Three Dimensional Finite Element Analysis of Pelvic Bone

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

Abstract - **Human hips based biomechanical studies for the analysis of two dimensional pelvic radiographs have turned out to be a reasonable compromise between the measurement accuracy and the feasibility in clinical setting. Through advances in three dimensional imaging with MRI and CT the ability for the estimation of resultant hip force with three dimensional imaging of the hip weight-bearing surface, contact hip stress for a given activity/body position has been greatly improved. Bioengineering concerns many important problems apply to human body. The pelvic joint and its correct working is one of them. The pelvic bone is one of the most important supporting elements in human pelvic joint but it is liable to suffer an injury. Very often before and after operations the knowledge of the stress and strain distribution in the pelvic bone is needed, our objective was to use a simple three-dimensional (3-D) mathematical model to evaluate the vonmises stress distribution, maximum principle stress distribution and fatigue for the peak contact force under different physiological loading condition, for the purposes of which right human hemi pelvic bone of four female patient age group between 35 to 45 is taken into account. Which shows that major weight bearing area sacro-iliac joint and the pubic symphysis.**

Index Terms - **Finite Element Method, pelvic bone, biomechanics, MIMICS**

I. INTRODUCTION

Pelvic bone is one of the most important weight bearing element of human skeleton. The bony pelvis is a complex structure with variations in geometrical and mechanical characteristics, it is a ring like structure form by the combination of two hemi pelvic which is symmetric to each other and sacral region. Each hemi pelvic is formed by three bone ileum, ischium and pubic symphysis having contribution 40% of the acetabulum, 40% of ischium and 20% of the pubis . In the skeletally immature pelvic these three bones are separated by the tri radiate cartilage , fusion of this starts to occur around the age of 14 – 16 years and is complete usually by the age of 23 [[12]]. The structure of pelvic is sandwiched type compose of thin layer of cortical bone in between which trabecular bone is exist. Due to the 'sandwich' behavior of the pelvic bone, stresses in the cortical shell are about 50 times higher than in the underlying trabecular bone [[19]]. The orientation of an acetabulum or an acetabular prosthesis may be described by its inclination and antervention. The anatomical, operative, and radiographic definitions of antervention and inclination are all different [[11], [2]]. Basic load transfer and stress distributions under physiological loading conditions shows that major part of the load is transferred through the cortical shell and principal areas of support for the pelvic bone are the sacro-iliac joint and the pubic symphysis, this caused the primary areas of load transfer to be found in the superior acetabular rim [[19], [28]]. Due to both shape and structural architecture, the mechanics of pelvic bone are also complex [[20]]. Biomechanics of hip was carried out, for which force exerted during stance phase was analyzed [[29], [22], [24]]. A common tool in numerical simulation is the finite element method (fem), which can be applied in various ways, e.g. in structure mechanics, thermodynamics or acoustics [[13]]. Mathematical estimation peak stress in the hip joint and other relevant radiographic and biomechanical parameters were assessed [[17], [6]]. For mechanical properties of pelvic bone mechanical testing was used to obtain young's module and Poisson's ratio in three orthogonal direction [[18]] load transfer across the pelvic bone during normal walking was observed for which a realistic FE model of the pelvic bone has been developed, which can be used in combination with the musculoskeletal model of forces to numerically predict stress distributions across the pelvis [[25], [4], [30]]. Static and dynamic three-dimensional finite element analysis of pelvic bone was analyzed ,the complex shape of the human pelvic bone was successfully imaged and modeled using subject-specific finite element FE processing [[21],[5]]. Parametric finite element studies of the human pelvis was carried out to determine the effect of both force magnitude [[16], [9]]. The numerical model is prepared on the ground of the geometrical data from 3d scanning or ct, and various load cases in numerical model of pelvic bone with artificial acetabulum is perform in which the layer structure of bone tissues is taking into account [[26],[1],[7]]. Boundary conditions for the FE model were applied to the pubis and the sacro-iliac joints [[3], [10]]. Loading according stance phase which is four times the weight of body is applied through the femur head or acetabulum of pelvic bone [[22], [4], [23] and [27]]. And the stress distribution of the intact pelvic model across the pelvis is obtained [[25], [15], [14], and [8]].

II. MATERIAL AND METHOD

The pelvic joint was visually screened for large-scale osteoarthritis prior to the study, a better understanding of the mechanics for the entire pelvis could lead to improved implant designs, surgical approaches, diagnosis, and may present the framework necessary for preoperative Surgical planning for design and replacement of pelvic bone of hip joint. In order to define the solid geometry of the pelvic bones an anatomic model of the left hemi pelvis was constructed, for which geometrical data of real human pelvic bone of four normal female patient of different age group is obtained from CT scan in form of DICOM files. A stack of images can be loaded into the software called Mimics which is key for converting anatomical data from images to 3D model. Process of converting 2d data into 3d model is called segmentation.

2.1. Image Acquisition

The CT/MRI data of total pelvic of normal individual female patients of 36 year, 40 year, 45 year and 40 year are collected in form of digital imaging and communication in medicine (dicom) file. This dicom is obtained from CT scanning of patient using GE ultra-high resolution multislice CT scanner (16 slice) (GE MEDICAL SYSTEM / BRIGHT SPEED 120KV 335 32mAs) 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 pixel. A DICOM image consists of a list of data elements (so-called attributes) which contain a multitude of image related information, including items such as name, ID, etc., and also one special attribute containing the image pixel data. Pixel data can be compressed using a variety of standards.

2.2. Image Segmentation

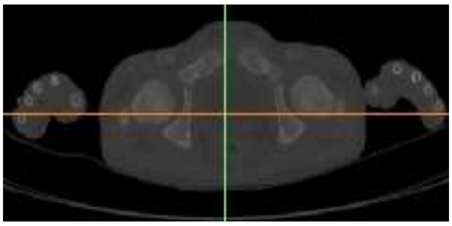


Fig.1 CT image of normal individual

Image segmentation is the process of partitioning a digital image into multiple segments. Three dimensional model of right hemi pelvic of four individual female patients are created: Model 1 of 45 years old and 63 kg female, Model 2 of 36 year old and 62 kg female, Model 3 of 40 year old and 60 kg female and model 4 of 40 years old and 56 kg female. The medical images coming from CT or MRI scanners consist of grayscale information, Mimics allows the user to create models based on the gray values (Hounsfield units in CT images) within these images. The fig 1 shows the CT image of normal individual of total pelvic acquire from DICOM file. A gray value is a number associated with an image pixel defining the shade (white, gray, or black) of the pixel. By grouping together similar gray values, the image data can be segmented, and models created. This type of segmentation is called thresholding and yields accurate models. Thresholding is the simplest method of image segmentation, it is used to create a first definition of the segmentation object, and from grayscale image binary image is created. Thresholding based on Hounsfield scale was used to separate each part of the pelvic bones tissues volume. The extracted bone tissue is put into a mask of volume 801479.5646 mm^3, 670594.8354 mm^3, 221865.4279 mm^3 and 458961.5170 mm^3 respectively, and number of pixels 1274618, 982404, 349389 and 855566 respectively. These pixels in the masks are modified using various tools successively using edit mask, region growing, and calculation of 3D mask. For a better visualization of the internal boundaries in the density masks, polylines were generate editing mass, polyline calculation, morph metric operation are the other tool of segmentation to obtain perfect geometry for three dimensional model. The three dimensional model of right human pelvic bone for all four female patient is created and properties of different models are

Model 1 having volume 155738.02mm^3, surfaces 91359.80 mm^2, triangle 117056, points 57994.
Model 2 having volume 117281.44 mm^3, surfaces 69979.91mm2, triangle 88544, points 43904
Model 3 having volume 104083.05 mm^3, surfaces 87052.28 mm^2, triangle 120868, points 59104
Model 4 having volume 116245.0 mm^3, surfaces 76064.24mm2, triangle 97438, points 48267
Fig 2 shows the different three dimensional model in mimics.

Fig.2 3-D models of all four female patient

2.2 Creation of FE model

To make the model simplify their use the process to reduce the number of faces. Three dimensional object can be remeshed using the remesh module. This is needed in order to raise the quality of the triangles so that the preprocessor of an FEA package can build a tetrahedron mesh from them. Surface mesh of equilateral triangle is generated in automatic remesh operation. The various tool used in remeshing are as follows.

- Local operation
- Auto remesh

- Triangle reduction
- Quality visualization
- Mark triangle
- Self-intersection test
- Smoothing

The properties of different surface remeshed model are.
Model 1 having surfaces 58395.26mm^2, triangle 76026, points 38021.
Model 2 having surfaces 48003.23mm2, triangle 55248, points 27644
Model 3 having surfaces 75171.03mm^2, triangle 103176, points 51602
Model 4 having surfaces 70822.66mm2, triangle 89192, points 44604
The surface meshed models of right hemi pelvic bone is imported in ABAQUS 6.10 , and these surface mesh is converted into volumetric mesh by using edit mesh tool by which mesh is converted from tri to tet (tri is for triangular type and tetra is for tetrahedral element). The volumetric meshed three dimensional finite element model of right hemi pelvic bone obtained from ABAQUS is consist of 727896, 530684, 899442 and 626799 elements and 138783 ,101221, 172964 and 124700 nodes respectively volumetric mesh obtained in ABAQUS is shown in figure 3

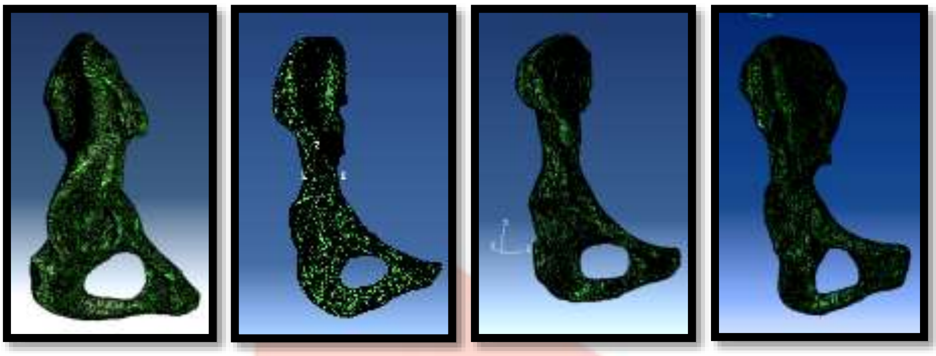


Fig.3 Volumetric meshed models of right hemi pelvic bone generated in ABAQUS 6.10

1.3 Material Assignment

The anatomical structures of bone have varying material properties throughout the structure. After loading a volumetric mesh, the FEA calculates an appropriate Hounsfield value for each element of the mesh based on scanned images. Mimic uses assigning materials with uniform method in which the range of Hounsfield units of volume mesh is subdivided into number of equal sized intervals in which each represents a material. Fig4 shows the assignment of material properties for hemi pelvic of different volumetric model in which Distributed material property assignment for an FEA analysis based on the gray values in a CT scan and Coloured visualization of the material properties of the different elements in a section of a pelvic bone is shown.

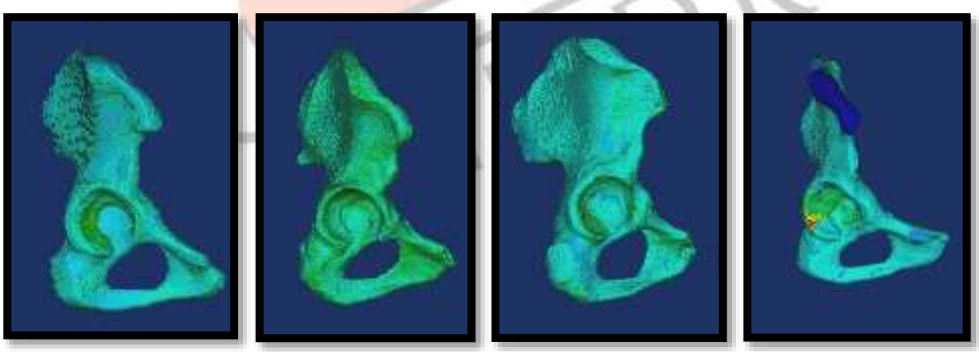


Fig4 3-D model material assignment using MIMICS

1.4 FE Analysis

The three dimensional models of hemi pelvic bone with volumetric mesh and realistic material assigned value are imported in ANSYS v 14.5. 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. Peak contact hip force during the stance phase of the gait cycle (approximately 4 times of BW) at different operational inclination of 30 degree and 45 degree is applied for which acetabulum cup has modeled by means of reaction force. The real body weight for different model are 63kg, 62kg, 60kg and 56kg respectively. Fig 5 shows constrain or boundary condition applied to every model.

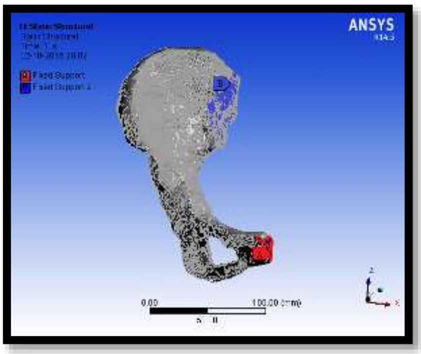


Fig 5 boundary condition applied on pelvic bone using ANSYS.

III. Results

The equivalent von mises stress , maximum principal stress, total deformation and safety factor evaluated in the three dimensional finite element analysis is shown in fig 6, fig 7,fig 8 and fig 9. This biomechanical study also shows the percentage variation in each model due to increase in operational inclination of load application.

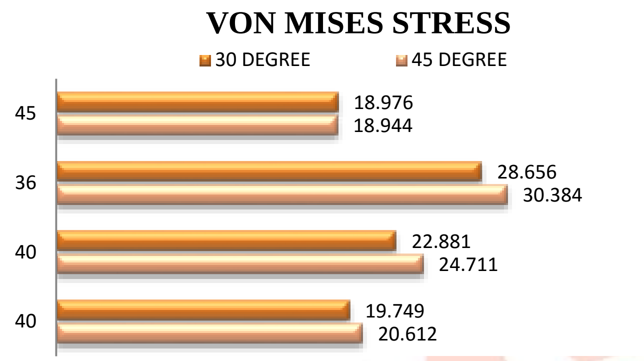


Fig. 6

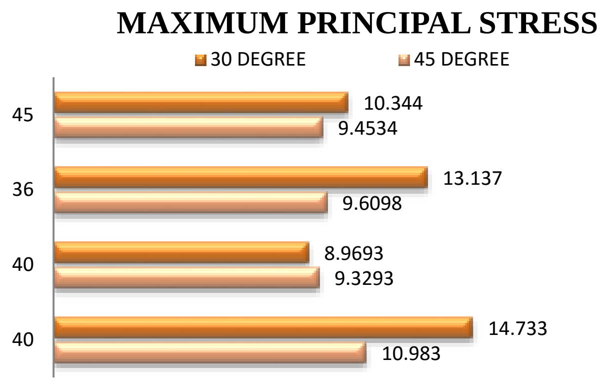


Fig. 7

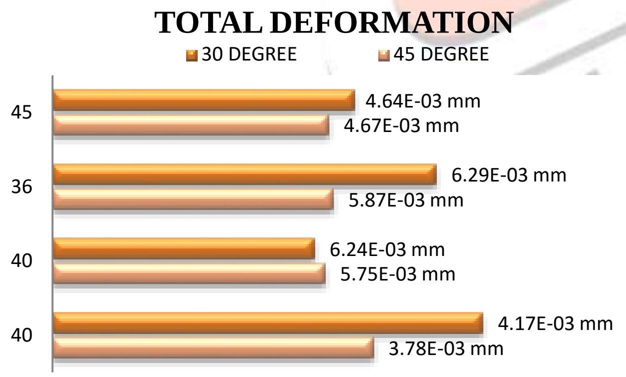


Fig. 8

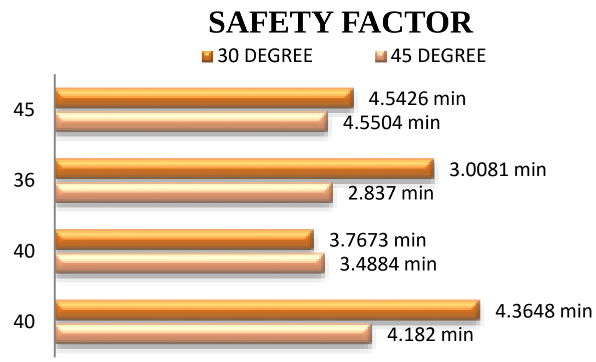
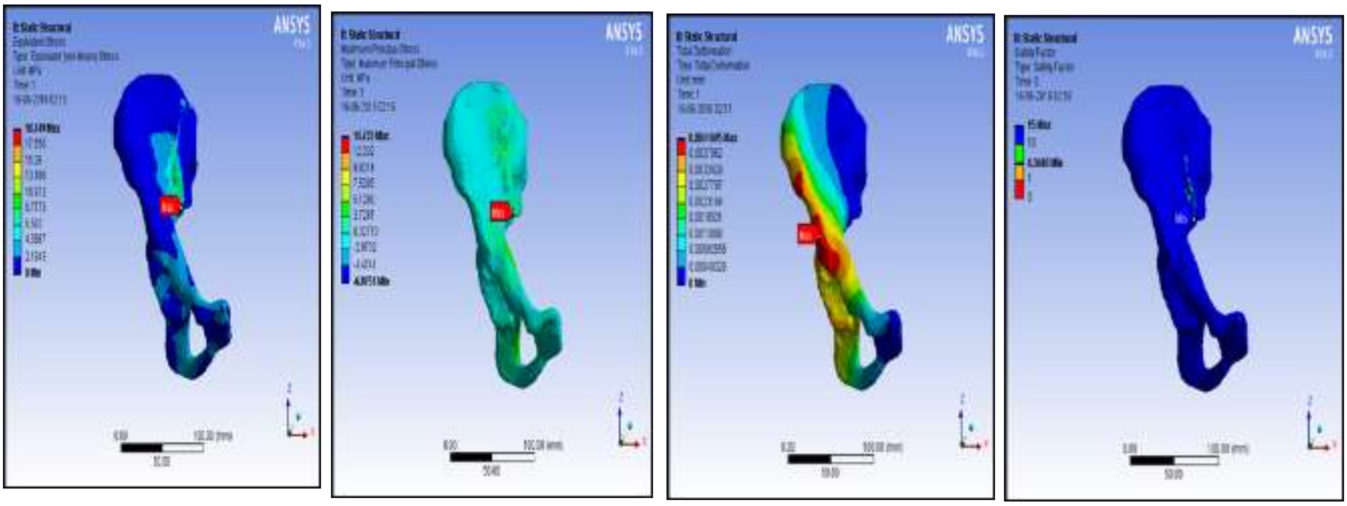


Fig.9

Table 1 Percentage variation due to increase in angle of inclination

	Model 1 40 Yrs 56 Kg	Model 2 40 Yrs 60 Kg	Model 3 36 Yrs 60 Kg	Model 4 45 Yrs 63 Kg
Von Mises Stress	0.17	5.69	7.41	4.19
Maximum Principal Stress	8.61	26.85	3.86	25.45
Total Deformation	0.79	6.77	7.88	9.38
Safety Factor	0.17	6.03	8.00	4.37



IV. Discussion And Conclusion

The geometry of pelvic bone is very complex from the mechanical point of view. By taking this into account two and three dimensional axisymmetric finite element models were developed for analysis. The model was analyzed according to the anatomical region of study. The zone of highest vonmises stresses varied superior pubic ramus and center of ilium. It is concluded that the presently developed approach for subject-specific FE modeling of the pelvis has the ability to predict behaviour of pelvic bone under physiological loading condition.

a) The equivalent von mises stress increases with increase in inclination angle.
b) Maximum principal stress decreases with increase in inclination angle.
c) Total deformation decreases with increase in inclination angle.
d) Safety factor also decreases with increase in inclination angle.

V. Acknowledgements

The authors would like to thank Dr. (Mrs) Shobha Katheria, Principal Medical Officer, Ordanance Factory Hospital, and Itarsi M.P. India Dr. Pushpraj Bhatele, for providing medical imaging data for our work.

VI. References

[1] A. John, P. Orantek, J.J. Telega, *The Load Cases In Numerical Model Of Pelvic Bone With Artificial Acetabulum , XXI ICTAM, 15-21 August 2004*, Warsaw, Poland ,Mechanics of 21st Century - ICTAM04 Proceedings.

[2] A. John, P. Orantek, J.J. Telega, *The Load Cases In Numerical Model Of Pelvic Bone With Artificial Acetabulum , XXI ICTAM, 15-21 August 2004, Warsaw, Poland ,Mechanics of 21st Century - ICTAM04 Proceedings*

[3] A.T.M. Phillip, P. Pankaj, C.R. Howie, A.S. Usmani, A.H.R.W. Simpson, *Finite element modeling of the pelvis, Inclusion of muscular and Ligamentous boundary condition Medical Engineering & Physics 29 (2007) 739–748* Elsevier Ltd.

[4] Ahmet C. Cilingira, Vahdet Ucara, Recep Kazana, *Three-Dimensional Anatomic Finite Element Modelling of Hemi-Arthroplasty of Human Hip Joint, Trends Biomater Artif organ Vol 2007.*

[5] Andrew E. Anderson (USA), *A Subject-Specific Finite Element Model of the Pelvis: Development, Validation and Sensitivity Studies, Journal on biomechanical engineering -2004.*

[6] Andrew E. Anderson (USA), *A Subject-Specific Finite Element Model of the Pelvis: Development, Validation and Sensitivity Studies, Journal on biomechanical engineering -2004.*

[7] Antun Alamon, Tomislav Alamon, Davorin and Ana Jo-Osvati, *Morphological Characteristics of the Acetabulum, Coll. Antropol. 28 Suppl. 2 (2004) 221–226 UCD 572.781:611.718* Original scientific paper.

[8] B. Mavcic, B. Pompe, V. Antoli, M. Daniel, *Mathematical estimation of stress distribution in normal and dysplastic human hips, B. Mavcic et al. / Journal of Orthopaedic Research 20 (2002) 1025–1*

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

[10] 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.

[11] D W Murray ,*Definition and measurement of acetabular orientation journal of bone joint and surgery 1993 75B :228-32*

[12] Damien P. Byrne, Kevin J. Mulhall and Joseph F. Baker, Anatomy & Biomechanics of the Hip, *The Open Sports Medicine Journal*, 2010, *4*, 51-57

[13] Daniel Kluess, Jan Wieding, Robert Souffrant, Wolfram Mittelmeier and Rainer Bader, *Finite Element Analysis in Orthopaedic Biomechanics Finite Element Analysis* Edited by David Moratal, Publisher Sciyo, InTech Europe University Campus, *ISBN 978-953-307-123-7 Hard cover, 688 pages*.

[14] Dr Akram Abood Jaffar, Dr. Sadiq Jaffar Abass, Huma Mohamed Abdullah, *Stress analysis of the hip bone*, Al-khwarizmi engineering journal pp. 61-72 Vol no 1 2005
[15] 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* .
[16] J. W. Plummer, M W Bidez, J. Alonso, *Parametric Finite Element Studies of the Human Pelvis: The Influence of Load Magnitude and Duration on Pelvic Tolerance During Side Impact* , Society of Automotive Engineers, Inc (1996)
[17] Jun Iwamoto and Tsuyoshi Takeda, *Stress fractures in athletes: review of 196 cases, J Orthop Sci (2003) 8:273–278.*
[18] M. Dalstra , R. Huiskes , A. Odgard, L. Van Erning , *Mechanical and textural properties of trabecular bone* , Pergamon press ltd 1993 (Netherlands)
[19] M. Dalstra and R. Huiskes, *Load transfer across the pelvic bone.* J biomechanics Pergamon 1994 Ltd. (Netherlands) Elsevier
[20] 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.*
[21] M. S. El-Asfoury, and M. A. El-Hadek, Static and Dynamic Three-Dimensional Finite Element Analysis of Pelvic Bone, International Journal of Engineering and Applied Sciences (2009).
[22] Mark Karadsheh, Hip Biomechanics - Basic Science - Orthobullets.com Topic updated on 06/29/13 2:38pm
[23] 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*
[24] P. K. Commean, K. E. Smith, M. W. Vannier, B. A. Szabot and R. L. Actist , *Finite element modeling and experimental verification of lower extremity shape change under load , Journal on biomechanics Vol 30 Elsevier Ltd. (1997) 531-536*
[25] 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.
[26] S K Senapati & S.Pal, Finite *Element Analysis: An Effective Tool for Prostheses Design, Trends Biomater. Artif. Organs. Vol. 17(2) pp 141-148 (2004)*
[27] 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
[28] 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).*
[29] T Quesnel ,P M Gueritey and G P Gonon *, journal on clinical anatomy (France) Springer -Verlag (1995) 249-253*
[30] Vickie B, Rocco P, Peter J. Hunter, A Robert, M.Streicher *Development and Validation of Patient-Specific Finite Element Models of the Hemi pelvis Generated From a Sparse CT Data Set (New Zealand)* Journal of Biomechanical Engineering *,ASME (OCTOBER 2008), Vol. 130 / 051010-1.*