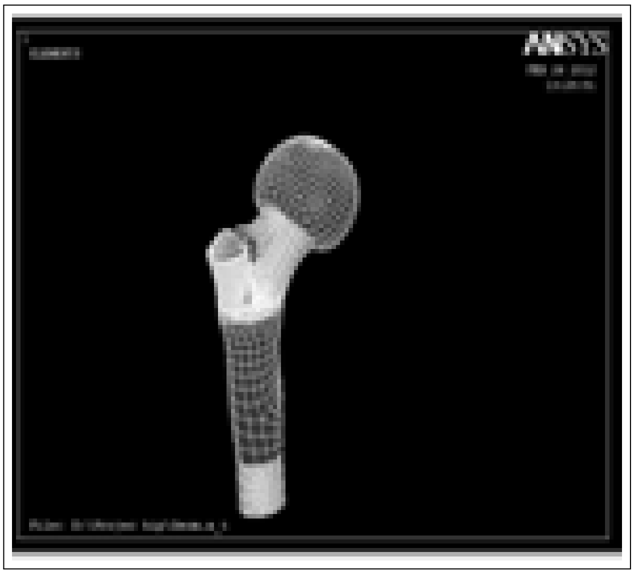

Figure 6: Mesh model of the hip joint.

Using the free body diagram technique as shown in Figure 4 and equilibrium equations, the maximum joint reaction force for each phases of the gait cycle is calculated [15]. The calculated joint motion force of the hip during each gait is shown in Figure 5.

The joint reaction force (J) is defined as the force generated within a joint in response to forces acting on the joint. It is due to the multiple muscles crossing the joint. A is the combined abductor muscle force and W is the body weight. From the free body diagram, the equilibrium equations for single legged stance phase are

$$A = 2W \quad (1)$$
$$Ax = A.\sin 30 = 0.5A = W \quad (2)$$
$$Ay = A.\cos 30 = 0.8A = 1.7W \quad (3)$$
$$J = Ax + Ay = 2.75W \quad (4)$$

The values obtained are as shown in Table 5. Figure 5 (A) shows the plot of the calculated joint motion force for each phase of a gait cycle and Figure 5 (B) shows the degree of tilt for each phase of a gait cycle.

2.3 Numerical Analysis

In order to perform FEA, the model needs to be divided into small regions. This process is known as Meshing and will help the computer to solve the problem efficiently. The element used to build the mesh is a tetrahedron and two or more elements are connected by nodes.

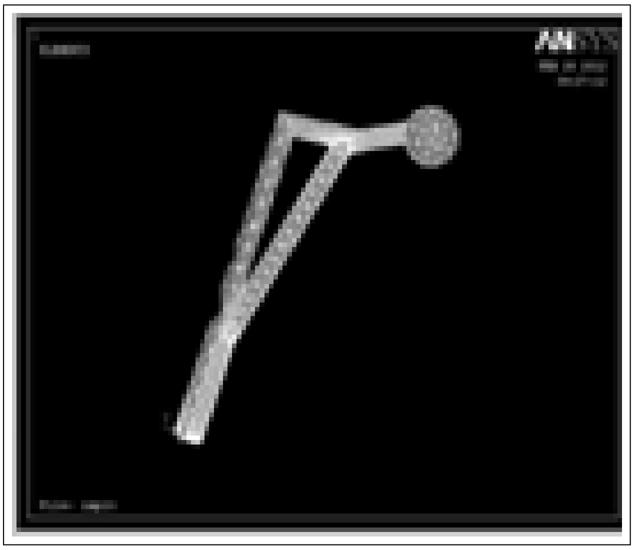

Figure 7: The designed implant model.

Table 6: Variation of design parameters of the implant.

Model no	Head diameter (mm)	Neck diameter (mm)	Neck angle (in degrees)
1	24	10	128
2	28	10	128
3	32	10	128
4	24	14	131
5	28	14	131
6	32	14	131
7	24	18	135
8	28	18	135
9	32	18	135

Various types of meshing modes are available in the software. 20 node 95 mode meshing is used for meshing the implant and free mesh h type mode is used for the modeled hip joint. Material properties are either selected manually or automatically from the ANSYS MATLIB file. The calculated loading conditions for each gait cycle are applied at the center of the femur head and the distribution of stress is observed. As a reaction to stress, strain is invoked. The software solves and determines these strain values invoked at the neck region of the hip joint. The meshed models of the hip joint and the implant are as shown in fig 6 and 7 respectively.

The results are obtained in the form of colour charts and animations. The red colour indicates maximum value of any particular result. Figure 8 shows the analysis of the femur bone during various phases of a gait cycle. It can be seen that the maximum stress is being held at the femoral head and is not being distributed to the shaft. The load is subsequently increased until the point of fracture. The point of fracture is considered to be the load at which the stress begins to penetrate down the length of the bone. This can be observed in Figure 8 (f).

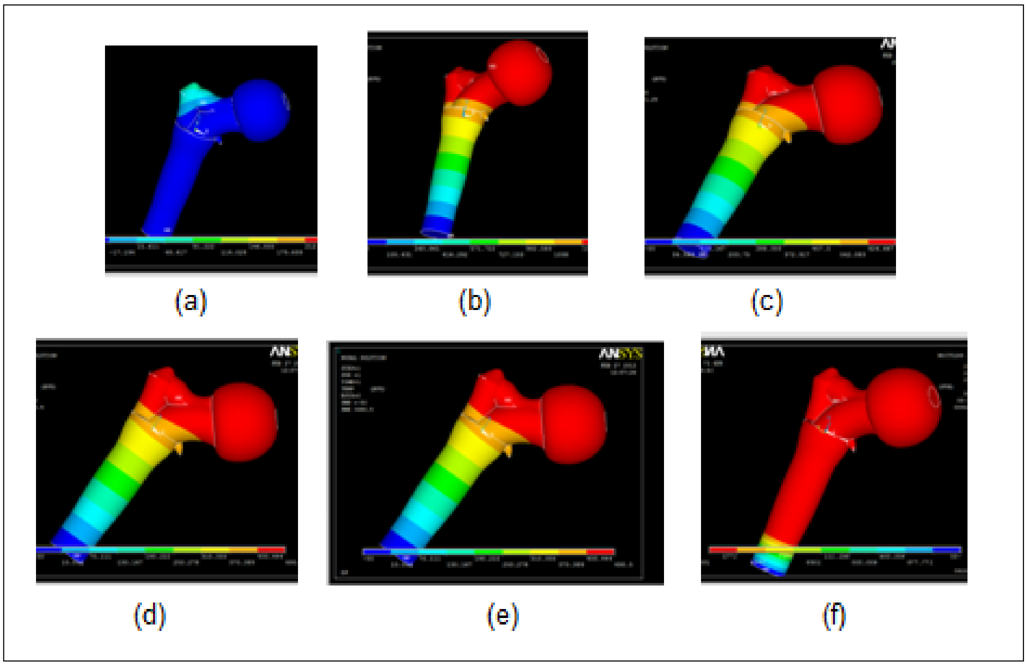

Figure 8: Hip joint analysis during (a). Two legged stance (b). Heel strike (c). Single legged stance (d). Terminal stance (e). Swing phase (f). Load of fracture.

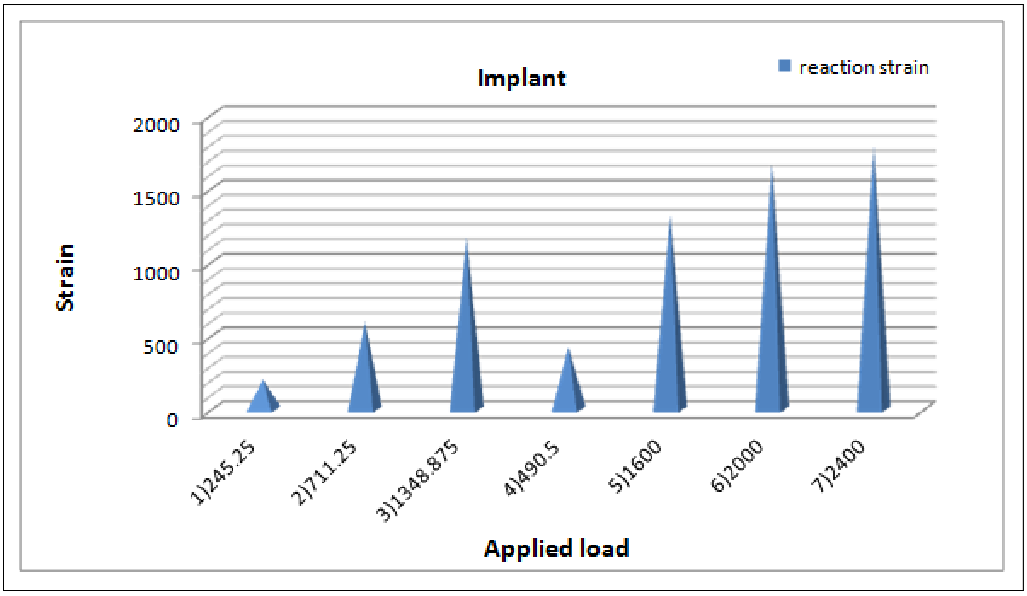

Figure 9: Graph depicting applied load versus reaction strain on hip joint for various gait phases 1. Two legged stance, 2. Terminal stance, 3. Heel strike, 4. Single legged stance, 5. Swing phase, 6. Initiation of fracture, 7. Load of fracture.

Different implant models are analyzed to identify the most optimal design. The first parameter considered, is the base cross section. Hexagon, pentagon, octal, and spherical surfaces are modeled to determine which cross section has a high load bearing capacity i.e. high Ultimate yield strength. Ultimate yield strength is defined as the maximum stress that a material can withstand while being stretched or pulled [13].

The second parameter considered is the bio-material used. Stainless steel 316L and UHMWPE, Ni-Ti alloy, Co-Cr alloy and Tantalum alloy are the chosen materials. They are analyzed based on the parameter Modulus of Rigidity, which measures the stiffness of the material. It helps to determine how the material deforms elastically and withstands heavy loads. Higher the modulus of rigidity better the implant suitable for implant [13].

The third parameter considered is the geometrical diametrical dimension. Head diameter, neck diameter and neck angle of the implant model are varied and the stress distribution is observed. The various dimensions considered have been tabulated in Table 6.

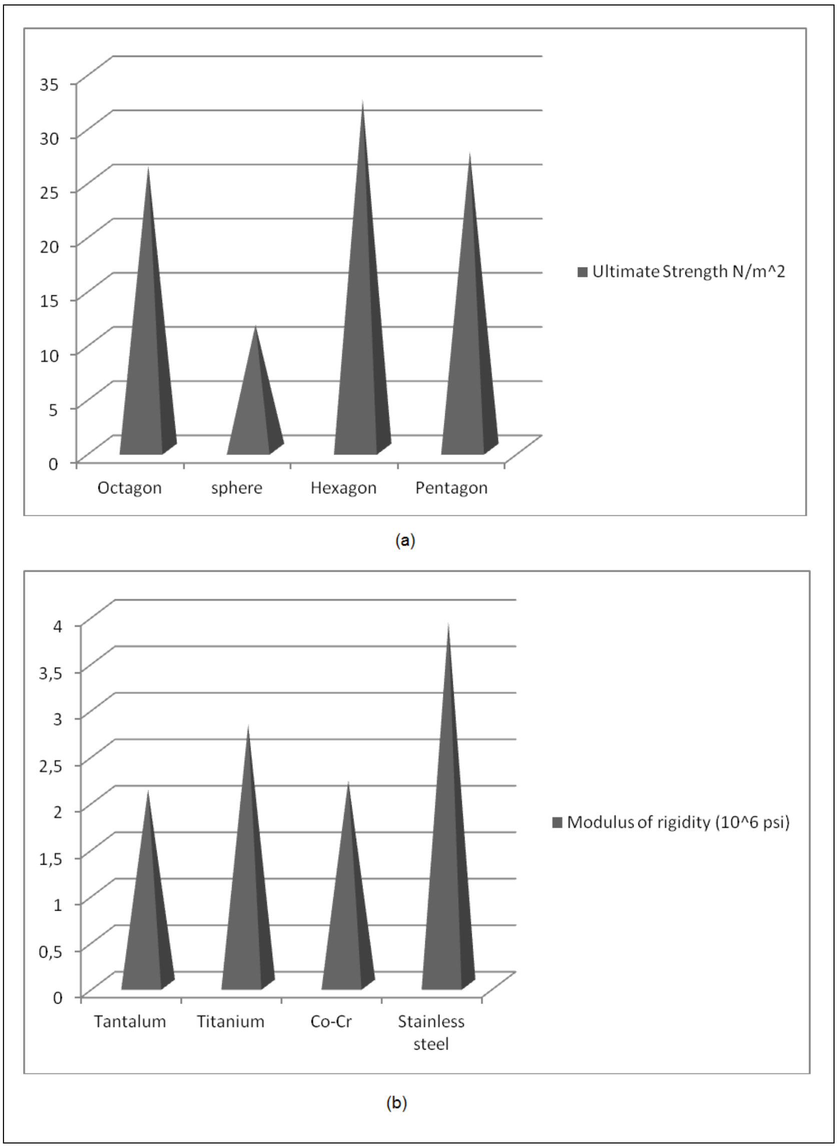

Figure 10: Graph depicting (a) the Ultimate Yield Strength for various base cross section (b) The Modulus Of Rigidity for various bio-materials.

3 Results

The model hip joint designed in CATIA proves to be a good representation of the femur geometry but it is not enough for further analysis. A more accurate method of modeling would be to reconstruct the femur bone from Computed Tomography scan data. The stress response can then be observed by actually inserting the implant into the bone as done during surgery.

According to analysis results shown in Figure 9, it is found that the initiation of fracture occurs at a load of 1600N and complete fracture occurs at 1600N and complete fracture occurs at 2000N. All other values are normal.

By analyzing the implants with above mentioned cross sections, it is found that hexagonal base cross section has better load bearing capacity with high Ultimate Yield Strength according to Figure 10(a). It has also been found that stainless steel together with UHMWPE has the highest modulus of rigidity according to Figure 10(b).

Since it is known to be bio compatible, it is considered to be optimal.

While analysing the models with variant dimensions, it is found that the model no 2 as in Table 6 with head diameter 28mm, neck diameter 10mm, neck angle 128 degrees has minimum strain at the neck region with a value of 0.65 and found to be suitable for implant design.

The final optimized implant model that implements the above mentioned results is shown in Figure 13.

Analysis is performed on the optimized implant for the same gait phases to comparatively analyze it's strength with that of the femur. The results of this are shown in Figure 11.

From Figure 12, it is found that initiation of fracture in the implant occurs at 2000N and complete fracture occurs at 2400N.

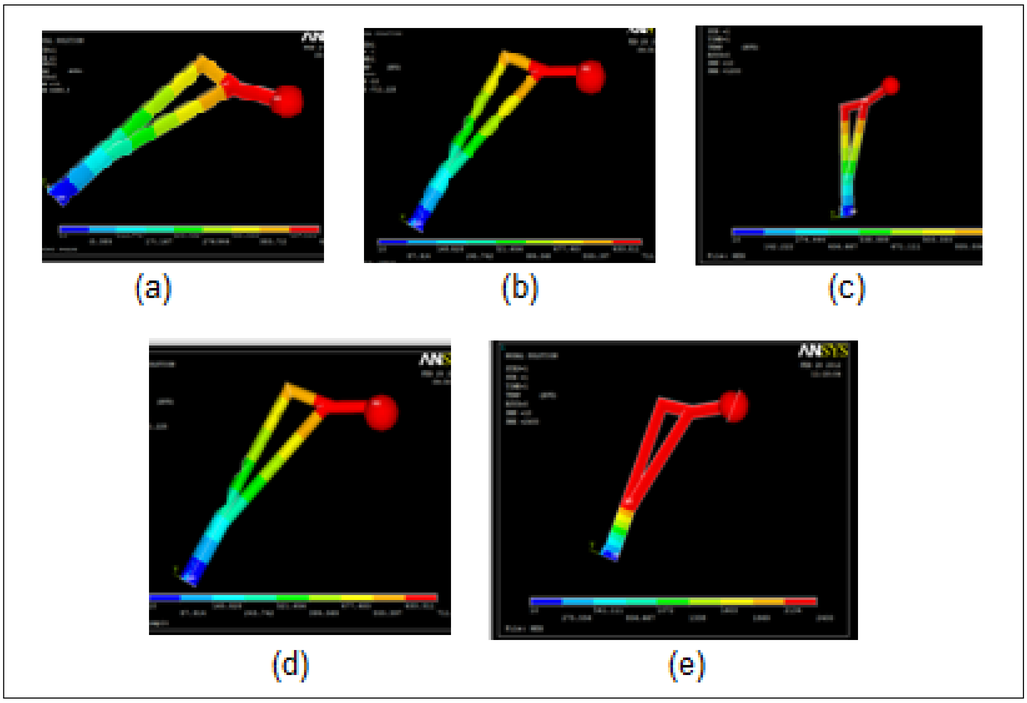

Figure 11: Depicts implant analysis for gait cycle phases (a) Two legged stance, (b) Heel strike, (c) single legged stance, (d) Swing phase, (e) Load of fracture.

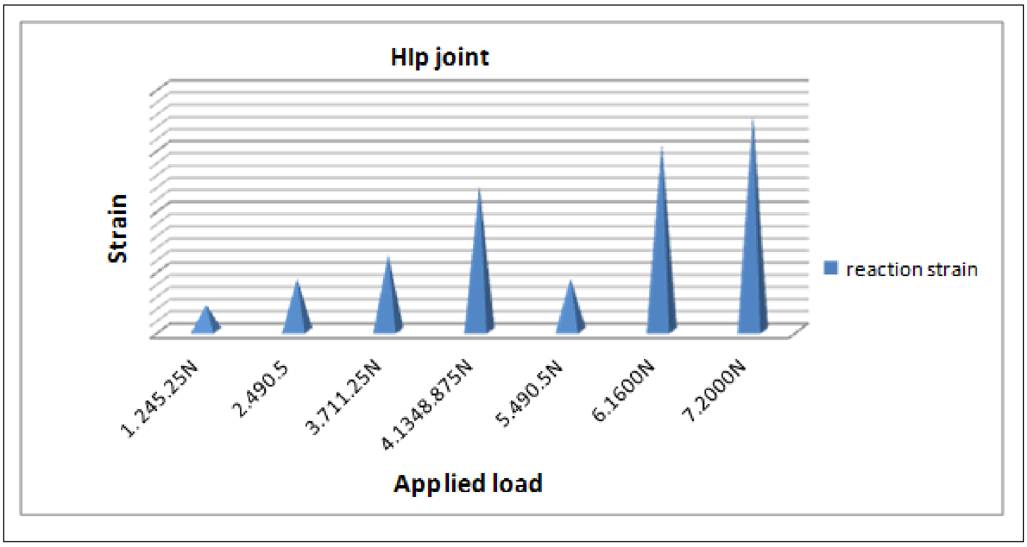

Figure 12: Graph depicting applied load versus reaction strain on the implant for various gait phase. 1. Two legged stance, 2. Terminal stance, 3. Heel strike, 4. Single legged stance, 5. Swing phase, 6. Initiation of fracture, 7. Load of fracture.

4 Conclusion and Discussion

It is found that the initiation of fracture occurs at a load of 1600N and complete fracture occurs at 2000N for the hip joint model, whereas for the optimized implant the fracture begins at 2000N and complete fracture occurs at 2400N. Hence the designed implant withstands better stress/load compared to the natural human hip joint. Distribution of stress and strain for various phases of gait is studied in both implant and the hip joint. It is found that most of the stress applied is withheld at the neck region of the implant without distributing it to the entire stem portion of the implant.

The multi-disciplinary approach presented in this paper is found to be suitable for the evaluation of stress-strain behavior of both the hip joint and the prosthetic hip implant components. The results were in accordance with some previous data.

However, better results can be obtained if the bone is modeled directly from imaging modalities such as CT or MRI. Such models will be a better representation of the real human hip joint. This study provides a non destructive approach towards implant designing, but the actual effectiveness can be determined only through laboratory testing and clinical trials. Computer Analysis can also be improved by using accurate values when considering material properties. For example, the trabecular and cortical bone regions have not been considered separately. Also, by obtaining material properties from gray level values, patient specific analysis can be done. Considerations like frictional forces, anatomy of the joint and other surface properties have not been considered in this analysis.

Considering the drastic increase in hip fractures over the last few years, it is important to produce patient specific implants as hip fractures cause both loss of finances and loss of life.

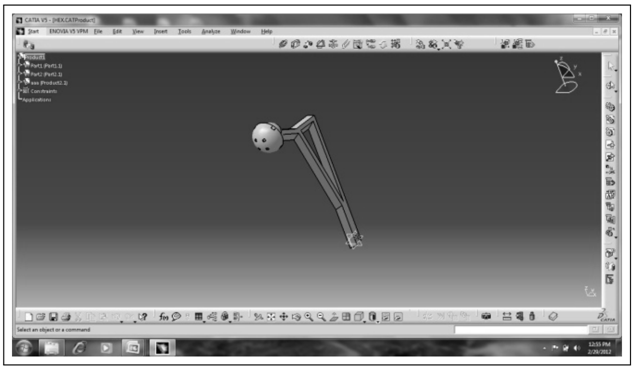

Figure 13: Optimized implant.

The 3D models are very useful in simulation of bone fractures and internal fixations with implants. In this work, the hip joint and implant model, developed in CATIA software, help to understand how these structures adapt to external forces disturbances25. This will help the doctors to chose the optimal implant for a particular patient.

References

[1] Andrew Edward Anderson, Computational Modeling Of Hip Joint Mechanics, The University of Utah, 2007.

[2] Anthony L., TarunGowsami, Hip implants VII: Finite element analysis and Optimization of cross-sections, J. Materials and Design 29:1438–1446, 2008.

[3] Alhassan S, Gowsami T, Wear rate model using UHMWPE in total hip and knee arthroplasty Mater. J. Materials and Design 29:289–96, 2010.

[4] Bauer T.W, Schils J , The pathology of total joint arthroplasty and Mechanics of implant failure Skeletal Radiol, Skeletal Radiol. 28(9):483-97, 1999.

[5] Blake Latham, GowsamiT,Effect of geometric parameters in the design of hip implants paper IV, J. Materials and Design 25:715–722, 2008.

[6] WillmannGerd , Coating of Implants with Hydroxyapaptite – Material Connections between Bone and Metal, Advanced Engineering Materials 1(2):5–105, 2007.

[7] Kannus P, Parkkari J, Sievänen H, Epidemiology of hip fractures, The Johns Hopkins University School of Hygiene and Public Health, Cambridge University Press 19:244-48, 1997.

[8] La Velle D G, Fractures of hip In: Campbell's Operative Orthopaedics, Mosby, Philadelphia 10:2873, 2003.

[9] Huddleston J M, Whitford K J, Medical care of elderly patients with hip fractures, Mayo Clin Proc 76:295, 2001.

[10] Mankin H J, The Reaction of Articular Cartilage to Injury and Osteoarthritis (Second of Two Parts), N Engl J Med 291:1335-40, 1974.

[11] Jonathan Wickert, Kemper E. Lewis ,An Introduction to Mechanical Engineering, 345-401, 2000.

[12] Dennis Jarvis ,Software Engineering And Manufacturing Systems For Case Tools, Chapman and Hall 28-29:302-307,1999.

[13] SaeedMoaveni, Finite element analysis: theory and application with ANSYS, 3:105-117, 2007.

[14] PaletiSrinivas, Sambana Krishna ChaitanyaDatti Rajesh Kumar, Finite Element Analysis Using Ansys 11.0, PHI Learning Pvt Limited 1:230-239, 2005.

[15] Margareta Nordin, Victor Hirsch Frankel, Basic biomechanics of the musculoskeletal system, Lippincott William Wilkins 3:430-505, 2000.

[16] BR Rawal, Rahul Ribeiro, Rajesh Malhotra, NareshBhatnagar, Anthropometric measurements to design best-fit femoral stem for the Indian population, Indian J Orthopedics46(1): 46-53, 2005.

[17] SudeshSivarasu,Pearline Beulah, Lazar Mathew, Optimization Of Skeletal Hip Implant Cross-Sections, VIT University, 2008.

[18] Bhatt Himanshu ,Finite Element Optimization Of Hip Implant Geometrical Parameters, North Gujarat University, 2006.

