A Comparative Three Dimensional Finite Element Analysis of Pelvic Bone

Sweta Kuraria[1] Prof. (Dr.) Veerendra Kumar[2]
[1]M.E. Student [2]Professor
[1,2]Department of Mechanical Engineering
[1,2]Government Engineering College, Jabalpur, M.P. India

Abstract— In this paper numerical model applicable to the pelvic bone biomechanics of reconstructed pelvic are presented. Geometric complexity and non linearity of the materials of pelvic make the analytical solutions of the mechanical behaviour of the hip joint difficult. The hip is the most complex joint within the human body. The aim is to achieve better understanding of mechanics of hip joint of reconstructed pelvic bone. Stress analysis of the pelvis during physiological loading conditions is necessary to understand the load transfer mechanism. A three-dimensional computer model of pelvic was used to examine the relationship between the position of the components and the range of motion considering simple inclination of 30 degree and both inclination and anter-vention of 30 degree and 20 degree, for which the right human hemi pelvic due to symmetry in shape of four female patient age group between 35 to 45 year has been taken into account. These data will allow surgeons to select implant Positions and also help to understand the biomechanical behaviour of pelvic bone.

Key words: Finite Element Method (FEM), Pelvic, Mimic

I. Introduction

The hip joint is one of the most flexible joints and allows a greater range of motion than all other joints in the body except for the shoulder. The hip joint consists of an articulation between the head of femur and acetabulum of the pelvis. Pelvis is formed from three bones: the ilium, ischium and pubis, which fuse together to form the ox coxae, or innominate bone. At the point of fusing they form the acetabulum. A round, cup-shaped structure on the os coax, known as the acetabulum, forms the socket for the hip joint. The acetabulum is a cup-like depression in the lateral side of the pelvis, the head of femur is hemispherical, and fits completely into the concavity of the acetabulum. The role of the pelvis is to transfer gravitational and external load across the sacro-illiac joints and the hip joints the pelvis forms a girdle which protects the digestive and reproductive organs. Pelvic bone due to both its shape and its structural architecture, the mechanics are also complex [1][27]. Bone densities throughout the pelvic bone and nondestructive mechanical testing was used to obtain young's moduli and poission,s ratio in three orthogonal direction [2] the major part of the load is transferred through the cortical shell. Although the magnitude of the hip joint force varies considerably[3] [21].developed a realistic 3d pelvis and finding out the stress pattern of the pelvic bone during normal walk specially in the area around the pubic ramus and the acetabular cavity and pressure force exerted on coxo femoral joint using geometric plane technique considering frontal and saggital plane is evaluated [4] [5]. Analysis for 13 side impact test from no fracture to acetabular fracture and lateral impact test to explore pelvis stiffness and pubic symphysis [6]. Analysis, based on the reynolds' data, is performed to determine significant parameters for each class of pelvis [7] and fem method is used to determine energy absorption capabilities in which result suggested that interior part of pelvis is most sensitive region [8]. Determined equivalent elastic modulus, yielding stress and damage plastic strain representing combined contributions of material properties and cortical bone thickness to pelvis bone resistance [9] 3d fem analysis of the hip bone to investigate magnitude , load direction, and stress distribution under physiological loading condition is performed[10] three cadaveric embalmed pelves were strain gauged and used in mechanical experiments in which FE models were generated from the spare ct data scans of the pelves and uses of sparse ct database for autogenerating accurate FE model of femur and pelvis [11] [13]. The pelvic bone geometry was reconstructed from a set of computed tomography images, and a hexahedral mesh was generated using a new octree based meshing technique[14] .the full pelvis model was validated against measured force-time impact responses to study the biomechanical response of the symphsis during the experimental impact [12][28] ct scans of 38 non-pathologic individuals were analyzed and functional orientation was computed as the density-weighted average of the acetabular surface normals based on surface density maps which suggested that average functional and anatomic abduction and anteversion angles ranged from 32°-58° and 22°-31° [15] problem of osteo -porotical changes in human pelvic bone was modelled in form of establishing of material property obtained from qct and decreasing of bone mass appears in individual region only the stresses decrease at the beginning and next increase while the bone mass still decrease [16] [17][29] modeling and simulation of biomechanical systems presented the results of stress and strain analysis of an orbital cavity and human pelvis loading rig for static and dynamic stress analysis [18] [19] structural meso-scale bone remodelling of the pelvis is performed in which model was to observe the difference in structure between a person who undertakes all activities and one which rarely uses stairs [20] [23] linear elastic finite element study is carried out to determine contact pressure distribution in acetabulum region using finite element analysis [22] twenty pubic bones taken from embalmed adult human cadavers (12 male, 8 female) were used to determine normal stress pattern and finite element analysis is used to assess the effects of inferior pubic rami and ischial fractures [24][25] [26]

II. Material and Method

Pelvic bone stresses under physiologic loading condition is essential for understanding the failure mechanism and implant design. The aim is to develop a computation model to accurately predict the bone architecture of the pelvis and justify structural features for which radiograph and ct scan of a normal hip were selected these digital ct images were used to create three-dimensional computer models of the hemi pelvis by using materialise interactive medical control system (mimics) and surface mesh is generated after which it convert into volumetric mesh using software called ABAQUS and then assigned material properties to these model using mimics and then model is transferred into ANSYS for finite element analysis under physiological loading condition .

A. Image Acquisition

The Computed Tomographic [CT] scan data of total pelvic bone of normal individual female patient of four female patient age from 35 years to 45 years are collected .this geometrical data of real proximal human pelvic bone is in the form of DICOM file which obtained from by using GE Ultrafast High Resolution Multislice CT scanner(16) slice containing total number of 326, 217, 316 and 321 slices respectively and pixel size of 0.826mm, 0.732mm, 0.793mm and 0. 797mm respectively, slice increment of 1.0 mm and resolution of 512 x 512.

B. Geometric Modeling

A comparative study is performed for which four model of right hemi pelvic of female patient is created model 1 of 35 year model 2 and 3 of 40 year and model 4 of 45 year old. MIMICS is used for visualization and segmentation of ct images then bone tissue is extracted by means of thersholding using default values range from 226hu to 3071 HU. The model is then developed by using various tools like edit mask, region growing,calculation of 3d mask and morphometric operation. The model developed is shown in the fig 1

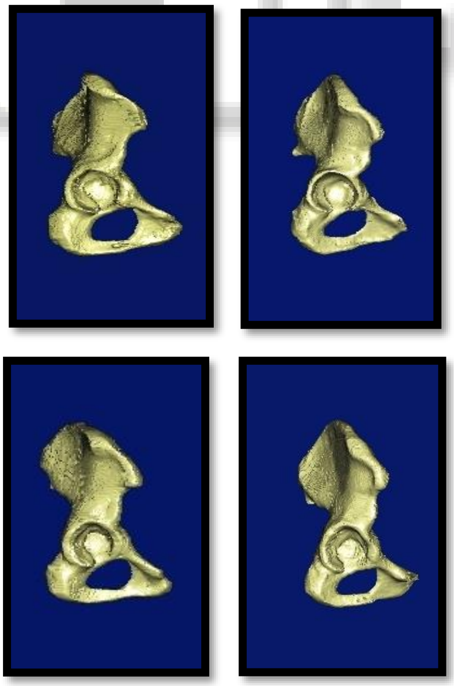

Fig. 1: Three Dimensional Model Of Proximal Femur

C. Creation of Finite Elemental Mesh

By creating 3D model in MIMICS, the surface mesh is generated by using remesh tool for 3D model. Automatic remesh operation is used for surface mesh in which equilateral triangle generated. Triangular reduction technique is used to optimize the parameters and preserve the quality of model the self intersection test is performed to eliminate intersecting triangle completely. After that these model are now imported in ABAQUS 6.10 to convert surface mesh into volumetric mesh by using edit mesh in which tri to tetra element. The four model are meshed with following number of tetrahedron element 727896, 530684, 899442 and 626799 respectively. Fig 2 shows volumetric mesh model of hemi pelvic bone obtained from ABAQUS.

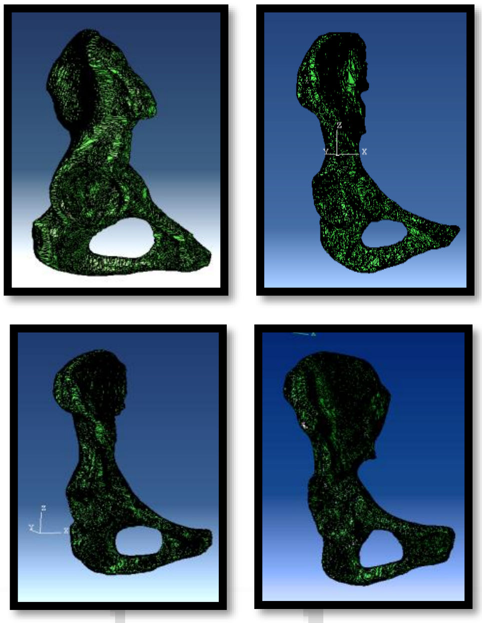

Fig. 2: Three Dimensional volumetric meshed models of right hemi pelvic bone generated in ABAQUS 6.10

D. Material Assignment

Pelvic bone is the highly irregular bone in human body it is complex structure composed of cortical and trabecular bone. Thus it is difficult to assign material along each direction of model. In mimic we use uniform method for realistic material material assignment for each model. Ten materials are assigned to each model and gray values are calculated before realistic material assignment .fig 3 shows material assigned model in mimics.

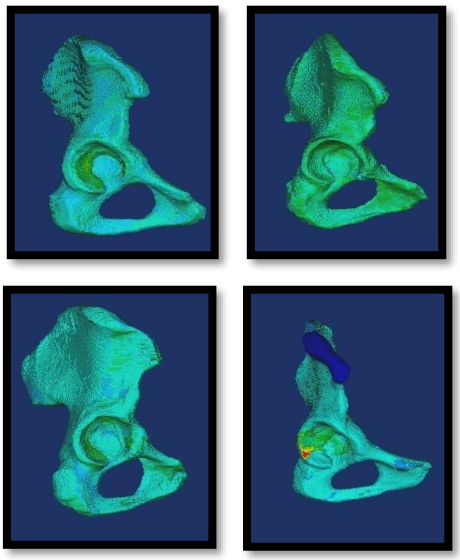

Fig. 3: 3D model shows distribution of material properties or material assignment using mimic

E. Finite Element Analysis

The three dimensional volumetric meshed Finite Element Models of hemi pelvic bone with realistic material assigned are imported into ANSYS v14. The model is nonlinear and curved in all three plane so model is first imported in finite element modeler then transferred to static structural module in ANSYS for FEA.

F. Boundary Condition

In human skeletal system pelvic bone is constrain between sacral and pubic symphysis region . After assigning constrain, loading condition is applied to each model based on previous studies for real body weight which is 63kg ,62kg, 60kg and 56kg respectively and applied four times the real body weight the load is applied considering simple inclination of 30 degree and inclination and antervention both of 30 and 20degree. The boundary condition applied to pelvic is shown in fig 4

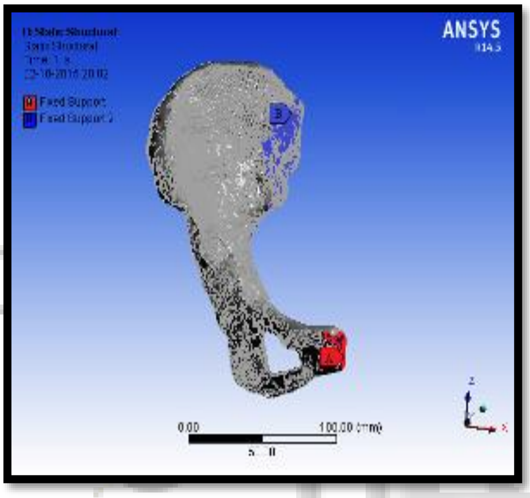

Fig. 4: 3d model shows boundary condition of pelvic bone using ANSYS

III. RESULTS

the equivalent vonmises stress and maximum principal stress is evaluated in the three dimensional Finite Element Analysis shown in fig 5 and fig 6. total deformation and safety factor is obtained throughout the pelvic for all four model of 36 yrs 62 kg,40 yrs 60 kg, 40 yrs 56kg and 45 yrs63kg human female patient. And percentage variation obtained is shown in figure.

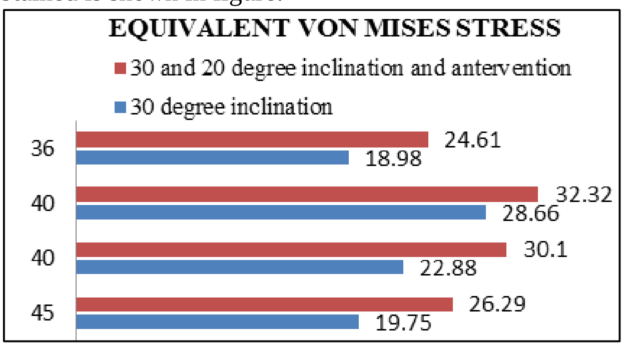

Fig. 5: Equivalent Von Mises Stress

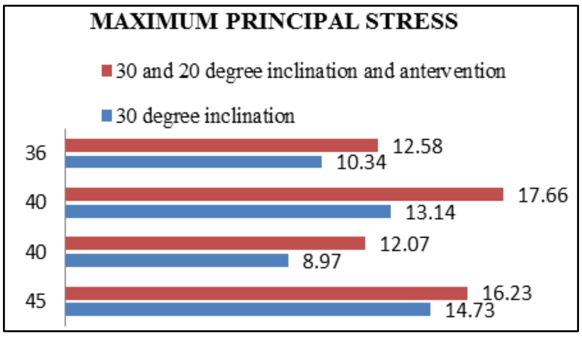

Fig. 6: Equivalent Von Mises Stress And Maximum Principal Stresses For All Model Considering Simple Inclination Of 30 Degree And Both Inclination Of Degree And Antervention 30 And 20 Degree

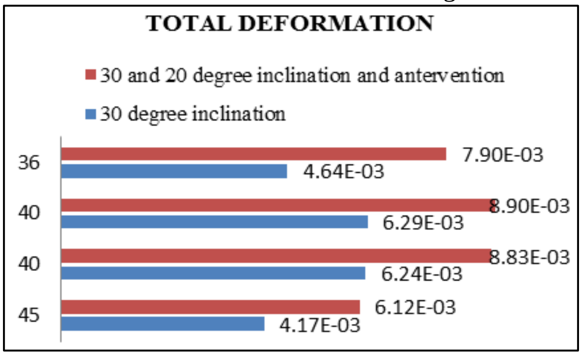

Fig. 7: Total Deformation

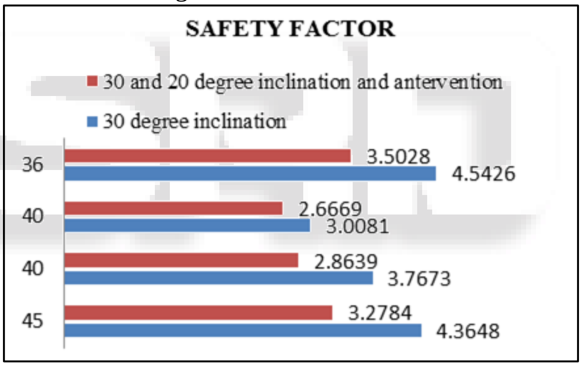

Fig. 8: total deformation and safety factor for all model considering simple inclination of 30 degree and both inclination of degree and antervention 30 and 20 degree

S.No	Model 1	Model 2	Model 3	Model 4
Vonmises Stress	24.89%	23.98%	11.34%	22.89%
Maximum Principal Stress	9.20%	25.68%	25.61%	17.80%
Total Deformation	31.92%	29.32%	29.31%	41.33%
Safety Factor	24.89%	23.98%	11.34%	22.89%

Table 1: Comparision of Results

IV. DISCUSSION AND CONCLUSION

For noninvasively clinical application such as fracture risk, prosthesis design and bone remodelling the development of subject specific finite element (FE) model using CT scan data is very powerful tool because of their high potential in clinical practice. Automatic mesh generation may provide good and quick response to geometrical representation of bones. Still due to variation in material property of different bone it become difficult for FE model reliable for clinical practice. On applying joint reaction force which is four times the real body weight on right hemipelvic bone under study at different loading angle at simple inclination of 30 degree and both antervention and inclination angle at 20 degree and 30 degree following conclusion is investigated.

a) Equivalent vonmises stress increases with increase in antervention from 0 degree to 20 degree considering constant inclination of 30 degree.
b) Maximum principal stress increases with increase in antervention from 0 degree to 20 degree considering constant inclination of 30 degree.
c) Total deformation increases with increase in antervention from 0 degree to 20 degree considering constant inclination of 30 degree.
d) Safety factor decreases with increase in antervention from 0 degree to 20 degree considering constant inclination of 30 degree
e) Percentage variation for vonmises stress is more for model 1 (female age 45yrs 63kg)
f) Percentage variation for maximum principal stress is least for model 1 (female age 45yrs 63kg).
g) Percentage variation for safety factor is more for model 1 (female age 45yrs 63kg).

REFERENCES

[1] M. Dalstra , R. Huiskes , L. Van Erning "Development and Validation of a Three-Dimensional Finite Element Model of the Pelvic Bone" , Journal Of Biomechanical Engineering ASME 1995

[2] M. Dalstra , R. Huiskes , A. Odgaard, L. Van Erning , "Mechanical and textural properties of trabecular bone", Pergamon Press Ltd 1993 (Netherlands)

[3] M. Dalstra and R. Huiskes , "Load transfer across the pelvic bone". J Biomechanics Pergamon 1994 Elsevier Ltd. (Netherlands)

[4] S. Majumder, A. Roychowdhury, S. Pal ," Variation of stress in pelvic bone during normal walking ,considering all active muscles". Trends Biomater. Artif. Organs. Vol. 17(2) pp 48-53(2004).

[5] T Quesnel ,P M Gueritey and G P Gonon , Journal On Clinical Anatomy (France) Springer -Verlag (1995) 249-253

[6] Fred J.Molz ,Paul D.george, Lani S Go, Martha W.Bidez, Albert I King , Jorge Alonso ,(Birmingham)" Simulated automotive side impact on the isolated human pelvis", Society of automotive engineer (1997).

[7] Benoit BESNAULT Franqois LAVASTE Laboratory of Biomechanics, Herd GUILLEMOT CEESAR, European Center of Safety and Risk Analysis, Stphane Robin , Jean Yves Le ,"Morphometric study of the human pelvis", Coz Paper Number 98-S9-P- 18 .

[8] John M. Dawson , Boris V. Khmelniker, Mark P. McAndrew , "Analysis of the structural behaviour of the pelvis during lateral impact using the finite element method ", 1998 Elsevier Science Ltd. All rights reserved. Accident Analysis and Prevention 31 (1999) 109–119.

[9] Eric Song, Laurent Fontaine, Xavier Trosseille, Hervé Guillemot, "Pelvis Bone Fracture Modeling In Lateral Impact", LAB PSA Peugeot-Citroën Renault, France, Paper Number 05-0247

[10] Dr Akram Abood Jaffar, Dr. Sadiq Jaffer Abass, Huma Mohamed Abdullah, "Stress analysis of the hip bone", Al-khwarizmi engineering journal pp 61-72 Vol no 1 2005

[11] Vickie B. Shim, Rocco P. Pitto, Robert M. Streicher,Peter J. Hunter, Iain A. Anderson, "The use of sparse CT database for autogenerating accurate FE model of femur and pelvis" , Journal of Biomechanics 40 (2007) 26–35

[12] Zuoping Li, Jong-Eun Kim, James S. Davidson, Brandon S. Etheridge, Jorge E. Alonso, Alan W. Eberhardt," Biomechanical Response of the Pubic symphysis in lateral pelvic impact. A Finite Element Study". Journal of Biomechanics 40 (2007) 2758–2766. Elsevier Ltd.

[13] Vickie B, Rocco P, Peter J. Hunter, A Robert, M.Streicher ."Development and Validation of Patient-Specific Finite Element Models of the Hemipelvis Generated From a Sparse CT Data Set" (New Zealand) Journal of Biomechanical Engineering ,ASME (OCTOBER 2008), Vol. 130 / 051010-1

[14] Jong-Eun Kim , ZuopingLi , YasushiIto , Christina D.Huber, AlanM.Shih , Alan W.Eberhardt , King H.Yang, AlbertI.King , BharatK.Soni , "Finite element development of a child pelvis with optimization-based material identification", Journal of Biomechanics 42 (2009) 2191–2195.

[15] Omri Lubovsky, David Wright, Michael Hardisty, Alex Kiss, Hans Kreder, Cari Whyne, "Acetabular orientation: anatomical and functional measurement, International Journal of Computer Assisted Radiology and Surgery" (impact factor: 1.48) 08/2011; 7(2):233-40. DOI:10.1007/s11548-011-0648-3.

[16] John, P. Wysota (Poland), "Computer aided diagnostic of risky state in human pelvic bone", Journal of achievements in material and manufacturing engineering (JAMME) 2011

[17] Antony John, Piotr Wysota (Poland)," Numerical simulation of osteoporotic change in Human Pelvic Bone" , Congress of European society of Biomechanics -ESB2012.

[18] J. Awrejcewicz, J. Mrozowski1, S. Młynarska, A. Dąbrowska-Wosiak, B. Zagrodny, S. Banasiak and L.V. Yakushevich, "Modelling and Simulation of Biomechanical Systems - An Orbital Cavity, a Pelvic Bone and Coupled DNA Bases, Numerical Analysis"-Theory and Application Edited by Prof. Jan Awrejcewicz ISBN 978-953-307-389-7 Hard cover, 626 pages Publisher InTech.

[19] Elisabetta M. Zanetti, Cristina Bignardi, Alberto L. Audenino, "Human pelvis loading rig for static and dynamic stress analysis", Acta of Bioengineering and Biomechanics Original paper Vol. 14, No. 2, 2012.

[20] P. Patel, A.T.M. Phillips, "Structural Meso-Scale Bone Remodelling of the Pelvis", July 2012 Department of Civil and Environmental Engineering, Imperial College London, Exhibition Road, London SW7 2AZ, UK

[21] Rajesh Ghosh, Bidyut Pal, Debatri Ghosh And Sanjay Gupta, "Load Transfer Across The Pelvic Bone During Normal Walking", Department of Mechanical Engineering, Indian Institute of Technology Kharagpur, West Bengal, India.

[22] Syed Zameer , Dr. Mohamed Haneef , Mohammed Mohsin Ali H, "Evaluation of Contact Stress Distribution of Hip Joint Model using Finite Element Method" Journal of Engineering Research & Technology (IJERT) Vol. 3 Issue 6, June - 2014 ISSN: 2278-0181).
[23] Jiong Mei, Ming Ni, You-Shui Gao and Zhi-Yuan Wang, "Femur performed better than tibia in autologous transplantation during hemi pelvis reconstruction", Mei et al. World Journal of Surgical Oncology 2014, 12:1
[24] Norio Imai , Yoichiro Dohmae, Ken Suda, Dai Miyasaka, Tomoyuki Ito, Naoto Endo, "Effects of Curved Periacetabular Osteotomy on the Stress Fields of the Pubic Rami and Ischium: A Finite Element Model Analysis", Open Journal of Orthopedics, 2013, 3, 199-203
[25] Cigdem Icke, Juergen Koebke , "Normal stress pattern of the pubic symphysis", Copyright © 2014. Anatomy & Cell Biology Accepted October 14, 2013
[26] Ching Lun Tai , Chi Li Lin, Hsien Wen Wang , De Mei Lee, Pang Hsing Hsieh, "Stress Distribution Of A Modified Periacetabular Osteotomy In Treatment Of Dysplastic Acetabulum", Journal Of Medical And Biological Engineering, 31(1): 53-58.
[27] Damien P. Byrne, Kevin J. Mulhall and Joseph F. Baker," Anatomy & Biomechanics of the Hip", The Open Sports Medicine Journal, 2010, 4, 51-57.
[28] S. Majumder, A. Roychowdhury (India), "A Finite Element Study on the Behavior of Human Pelvis under Impact through Car Door", Dept. of Applied Mechanics, Bengal Engineering College (Deemed University), Howrah, India. School Bio-science and Engineering, Jadavpur University, Kolkata, India page no 197-203
[29] Ani Ural, Deepak Vashishth (USA) "Cohesive finite element modeling of age-related toughness loss in human cortical bone", Journal of Biomechanics 39 (2006) 2974–2982, Elsevier Ltd.