International Journal for Research in Engineering Application & Management (IJREAM)
ISSN : 2454-9150 Vol-05, Issue-02, May 2019

Analysis of Femur Bone at various loads by using Finite Element Analysis

[1]B. Sreehari & [2]M. Vijay Kumar Reddy

[1&2]Assistant Professor, Department of Mechanical Engineering

Annamacharya Institute of Technology & Sciences, Tirupati, Andhra Pradesh, India.

[1]haribathala642@gmail.com

Abstract— Biomechanics is a field that combines with disciplines of biology, engineering mechanics, and mathematics, utilizes the tools of physics, computer aided design, computer aided manufacturing, and engineering to describe the properties of biological materials. In this work, Three dimensional models of human femur bones from CT Scan data in terms of DICOM form are be modeled by using MIMICS software and analysis is done by using ANSYS 14.5 software with three different materials at different loading conditions. From these analysis, results from three different materials shows that the behavior of bones at different stages and comparison graphs were drawn for each material. This work may help to reduce the complexity in critical surgeries, it can predict type of bone fractures and also the surgeon can able to analyze the complexity of risk and to provide quick treatment to the patients. From these analysis the comparative statement drawn among the three different materials (Stainless steel,Ti-6Al-4V and PMMA). PMMA shows the minimum deformation.

Keywords—: Computer Tomography, MIMICS (Materialize Interactive Medical Image Control System), Ansys.

I. INTRODUCTION

The word "Tomography" comes from the Greek: Tomo means slice, Grapy stands for to write. So, tomography literally means "writing slices". A computed tomography (CT) scan is a medical image developer that utilizes computer-processed X-rays that produce Tomographic images or 'slices' [1] of required specific areas of the body. The patient will lie on a narrow examination table that slides into and out of this tunnel and also rotating around patient. The x-ray tube and electronic x-ray detectors are located opposite each other in a ring, called a gantry [1]. The femur is the thigh bone, this connects from the hip joint down to the knee joint. The femur bone is a very strong bone and tremendous force may be required to cause fracture of the femur. Femur fractures are divided according to anatomic location. Fractures at the upper end of the femur are referred to as "hip fractures". Fractures of the femoral shaft involve the long tubular portion of the femur between the upper end and the lower end of the femur, and these are the fractures generally require higher energy for occurrence. Fractures of the femur just above the knee are referred to as "supracondylar femur fractures".

II. METHODOLOGY

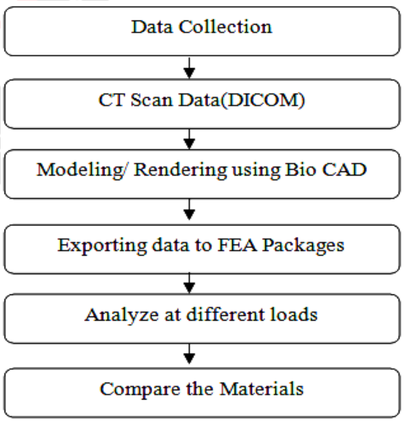


Fig.1 Methodology

2.1 DATA COLLECTION

Firstly need to understand the nature of the problem and its type. This first step in solving the problem to identify it. The general used material properties. Here the CT scan data is collected from different age groups. The CT scan Data is collected in the form of DICOM (Digital Imaging and Communications in Medicine) data sets (.dcm). Material properties of femur bone are collected from web and some information is collected from literature survey.

 DOI : 10.35291/2454-9150.2019.0018

2.1.1 MODELLING

Software used: Mimics (Materialize Medical Image Control System)

Procedure:

Step-1: Importing the person's DICOM files in the form of 'dcm; format.

Step-2: Volume Rendering process using options as follows;

-Thresholding

-Editing the mask in 3D

-Region Growing

-Calculating 3D

-Smoothing options

Step-3: The model is meshed by using the 3matic model

-Surface Meshing

-Volume Meshing

Step-4: Exporting the model to STEP (Standard for the Exchange of Product Data) [4] format.

The mimics data consists of 3D modeling images are in the form of STEP. These data sets are imported into ANSYS software. After importing the data, selecting of generating model is an important aspect. Initially the ten number of persons anatomical structures evolved from CT (Computer Tomography) scans are collected in the form of DICOM (Digital Imaging and Communications in Medicine) data sets (.dcm) format [4]. These data files are imported directly into the MIMICS software. Here, in this tool the 2D scan data is converted into 3D model by using various modules presented in it. The process of converting 2D images to 3D model is known as segmentation or Volume Rendering process.

- Importing the CT scan data into MIMICS
- Thresholding
- Editing Mask in 3D
- Dynamic Region Growing
- Calculate 3D
- Smoothing
- Surface Mesh
- Volumetric Mesh

2.1.2 IMPORTING THE DATA

The CT scan data consists of 2D scanned images are in the form of dicom data sets. The data sets are imported into MIMICS software. After importing the data, selection of planes is an important aspect. Generally in MIMICS, three planes XY (Coronal), YZ (Sagittal) and XZ (axial), Coronal defines the front view, side and top view of the images.

2.1.3 THRESHOLDING

Segmentation process stats with Thresholding. Thresholding is used to classify all pixels within a certain Hounsfield range as the same color or mask. By setting the lower threshold value, all pixels higher or equal to the set value will comprise the same mask. This helps in highlighting the bones, soft tissues, blood vessels and other body organs. By selecting bone(CT) option in the Thresholding module, system identifies the pixels that containing hard tissue nothing but it is a bone which having color compared to soft tissue that having dark cooler.

2.1.4 EDITING MASK IN 3D

Mimics helps in providing the mask editor that works in both 3D volumes or 2D slices. This masking involves adding pixels that have not been included in the segmentation process or removing excess pixels those bending to the part. The making operation works by painting over pixels that need to be selected for precise, a threshold range can also be set. Similarly, the 3D making operation performs a 3D erase along a cylindrical axis. Editing in 3D module helps in avoiding the contact between the hip region and femur neck region in all slices. Due to this femur bone separation becomes possible from CT scan dicoms.

2.1.5 DYNAMIC REGION GROWING

Dynamic Region growing option is used to separate the masks into different parts. This option is followed by only after doing the Thresholding option which highlights the bone position in 2D scanned images. This Region Growing helps in determining the new part from the anatomical structures, after completing the mask the femur bone gets separated from hip region. After its getting separated the new mask can be created by using the Dynamic Region Growing option present in the mimics software. As show in fig:1 [1], with this, the new mask is generated for femur bone.

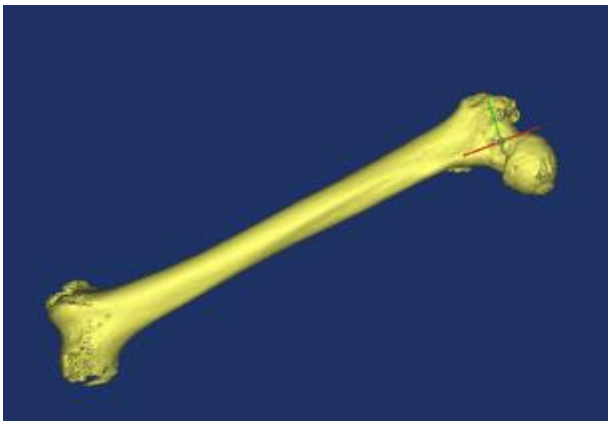


Fig2: Dynamic Region Growing.

DOI : 10.35291/2454-9150.2019.0018

2.1.6 CALCULATE 3D

By using the calculate 3D option, the 3D model can be transformed from the 2D images. After region growing, the new mask is generated from the femur bone. The femur bone mask that containing highlighter pixels are selected for creating the femur bone model by using the calculate 3D option.

2.1.7 SMOOTHING

The smooth option that provides the high-quality surface finish to the 3Dmodel. This operation helps in eliminating the irregular and unwanted surfaces that formed on the femur bone. The irregular surfaces present on the model may lead to bring out improper results when the model is analyzed. The smoothening operation that brings the fine results.

2.1.8 SURFACE MESH

The 3D model is described by the small elements called triangular elements. The sharp elongation are not suitable for analysis because the stress at one end of the triangle is different from other end. The CAE software requires the optimized shape of the triangles to reduce the computation time, so the number of triangles in mesh can be reduced. The process of optimizing triangular shape and its reduction is called surface meshing. This helps in fast computing the model that leads to easy interface between the modelling and analysis software.

2.1.9 VOLUME MESH

To do analysis using finite element method a complete volume description is needed. Generating a volume mesh from optimized surface mesh. The tetrahedral volume mesh can be generated from triangular surface mesh. The tetrahedral mesh can be created by using the 3-matic module present in the mimics software. This volume mesh provides the flexibility in determining the size, shape and parameters of the FEA software in the form of STEP format.

2.2 ANALYSIS

2.2.1. ASSIGN MATERIAL PROPERTIES

In this work, Materials like PMMA, Stainless Steel and Ti-6Al-4V are selected to carry out the analysis process. It is used to input the material properties of selected materials like density, Modulus of Elasticity, Poisson's Ration. in addition to it, problem geometry, geometry the grid, define the parameter and the boundary conditions to the code, The following boundary conditions are defined in the present structure analysis of model.

2.2.2. APPLY BOUNDARY CONDITIONS

Fixed support at the tibia/bottom end of femur bone by selecting all the face of bottom end of femur as show in fig:3. Its means it is fixed in all three directions and free direction and free at the free at the top/head end in all three directions.

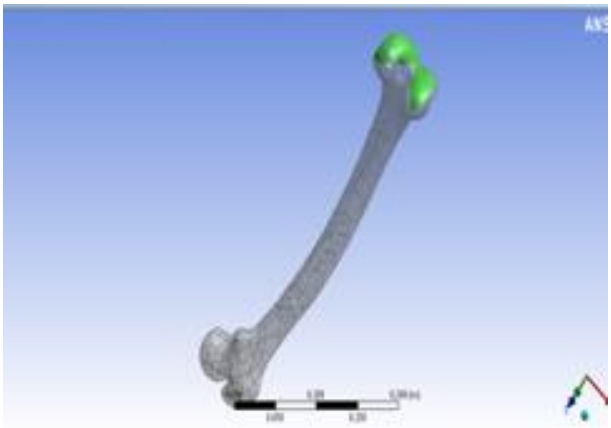


Fig.3 Applying Boundary Conditions

2.2.3. APPLY LOAD

Applied force of different loads of different loads will act in downward Z-direction in current case because we assumes the average weight of adult person is nearly comes of define weight at the femur head in downward direction applied by selecting the circular/circumferential area which shows in Fig:4. Green color.

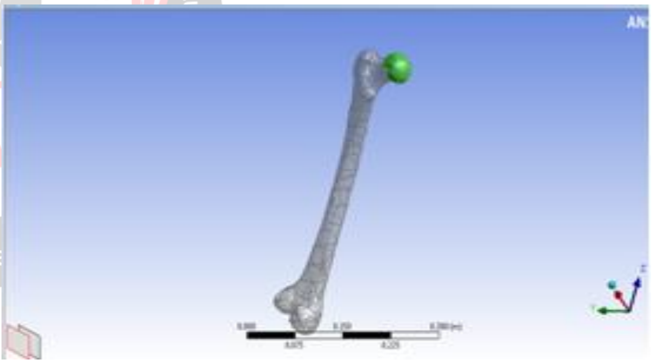


Fig.4 Load at hip joint.

III. RESULTS & DISCUSSION

From Finite Element Analysis, Analysis was carried out for the femur bone at different loads for three selected materials like PMMA, Stainless Steel and Ti-6Al-4V. Table.2 shows the analysis was done with different loads for PMMA material. Table.3,4 and 5 shows that PMMA, Ti-6Al-4V and Stainless Steel material behaviour of Total deformation, maximum principal stress, Equivalent stress increases with increase with increase in load.

 DOI : 10.35291/2454-9150.2019.0018

ISSN : 2454-9150 Vol-05, Issue-02, May 2019

Table:2 Results analysis of femur bone at the age of 35years for PMMA

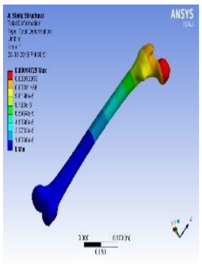
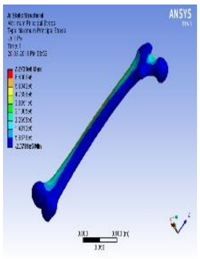
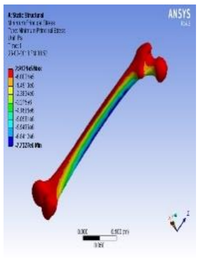
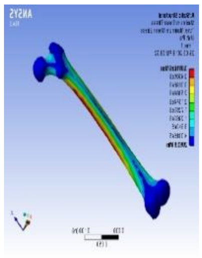
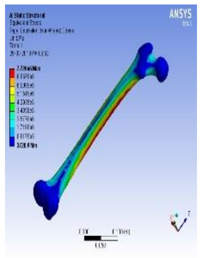
Material and person at load (45N)	Total Deformation	Max-Principal stress	Min-Principal stress	Maximum shear stress	Equivalent Stress
PMMA					

Table:3 Results for PMMA at different loads

S. No	Age (yrs.)	Gender	Load N	Total Deformation (mm)	Equivalent Stress (N/mm^2)
1	35	M	30	0.030364	3.366
			35	0.035425	3.9271
			40	0.040486	404881
			45	0.045471	5.0491
			60	0.060729	6.7321
			70	0.070852	7.8541
			80	0.080972	8.9761
			90	0.091093	10.098

Table:4 Results for Ti-6Al-4V at different loads

S. No	Age (yrs.)	Gender	Load N	Total Deformation (mm)	Equivalent Stress (N/mm^2)
1	35	M	30	0.065462	3.9828
			35	0.076373	4.6466
			40	0.087283	5.3104
			45	0.098193	5.9742
			60	0.13092	7.9655
			70	0.15275	9.2931
			80	0.17457	10.621
			90	0.19639	11.9848

Table:5 Results for Stainless Steel at different loads

S. No	Age (yrs.)	Gender	Load N	Total Deformation (mm)	Equivalent Stress (N/mm^2)
1	35	M	30	0.36813	21.467
			35	0.42949	25.045
			40	0.49084	28.623
			45	0.5522	32.201
			60	0.73625	42.935
			70	0.85898	50.091
			80	0.98169	57.7247
			90	1.1044	64.402

Finally, all the materials were compared at different load conditions by fixing the age of the femur bone at 35years. Here, from the Table.6 shows that comparison of three selected materials under different loads. The results shows that the PMMA material exhibiting better material behavior compared to other selected materials at femur spherical ball.

 DOI : 10.35291/2454-9150.2019.0018

Table:6 Results parameters Comparison of femur bone model for different materials at various loads.

S. No	M/F	Age	Load	Total Deformation (mm)			Equivalent Stress (N/mm^2)		
				PMMA	Ti-6Al-V4	Stainless Steel	PMMA	Ti-6Al-V4	Stainless Steel
1	M	35	30	0.030364	0.065462	0.36813	3.366	3.9828	21.467
			35	0.035425	0.076373	0.42949	3.9271	4.6466	25.045
			40	0.040486	0.087283	0.49084	404881	5.3104	28.623
			45	0.04547	0.098193	0.5522	5.0491	5.9742	32.201
			60	0.060729	0.13092	0.73625	6.7321	7.9655	42.935
			70	0.07085	0.15275	0.85898	7.8541	9.2931	50.091
			80	0.080972	0.17457	0.98169	8.9761	10.621	57.7247
			90	0.091093	0.19639	1.1044	10.098	11.9848	64.402

Graph plotted by considering load on horizontal axis and total derormation on vertical axis. From these graph it is clearly shows that as the load keepon increasing deformation also increased. From the graph as shown in Fig.5, stainless steel has maximum deformation and Ti-6Al-4V also has less with comparison with stainless steel but not with PMMA. PMMA has minimum deformation comparing with other selected materials for analysis. Like wise, Equivalent stress were analysed parallely for the three chosen materials as shown in Fig.6 and 7.

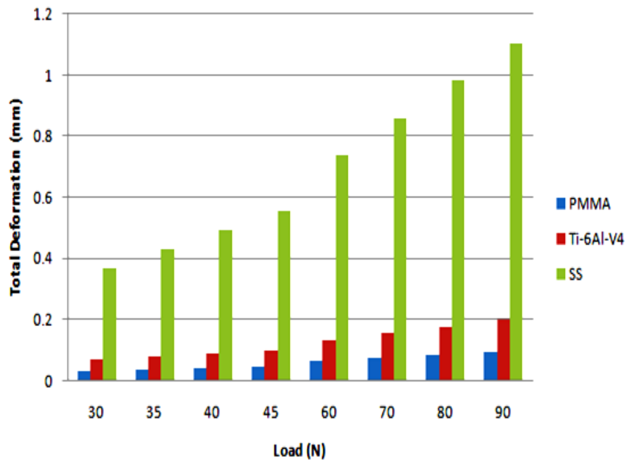


Fig.5 Graph plotted between Load and Deformation

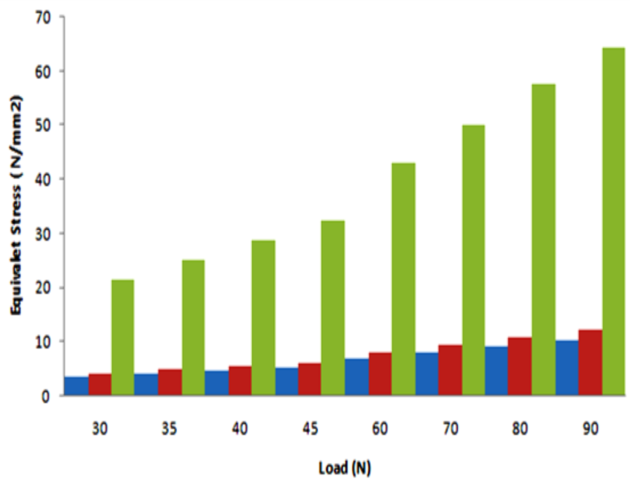


Fig.6 Graph plotted between Load and Equivalent Stress

IV. CONCLUSION

This work may help to reduce the complexity in critical surgeries, can predict type of bone fractures and also the surgeon can able to analyze the complexity of risk and to provide quick treatment to the patients. From these analysis results, it is clear that the deformation on the three different materials (Stainless Steel, Ti-6Al-4V and PMMA) there is minimum deformation in PMMA.

REFERENCES

[1] Baradeswaran. A, Joshua Selvakumar.L and Padma Priya. R, Reconstruction of Images into 3D Models using CAD Techniques,European Journal of Applied Engineering and Scientific Research, 2014,3(1):1-8.

[2] Ashish B. Deoghare, P.M.Padole Finite Element Analysis of Three Dimensional Medical Model Generated from CT Scan data. 13th National Conference on Mechanism and Machines (NaCoMM 07), IISc, Bangalore , India, December 12-13, 2007.

[3] Ajay Dhanopia, Prof.(Dr.)Manish Bhargava, Finite Element Analysis Of Human Fractured Femur Bone Implantation With PMMA Thermoplastic Prosthetic Plate, Procedia Engineering 173 (2017) 1658 – 1665.

[4] M Vijay Kumar Reddy,BKC Ganesh , KCK Bharathi and P ChittiBabu Use of Finite Element Analysis to Predict Type of Bone Fractures and Fracture Risks in Femur due to Osteoporosis Reddy et al., J Osteopor Phys Act 2016, 4:3.

[5] W. Kajzer, A. Kajzer, J. Marciniak, FEM analysis of expandable intramedu-llary nails in bone, journal of Achievements in Materials and Manufacturing Engineering, Volume 37, 2009.

[6] Mimics Student Edition Course Book,

[7] https://www.scribed.com/doc/214195083/Mimics - Student-Edition-Course-Book.

 DOI : 10.35291/2454-9150.2019.0018