International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:3, No:9, 2009

Static and Dynamic Three-Dimensional Finite Element Analysis of Pelvic Bone

M. S. El-Asfoury, and M. A. El-Hadek

Abstract**—**The complex shape of the human pelvic bone was successfully imaged and modeled using finite element FE processing. The bone was subjected to quasi-static and dynamic loading conditions simulating the effect of both weight gain and impact. Loads varying between 500 – 2500 N (~50 – 250 Kg of weight) was used to simulate 3D quasi-static weight gain. Two different 3D dynamic analyses, body free fall at two different heights (1 and 2 m) and forced side impact at two different velocities (20 and 40 Km/hr) were also studied. The computed resulted stresses were compared for the four loading cases, where Von Misses stresses increases linearly with the weight gain increase under quasi-static loading. For the dynamic models, the Von Misses stress history behaviors were studied for the affected area and effected load with respect to time. The normalization Von Misses stresses with respect to the applied load were used for comparing the free fall and the forced impact load results. It was found that under the forced impact loading condition an over lapping behavior was noticed, where as for the free fall the normalized Von Misses stresses behavior was found to nonlinearly different. This phenomenon was explained through the energy dissipation concept. This study will help designers in different specialization in defining the weakest spots for designing different supporting systems.

Keywords**—**Pelvic Bone, Static and Dynamic Analysis, Three-Dimensional Finite Element Analysis.

I. Introduction

THE human pelvis bone is one of the most important support elements in musculoskeletal system. It is a ring-like structure that connects the upper part of the human body with the lower extremities. Finite element (FE) method has been widely used to study the pelvis mechanics during past decades under different loading and boundary conditions [1]. These earlier computational models mainly considered the bony structure of the human pelvis, and the soft tissues in pelvis articular joints; however, were often simplified or not modeled; the deformation characteristics of the soft tissues, therefore, were not emphasized [2]. Recent advance in techniques of image processing and mesh generation, constitutive modeleling of soft tissues, computational biomechanics, and FE method make it possible to generate a realistic FE model of the human pelvis, which includes not only cortical and trabecular bones but also the soft tissues in the pelvic articulations and the surrounding ligaments.

The kinematics of the articulations and biomechanical properties of the soft tissues are important [3]-[8] for further understanding the relative motion of the two in nominates and the injury mechanism the pelvis sustains during side impact crashes. When an external force such as weight-bearing or muscle action is applied to bone, it will be deformed from its original shape, and an internal stress will be generated to counter the applied force. This internal stress is equal in magnitude but opposite in direction to the external force applied. The term “strain” is used in bone biomechanics to describe the deformation in shape and size that bone undergoes under an external force [9]. More impact tests on pelvis have been performed and published ever since. Renaudin [10] performed 10 cadaver tests with a circular butt flat impactor of 23.4 kg. Tolerance in terms of impact force revealed to be higher. Bouquet [11] performed cadaver tests also with an impactor of 23.4 kg. The impact surface was nevertheless a rectangular rigid plate of 200x100 mm2. They showed a lower tolerance level in terms of impact force: around 8 KN. Bouquet performed more cadaver tests but with a larger impact surface (200x200 mm) in order to include the contribution of iliac wing. Impactor mass and impactor velocity were designed in such a way so that they can examine which one, between mass and velocity, is dominant for a given energy level. In fact they found that to represent car crashes, the impacting masses should be lower than the famous 23.4 kg impactor, and considered essential to know the pelvis behavior in new impact conditions. Based on their new cadaver tests, they concluded that for a given impactor energy, neither its mass nor velocity seemed to be dominant. Side impact dummies were evaluated with respect to some configurations of above cadaver tests.

References [13]-[12] developed a spring mass model of pelvis for lateral impact. However, it was limited to give only a global response in terms of force, displacement, or acceleration, in monoaxial conditions. Cesari [14] developed a finite element model of pelvis. Considering that the trabecular bone had a low influence in terms of overall stiffness of the pelvis [15], they represented pelvis bone by only shell elements, corresponding to the external surface of the structure. The model was designed from a metallic model. Moreover, thicknesses from 1 to 4 millimeters, measured on experimented pelvis, were attributed to the shell elements. Nodal masses were distributed to correspond to the global

M. S. El-Asfoury, Graduate Student, in the department of Mechanical Design & Production Department, Faculty of Engineering, Port-Said, Suez Canal University, Egypt.
M A. El-Hadek, Assistant Professor in the department of Mechanical Design & Production Department, Faculty of Engineering, Port-Said, Suez Canal University, Egypt. Phone: +2 010 827-1778, e-mail: m.elhadek@scuegypt.edu.eg. The author to whom all correspondence should be addressed.

International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:3, No:9, 2009

characteristics of a pelvis. The Young's modulus in this model was low, around 3000 MPa [14]. Static tests [16] were first conducted under side loading conditions, in order to validate this model. Hartemann [17] improved this model by adding geometrical parameters to adapt it to different tested bones, using a kriging technique. Plummer [18] proposed a modified version of a model of Hartemann, built from Computerized axial Tomography (CT) data scan slices, which aimed at the study of pelvis fracture etiology, in the context of automotive side impact conditions. Nevertheless, this model did not represent a whole pelvis: a coxal bone was modeled, but the sacrum and the contralateralilium were not taken into account. Finally the acetabulum was fitted with a hip prosthesis. Dawson [19] proposed a model, also dedicated to lateral impacts in the field of car accidents. The model was created from 74 CT scan slices, and distorted by scale factors. The two coxal bones and the sacrum were built by 8-node elements, and connected to each other by 32 springs for the sacro-iliac joints and 8 springs for the pubic symphisis. Joint properties were established from the literature [20]-[24]. Bone characteristics were given element by element, from CT scan density levels, and range from 250 to 1500 MPa for the trabecular bone Young's modulus. It should be reported that the mesh was too course (1511 8-node elements and 3769 nodes) relatively to capture the bone behavior.

Through the literature survey, the simulation of the pelvic bone is currently lacking, therefore, the objective of this research is to image and model the pelvic bone. The pelvic bone will be subjected to static loads varying between 500 – 2500 N (~50 – 250 Kg of weight) which simulate the effect of weight gain. This range was according to the most common average weight of human body. The pelvic will subjected to dynamic loadings with different analysis. Also, Free fall dynamic load under 1 and 2 meter to simulate the falling accident of tall humans, who fall under these heights. Finally, forced side impact analysis at two different velocities (20 and 40 Km/hr) to simulate impact accidents. The forced side impact simulate the car accident with the selected velocity, It recommended that impact peak was done after break which remove 60% of the velocity, so the rang of velocity selected in analysis was suitable. The loading conditions were selected to help designers in different specialization in defining the weakest spots for support.

II. Benchmark analysis

It was important to start this study defining a point of reference from which measurements can be made. This analysis was used as reference of our FE and was validated based on a comparison between the computational simulations and closed form solution. The three FE numerical models were calculated under same Boundary Conditions (BC's) as circular plate with clapped edges and uniform distributed load were used exactly as the BC's for the closed form solution. When comparing between these models and the closed form solution, it was found that the best agreeing results could be found between the model with 30 element division (solid 186) mapped mesh as presented in Fig.1, the agreement were found to be 98.1%.

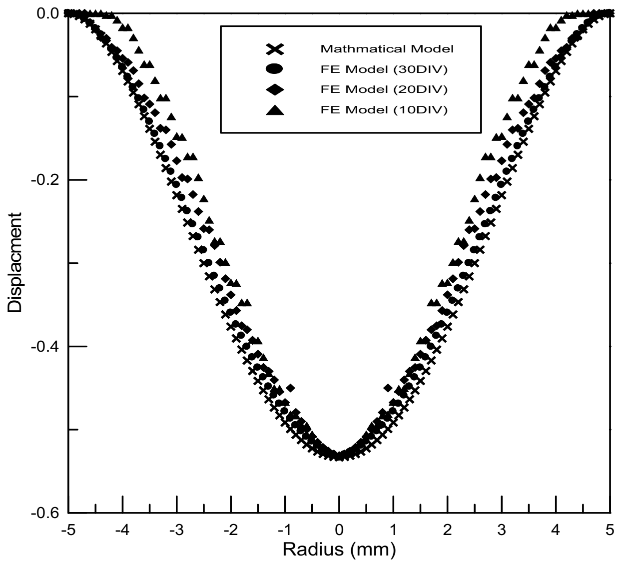


Fig. 1-a The relationship between position and out of plane deformation (deflection) of the FE models with different mesh options and the mathematical closed form solution results.

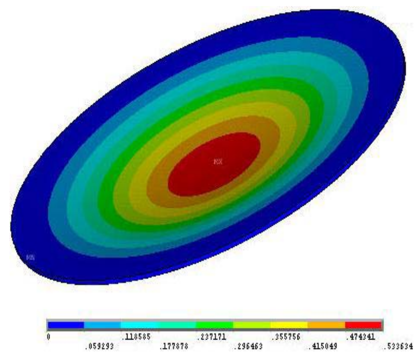


Fig. 1-b The 30 division brick element model results.

III. Pelvic Bone FE Model Construction

To construct a geometrical accurate 3D pelvis model, a Computerized axial Tomography (CT) scan data of the pelvic bone were acquired. The CT data are a series of two dimensional (2D) high resolution x-ray images that displays a density map of the scanned pelvis. Stacking these images can create a 3D representation of the scanned model. Differentiation of tissues in CT scan is accomplished through contrast segmentation, and the grayscale value of each voxel is determined solely by tissue density. The CT data is usually measured by Hounsfield unit (HU) and calibrated to bone mineral density (BMD). Since different morphological structures do not have a unique HU value, completely automatic segmentation of complex images is generally impossible and, therefore, needs to be performed manually and interactively [25]. A final geometrical model basically consists of nodes, surfaces, and volumetric elements as well as boundary identification number associated with these basic structures. The connectivity between the different tissue sub-regions is important for modeling, so that the image data can be segmented, visualized, and reconstructed [26]-[28]. So the optimal CT image processing program used in our study was required to:

1. Automatically generate 3D finite element model which can be read by FE program, with higher accuracy of model.
2. Automatically convert the apparent bone density information to bone elastic constants and assign them to the meshed 3D finite element model.
3. Automatically generate FE models.

The volumetric mesh obtained and the resulting finite element mesh of the Pelvic was composed of 450,168 nodes and 343,690 elements as demonstrated in Fig.2.

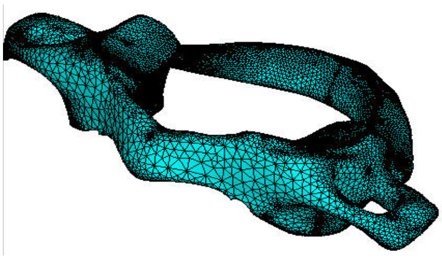


Fig. 2 The composite pelvic bone after meshing with 450,168 nodes and 343,690 elements.

IV. Material Properties

The mechanical properties of bone have been extensively studied. From a mechanical theory perspective, bone is both anisotropic and heterogeneous. In addition, the degree of anisotropy of bone can vary with position so that the material properties of bone are functions of anatomical position [29]. Due to these features, the traditional engineering methods for material property determination are difficult to apply to bone. This reflects the complexity of determination of mechanical properties of bone that are functions of age, sex, and anatomical site. The cancellous bone, there are several studies to determine the relationship between structural density and Young's modulus [31]. The pelvic bone was modeled as linearly elastic, because the bone behaves are more elastically at high strain rate (such as impact) than at low strain rates. Appling two material properties with significantly difference between them over a small area, could lead to delamination and unrealistic results [29], [30], [32], therefore through the literature the pelvic bone was reported to use a model with homogeneously distributed properties. In current analysis the pelvic bone material properties used in the FE simulation are an average Young's modulus, density and Poisson's ratio as 17GPa, 3000Kg/m^3, 0.3, respectively.

V. Finite Element Analysis & Loading

A 3-D Pelvic bone model was imported to ANSYS® finite-elements software. The analysis included static and dynamic. The quest-static loading where applied to the bone for simulating the effect of weight gain in humans on the composite pelvic bone as shown in Fig.3. The Von Misses stresses where computed for loadings varying between 500 – 2500 N (~50 – 250 Kg of weight).

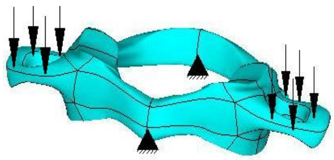
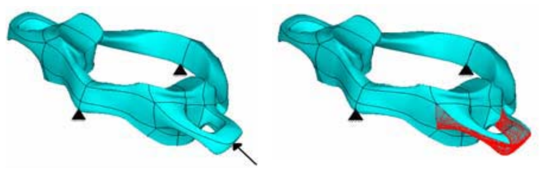


Fig. 3 The boundary condition applied on the Pelvic model on Static model.

The dynamic analysis was applied in two ways, the first one was free falling under gravity, the second was forced side impact simulating cad velocity with 20 and 40 Km/hr. To produce impact loading, the time of force application should be less than the maximum period of natural vibration. Both of dynamic models were applied over a time span of 100µs. Boundary condition of FEA loading was shown in Fig. 4(a,b) to free fall and forced side impact model respectively. The analysis simulating human accidents, so to more accuracy in accident event, the free force was applied in short area (some nodes was selected), otherwise all side area was selected in forced impact dynamic analyses. To predict local bone behavior, the Von Misses stress yield criteria was used for the pelvic bone side to side with the strain energy criteria as (1,2) show. Von Misses stress was chosen because of its convenience to represent general stress distribution within the structure. The analyses were conducted without fixing the model in space (some points was zero displacement specified). The pelvic bone was allowed to move as deformable bodies in any direction but was inhibited by their own inertia. The influences of the upper and lower extremities were neglected, so all safety devices are not an important factor during the main phase of loading.



(a) (b)

Fig. 4 The boundary condition applied on the Pelvic model on (a) free fall dynamic model, (b) forced side Impact model.

$$\sigma_{von} = \frac{1}{\sqrt{2}}\sqrt{(\sigma_x-\sigma_y)^2+(\sigma_y-\sigma_z)^2+(\sigma_z-\sigma_x)^2+6(\tau_{xy}{}^2+\tau_{yz}{}^2+\tau_{zx}{}^2)} \quad (1)$$

$$U = \frac{1}{2E}\begin{bmatrix}(\sigma_x^2+\sigma_y^2+\sigma_z^2)-2\nu(\sigma_x\sigma_y+\sigma_y\sigma_z+\sigma_z\sigma_x)\\ +2(1+\nu)(\tau_{xy}^2+\tau_{yz}^2+\tau_{zx}^2)\end{bmatrix} \quad (2)$$

VI. Results

After validating the model, three types of analysis were considered. The Von-Misses stress was computed for the composite pelvic bone subjected to quasi-static loading ranging from 500 – 2500 N (~50 – 250 Kg of weight). This range was chosen to simulate the human weight gain and its effect of the bone structure itself. The detailed loading with the maximum and minimum stresses computed are presented in table.

TABLE I THE FOUR DIFFERENT STATIC LOADING APPLIED ON THE PELVIC BONE AND THE RESULTS OF MAX., AND MIN. VON MISSES STRESS

Static load (N)	**Max. Stress** (MPa)	**Min. Stress** (MPa)
500	45117.4	48.1763
1000	90234.7	96.3526
2000	180e3	192.7
2500	226e3	240.9

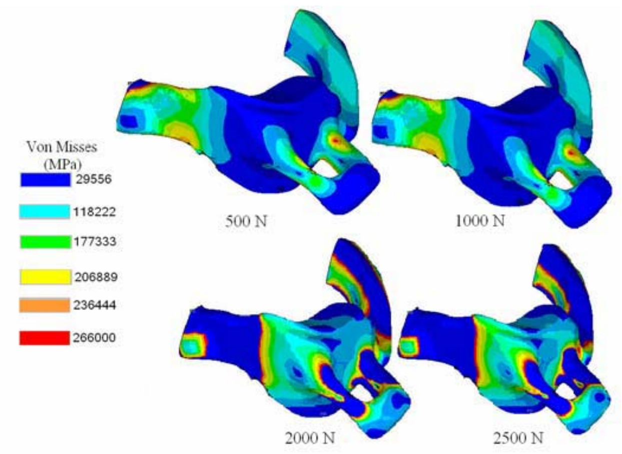
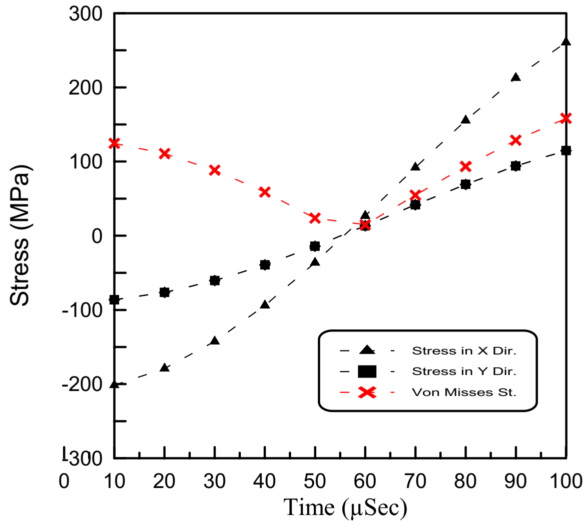


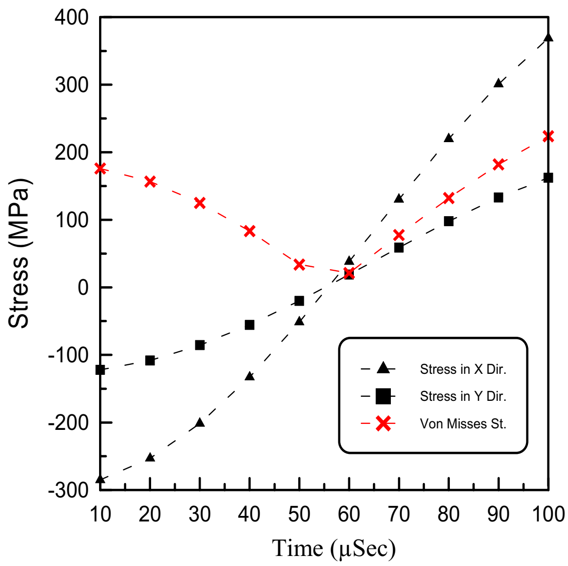
Fig. 5 The results of four cases static loading of pelvic bone under weight gain.

The location of the maximum and minimum von-misses stresses can be observed in Fig. 5. It can be noticed that the nodes were the max. and min. stresses are located in exactly the same node in the four cases above. Also, the increase from the smaller load to the larger one increases in a linearly pattern although the non linear parameters that was used both in the FE analysis as well as the material properties. The pubic was the reference part of stresses increasing, as shown the dark color (dark blue) decreases with the increase of static load, otherwise the light color (light red) which refers to highest stress increases with the decrease of static load.

Pelvic bone under free fall dynamic loading at two heights 1 and 2 meter was modeled and computed the stresses components in x, y, z direction and Von Misses. These heights were extracted from natural conditions acts on the pelvic bone. Fig. 6 show the stresses in MPa variation with time in µsec, stresses in X, Y and Z directions were increase as falling height increase. Where as, the stress in X-direction starts with the smallest stress values and increases with time. The stress in direction of z and y are overlapping in each other, there behavior were inversed with X one. Between 50 and 60 µsec the stresses component were equals, at same time the shear stresses were approximate equal to zero, which effect on strain energy values at that time. Von Misses stress was once in its behavior, decreases with time increase even the half of the time and after that going increase to the end of operation. This behavior will affect the strain energy since it depends on the normal and shear stress as explained in (2). For the free fall of two meter model, the same behavior is observed.



(a) Under 1m free fall

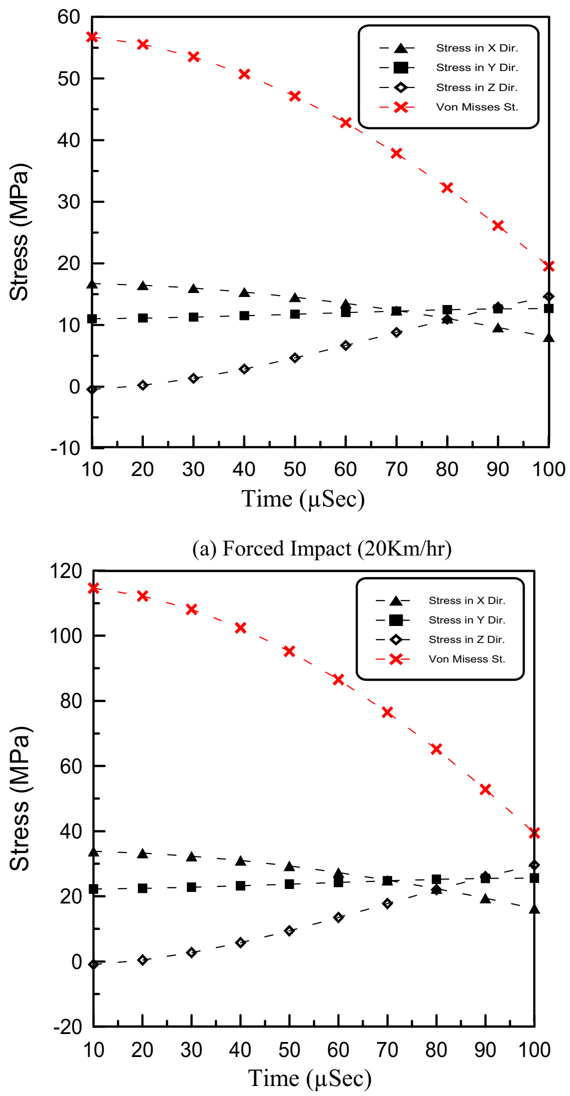


(b) Under 2m free fall

Fig. 6 Von Misses and the three stresses components behavior at every time step during the free falling dynamic model.

Using normalized Von Misses stresses with respect to the applied load, on the free fall dynamic models, the behavior was found to be significantly different. This could be explained due to the effected area of analysis, which is a concentrated area. The impact force dissipates in a relatively lower rate as the area of impact decreases. This results in higher stresses as the impact force increase as a nonlinear behavior [32].

International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:3, No:9, 2009



(a) Forced Impact (20Km/hr)

(b) Forced Impact (40Km/hr)

Fig. 7 Von Misses and the three stresses components behavior at every time step during the forced side impact model.

It can be observed from Fig.7, that under forced side impact loading, the behavior of pelvic bone didn't change in Y direction, where as it changes in X and Z direction. As noted before the time of the operation was constant, so the displacement applied was doubled with the doubled velocity. With linear material used, we recorded high Von Misses stress under 40 Km/hr. The strength of the model decreases with time increasing. Both normalized stresses model behaviors are the same.

When the elastic body is loaded, the work done by the applied forces is stored as a form of energy, which is referred to as the strain energy (U) or the elastic energy, or the deformation energy. Strain energy was extracted for the quasi-static loading, free fall at 1, 2 m and forced side impact at 20, 40 Km /hr dynamic loading. Fig.8, Show the strain energy behavior for the quasi-static loading. The strain energy where increasing monotonically with the increase of the static load. The best fit were selected to fit the resulting strain energy points, and it was found to be a polynomial of second degree, due to the nonlinear curve behavior. The nonlinear second degree polynomial curve is less than linear curve, which could be explained as the decreasing capacity of the pelvic bone to store energy with loading increase.

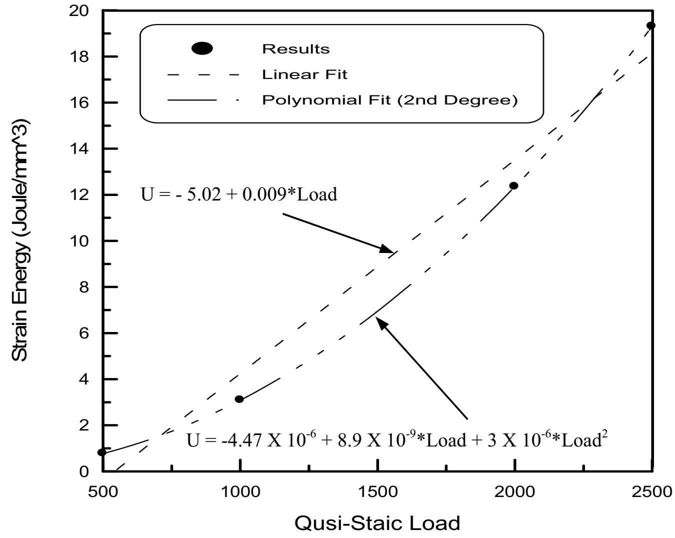


Fig. 8 Strain energy behavior for quasi-static loading.

Fig.9, Fig.10 shows the strain energy results for both the non-forced (free fall) and forced impact. It is observed that for the free fall model at 2m it stored the largest energy. This could be explained by energy dissipation which depend on the loaded area and effected time. It is visible in Fig.9 that the strain energy behavior decrease with time increase up to 50-60µsec approximate, then go to increase even operation done. That drop in strain energy behavior is related to the stresses value at that time, which explained by Fig.6. In the operation beginning, just some on nodes were affected with load, during time the effect was distributed through the structure so the amount of energy stored increase. The different between two curve for free fall models, keep depend on the load effect on body, which with load increase, stresses components increase and strain energy increase with respect to the behavior of each with the time steps.

For forced side impact models, as shown in Fig.10 the curve describe how the load distributed in area, so the energy stored began large and decreased with time increase. This related to the effected area was large in the start of forced impact and during time, only some nodes delivered that affect to the body, so the energy was loosed.

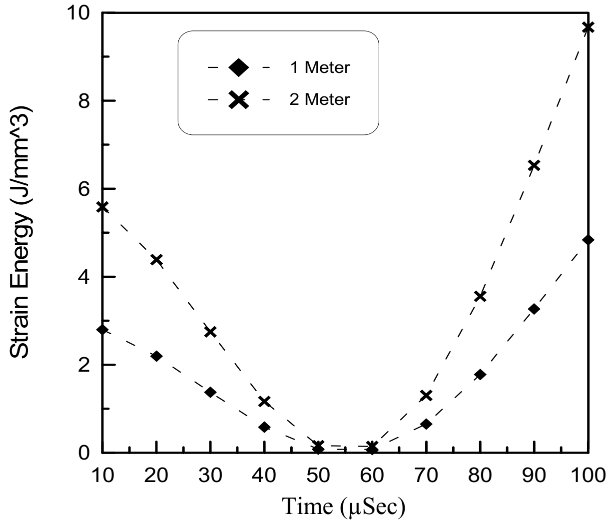


Fig. 9 Distribution of strain energy during free fall models at 1&2 m

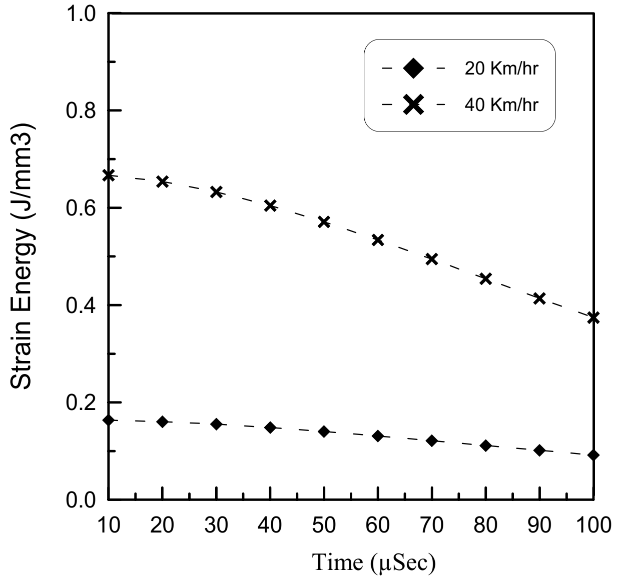


Fig. 10 Distribution of strain energy during forced side impact model at 20&40 Km/hr

VII. Conclusion

Complex shape of the human pelvic bone was successfully captured and modeled. The following remarks can be concluded:

1. Several FE models of the human pelvis have been successfully employed to investigate the pelvis mechanics under gain and dynamic loadings, these earlier FE models mainly considered approximate bone structure where as the actual pelvic joints behaviors was considered through experimental tests which is done under many loading conditions varying from low to high. In this study a fully real pelvic bone using real structure and real material properties were studied. Also, a benchmark analysis was carried out to validate the model boundary conditions.
2. Complete stress and strain analysis in three directions were adopted to explain the behavior of the all different analyses, quasi-static, free falling and forced side impact.
3. The Von Misses stress were compared for four quasi-static loadings cases 500 – 2500 N (~50 – 250 Kg of weight) simulating the effect of weight gain in humans on the composite pelvic bone. Linear correlation between static load and von Misses stresses was observed as expected.
4. For 20 and 40 Km/hr dynamic side impact, the strength of the bone decreases with time increasing. Both model behaviors are the same within shifted results.
5. Using normalized Von Misses stresses with respect to the applied load, on the free fall dynamic models, the behavior was found to be significantly different. Where as, under the forced impact loading condition an over lapping behavior was noticed with respect to same time steps. This could be explained due to the effected area of analysis, where the forced impact acts in large area relatively compared to the free fall one.
6. Results were also examined the strain energy, for static load, it was depend on material properties of pelvic bone, also, it noticed that with the change of the effected loaded area the energy dissipation changed, which could be explained as the dissipation in the case of free fall are limited to a smaller area in limited time, so the effect of load distributed in the whole bone and highest strain energy values were recorded. Where as in the forced impact case the effected area of dissipation is the whole bone.

References

[1] Dalstra, M., Huiskes, R., and Van Erning, L., ''Development and Validation of a Three-Dimensional Finite Element Model of the Pelvic Bone,'' Journal of Biomechanical Engineering. , Vol. 117, pp. 272–278, 1995.

[2] Tile, M., ''Pelvic Ring Fractures: Should They Be Fixed?'' The Journal of Bone and Joint Surgery., Vol. 70(B), pp. 1–12, (1988).

[3] Simonian, P. T., and Routt, M. L., ''Biomechanics of Pelvic Fixation,'' Journal of the American Academy of Orthopaedic Surgeons., Vol. 28 (3), pp. 351–367, 1997.

[4] Routt, M. L., Simonian, P. T., and Swiontkowski, M. F., ''Stabilization of Pelvic Ring Disruptions,'' Journal of the American Academy of Orthopaedic Surgeons., Vol. 28 (3), pp. 369–388, 1997.

[5] Bell, A. L., ''Comparative Study of the Orthofix and Pittsburgh Frames for External Fixation of Unstable Pelvic Ring Fractures,'' Journal of Orthopedic Trauma, Vol. 2, pp. 138–140, 1988.

[6] Simonian, P. T., ''Biomechanical Simulation of the Anteroposterior Compression Injury of the Pelvis,'' Clinical Orthopaedics and Related Research, Vol. 309, pp. 245–256, 1994.

[7] Holm, N. J., ''The Development of a Two-Dimensional Stress-Optical Model of the Os Coxae,'' Acta Orthopaedica Scandinavica., Vol. 52, pp. 135–143, 1981.

[8] Yeh, O., and Keaveny, T.M., "Relative Roles of Microdamage and Microfracture in the Mechanical Behavior of Trabecular Bone," Journal of Orthopaedic Research, Vol. 19, pp. 1001–1007, 2001.

[9] Maurice G.,Taghite M., and Taous K., "An experimental and theoretical approach of elasticity and viscoelasticity of compact and spongy bone with periodic homogenization," Computer Methods in Biomechanics and Biomedical Engineering, Vol. 10, pp. 195-207, 1997.

[10] Renaudin, F., Lavaste, F., Skalli, W., Pecheus, C., and Scmitt, V., ‘‘A 3D Finite Element Model of Pelvis in Side Impact,’’ Proceedings of The Congress European Society of Biomechanics, Vol. 8, pp. 180-194, 1992.
[11] Bouquet, R., Ramet, M., Bermond, F., Caire, Y., Talantikite, Y., Robin, S.,Voiglio, E. Pelvis human response to lateral impact. In: Proceedings of the 16th International Technical conference on the Enhanced Safety of Vehicles. Windsor, ON, Canada, 1998, pp. 1665-1686.
[12] Brugess, A., and Tile, M., ‘‘External Fixation in Fractures of the Pelvis and Acetabulum’’, Williams & Wilkins, Baltimore, pp. 135–149, (1995).
[13] Landjerit, B., Jacquard-Simon, N., Thourot, M. and Massin,P. H., ‘‘Physiological loadings on human pelvis: a comparison between numerical and experimental simulations.’’, Proceedings of The Congress European Society of Biomechanics, Vol. 8, pp. 195-206, 1992.
[14] Cesari D., Ramet M., and lair P., ‘‘Evaluation Of Pelvic Fracture Tolerance In Side Impact’’,Stapp Car Crash Conference, Warrendale, PA, USA, pp. 231-253, (1980).
[15] Cesari D., and Ramet M., ‘‘Pelvic Tolerance and Protection Criteria in Side Impact’’, Stapp Car Crash Conference, Warrendale, PA, USA, pp.145-154, 1982.
[16] Cesari D., Bouquet R., and Zac R., ‘‘A New Pelvis Design For The European Side Impact Dummy’’, Stapp Car Crash Conference, Warrendale, PA, USA, pp. l-11, 1984.
[17] Hartemann F., Thomas C., Foret-Bruno JY., Henry C., Fayon A., and Tarriere C., ‘‘Occupant Protection In Lateral Impacts’’, Stapp Car Crash Conference, Warrendale, PA, USA, pp. 19l-219, 1976.
[18] Plummer, J.W., Eberhardt, A.W., Alonso, J.E., Mann, K.A., "Parametric tests of the human pelvis: the influence of load rate and boundary condition on peak stress location during simulated side impact. Advances in Bioengineering",1998, ASME BED, vol. 39, pp. 165-166.
[19] Dawson, J.M., Khmelniker, B.V., McAndrew, M.P. Analysis of the structural behavior of the pelvis during lateral side impact using the finite element method. Accident Analysis and prevention 31, 109-119, 1999.
[20] Renaudin F., Guillemot H., Lavaste F., and Skalli W., ‘‘A 3D Finite Element Model Of Pelvis In Side Impact’’, Stapp Car Crash Conference, Society of Automotive Engineers, Warrendale, PA, USA, pp. 242-264, (November 1993).
[21] Simonian, P. T., and Routt, M. L., ‘‘Biomechanics of Pelvic Fixation,’’ Journal of the American Academy of Orthopaedic Surgeons., Vol. 28 (3), pp. 351–367, 1997.
[22] Viano DC., ‘‘Biomechanical Responses And Injuries In Blunt Lateral Impact’’,Stapp Car Crash Conference, Society of Automotive Engineers, Warrendale, PA, USA, pp. 113-142, 1989.
[23] Zhu JY., Cavanaugh JM., and King AI., ‘‘Pelvic Biomechanical Response and Padding Benefits in Side Impact Based on a Cadaveric Test Series’’, Stapp Car Crash Conference, Society of Automotive Engineers, Warrendale, PA, USA, pp. 128-139, (November 1993).
[24] Ansel C.,and Saul K., ‘‘Advanced Strength and Applied Elasticity”, Prentice Hall, fourth edition. pp.65-70, 2003.
[25] Boissonnat, J.-D., “Shape Reconstruction from Planar Cross-Sections,” Computer Vision, Graphics, and Image Processing., Vol. 44, pp. 1–29, 1988.
[26] Schroeder, W. J., Zarge, J., and Lorenson, W. E., “Decimation of Triangle Meshes,” ACM SIGGRAPH Computer Graphics, Vol. 26, pp. 65–70, 1992.
[27] Taubin, G., Zhang, T., and Golub, G., “Optimal Surface Smoothing as Filter Design,” Stanford University IBM RC-20404, pp. 212-225, 1996.
[28] Viano D., “Biomechanics of bone and tissue: a review of material properties and failure characteristics”, The Stapp Car Crash Conference, Society of Automotive Engineers, Warrendale, PA, USA, pp.33-63, 1986.
[29] Kuhn J.L., Goldstein S.A., Choi K., London M., Feldkamp L.A., and Matthews L.S., “Comparison of the trabecular and cortical tissue moduli from iliac crests”, Journal of orthopedic research, Vol. 7 (6), pp. 876-884, 1989.
[30] Mak A.F., “The apparent visco-elastic behavior of articular cartilage - The contributions from the intrinsic matrix visco-elasticity and interstitial fluid flows”. Jouranl of biomechanical engineering, pp. 178-196, 1986.
[31] Oonishi H., and Isha H., Hasegawa T., “Mechanical analysis of the human pelvis and its application to the artificial hip joint-by means of the three dimensional finite element method”, Journal of Biomechanics, Vol. 16 (6), pp. 427-444, 1983.
[32] Fung Y.C., “Biomechanics – Mechanical properties of living tissues”, Springer-Verlag., Vol. 2, pp. 265-286, 1965.

M. S. El-Asfoury Graduate Student in the department of Mechanical Design & Production Department, Faculty of Engineering, Port-Said, Suez Canal University, Egypt. He earned his B.Sc. at June 2006 from Faculty of Engineering at Port-Said, Suez-Canal University. His major fields of study are materials characterization and behavior, static and dynamic FEA simulations, and NDE.

M. A. El-Hadek Assistant Professor in the department of Mechanical Design & Production Department, Faculty of Engineering, Port-Said, Suez Canal University, Egypt. Phone: +2 010 827-1778, e-mail: m.elhadek@scuegypt.edu.eg. The author to whom all correspondence should be addressed. He earned his B.Sc. in Mechanical Engineering at 1996, from Mansoura University, Egypt. His M.Sc. in Mechanical Engineering at 1999, from Tuskegee University, USA. His P.hD. in Mechanical Engineering at 2003, from Auburn University, USA. His major fields of study are Experimental Mechanics, Fracture Mechanics, Multifunctional Materials (Functionally Graded Materials, Syntactic Foams, Interpenetrating Network Composites, Nano-Composites), Optical and Infrared Sensors and Micro-Mechanical Testing (NDE). ASME member since 1999, and SEM member since 2001. The receptiant of Harry Merriwether Fellowship, for Best Engineering Researcher 2002-2003. He is also, a Technical Expert at the Egyptian Higher Education Minstery, for Enhancing and Education Development, Egyptian Educational Development Fund EDF. He has over 30 peer reviewed international publications.