S. Majumder, A. Roychowdhury
Dept. of Applied Mechanics, Bengal Engineering College (Deemed University), Howrah, India

S. Pal
School Bio-science and Engineering, Jadavpur University, Kolkata, India

A Finite Element Study on the Behavior of Human Pelvis under Impact through Car Door

Abstract

Pelvic fracture, cracking or breaking of a portion of the pelvis are extremely common injuries in the side impact collisions of motor vehicles. Due to both its shape and structural architecture, mechanics of the pelvic bone is complicated. There is a lack of knowledge regarding the dynamic behavior of the pelvis and its biomechanical tolerance under impact environment. Hence this study is aimed at the understanding of the mechanical response of the human pelvis with three-dimensional finite element (FE) models, under side impact load, applied through a structure, equivalent to a car door. The door structure was modeled, considering few layers, consisting of foam (Styrodur®, 3035 CS), plastic (UHMWPE), steel, glass and steel, putting them in series. A soft tissue layer (equivalent to fat) was also considered on the greater trochanter location. These FE models (with and without the car door structure) were analyzed with ANSYS-LS-DYNA® dynamic finite element software to compare the effect of the car door padding system for shock absorption. It was observed that with proper combination of shock absorbing material (foam, etc.) and its thickness, the transmission of impact load to the body part (pelvis, etc.) from the outer surface of the car door could be reduced.

Introduction

The Pelvis is most susceptible to severe fractures in near-side motor vehicle crashes due to intrusion of door and impact loading through greater trochanter. Car manufacturers are giving more importance to the protection of the occupants in lateral impacts. With the rapid industrialization and advancement of technology, uses of high-speed cars are increasing day by day. As a result, the car accidents are also increasing. A survey showed that side-impacts represented 15-30% of the collision [9]. Due to these accidents, the pelvis (8-14%) is the one of the most affected areas of the human system [2]. Pelvic fracture, cracking or breaking of a portion of the pelvis [14, 20] are extremely common injuries in side-impact collisions of motor vehicles. The victim of a side-impact car collision is likely to end up with a fractured pelvis, an injury that may take weeks or months to heal. Otherwise, the victim may walk away only to discover years later that he or she is suffering from post-traumatic arthritis, a long-term disability caused by undiagnosed cartilage damage. Fractures at pubic rami, acetabuli, iliac wing, pelvic ring disruption, or posterior injuries such as sacral fracture are commonly observed [8, 19].

In the musculoskeletal system, the pelvis is one of the most vital components. The lower extremities are connected to the upper portion of the body through the pelvis. The role of the pelvis is to transfer gravitational and external load across the sacro-iliac joints and the hip joints. The pelvic bone contains mainly of low-density trabecular bone that is covered by the high strength cortical bone of varying thickness in the form of a thin shell. Due to both its shape and structural architecture, the mechanics of the pelvic bone is complicated.

To establish the biomechanical response and injury tolerance, automotive side impact conditions were simulated through experimental analyses [2, 3, 7, 12, 13, 21]. Through these investigations, different fracture tolerance criteria were established, with the help of different testing protocols. In addition to experimental testing, only few investigators [5, 11, 15, 17] performed analytical simulations to address the issue of impact load tolerance of the pelvis with the help of the finite element method (FEM).

Objectives of this study were to develop a three-dimensional finite element model of the pelvis along with an equivalent car door structure and analyze them under dynamic load, resulting from motor vehicle side impact condition. Thus the present study was aimed at a better understanding of the mechanical response of the pelvis under dynamic loading through a car door and impact absorbing capability of the car door with proper padding.

Methodology

Solid modeling, finite element mesh generation, selection of material properties, imposition of boundary conditions (loads and constraints), solutions, dynamic analysis and interpretation were done with the help of the commercially available finite element modeling and dynamic analysis software ANSYS® (ANSYS, Inc. Pennsylvania, USA) and ANSYS-LS-DYNA® (LSTC Corporation, USA and ANSYS, Inc. Pennsylvania, USA).

Finite Element Modeling

The three-dimensional finite element model of the pelvis, used in this study, was a modified version of our earlier models (figure 1a–c, MAJUMDER et al. [10] and figure 1d–f, MAJUMDER et al. [11]), developed from co-ordinate data [6] of a dried pelvis cadaver. An idealized sacral bone was modeled for this study, in the form of a solid bar (figure 1d–f) whose cross-section was close to sacro-iliac articulating surface. As the side impact

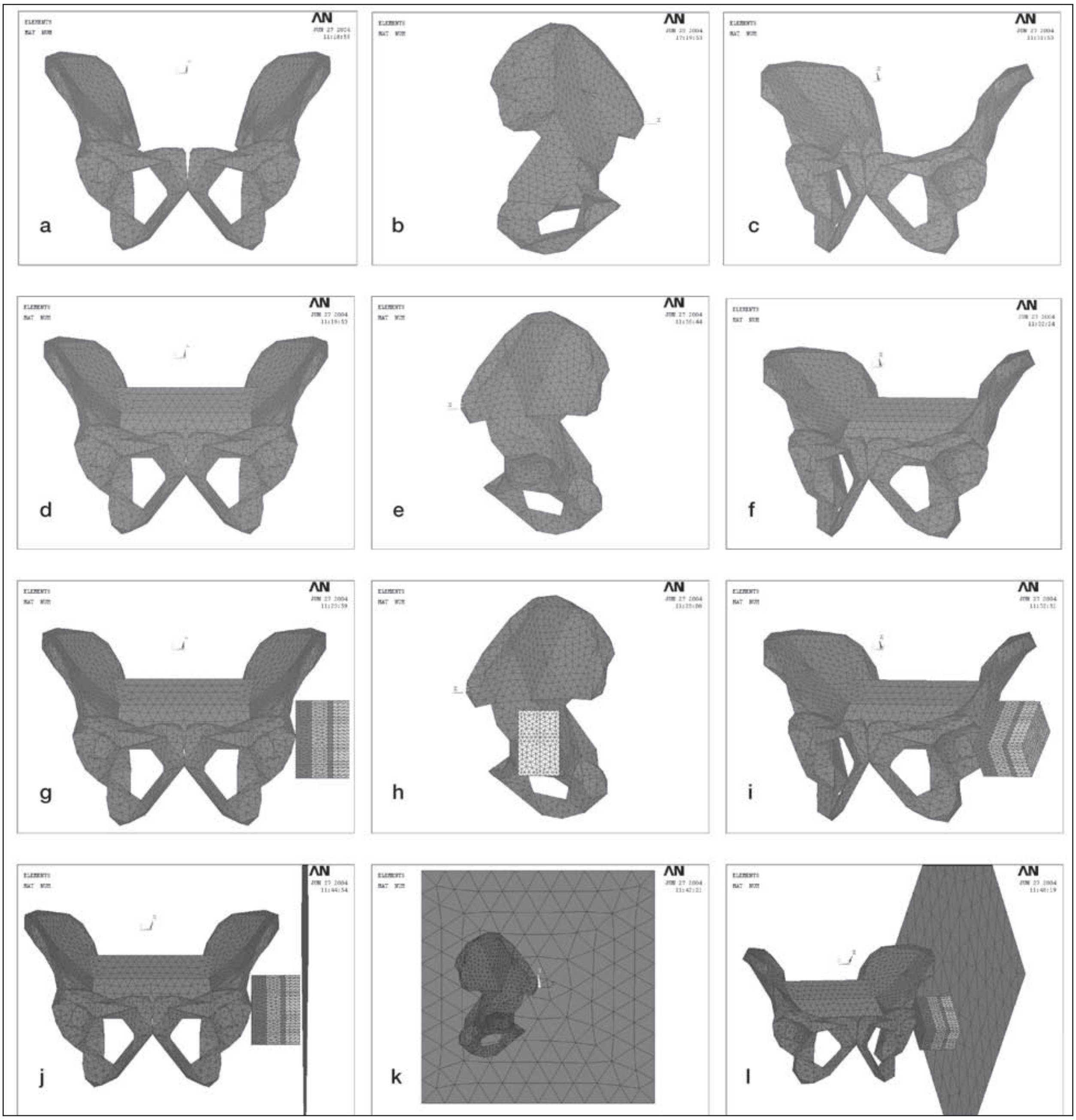

Fig. 1: Different views of the three-dimensional finite element models, (a-c): Only pelvis without sacrum, from MAJUMDER et al. [10]|; (d-f): Full pelvis with idealized sacrum for this study and from MAJUMDER et al. [11]; (g-i): Full pelvis with equivalent car door structure; (j-l): Rigid wall with which full pelvis with equivalent car door structure collided

is transmitting through the car door, an equivalent car door structure attached to the left greater trochanter was modeled (figure 1g–l). This equivalent structure of a car door was simulated by considering few layers (‘b’, ‘c’, ‘d’, ‘e’ and ‘f’ in figure 2) consisting of foam (Styrodur®, 3035 CS), plastic (UHMWPE), steel, glass and steel, putting them in series (with thickness of 15, 5, 5, 5 and 5mm respectively). A soft tissue layer (equivalent to soft tissue and fat) of 15mm thickness was also considered on the greater trochanter location (layer ‘a’ in figure 2). Assuming the car hitting the wall on lateral impact, an equivalent rigid wall was modeled (figure 1j–l) very near (6 mm) to the pelvis (acetabulum) and car door structure.

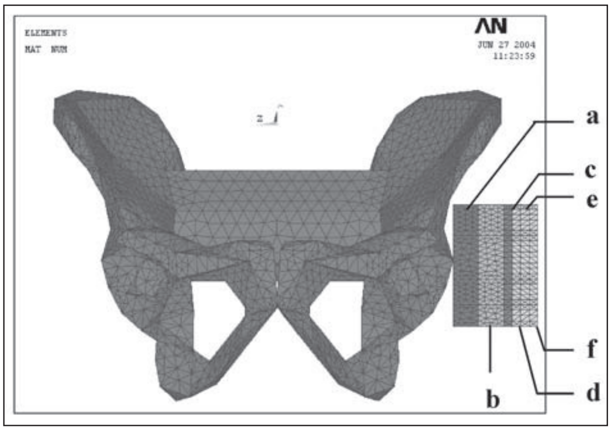

Fig. 2: Three-dimensional finite element model (22,967) tetrahedral and 5,820 shell elements through 5,824 nodes) of human pelvis with layers of soft tissue and equivalent car door structure (a – Soft tissue; b – Styrodure® Foam; c – Plastic (UHMWPE); d – Steel; e – Glass; f – Steel)

	E (MPa)	ρ (kg/m³)	ν
Cortical Bone*	17,000	2,000	0.3
Trabecular Bone*	70	1,500	0.2
Soft Tissue**	20	750	0.49
Styrodur® Foam***	20	33	0.4
Plastic (UHMWPE)**	1,100	937	0.34
Glass	62,000	2,230	0.22
Steel	200,000	7,850	0.3

E – Young’s modulus of elasticity
ρ – Density and ν – Poisson’s ratio
* from DALSTRA and HUISKES [4]
** from ROYCOWDHURY [18] and
*** from BASF Aktiengesellschaft [1]

Tab. 1: Material properties, used for FE model of the pelvic bone, soft tissue and equivalent car door structure

The shell element and solid (tetrahedral) element were used to represent the cortical bone and trabecular bone of the pelvis respectively. Soft tissue (fat) and five layers of the equivalent car door structure were modeled with solid (tetrahedral) elements. The degrees of freedom (dof) for the ANSYS® solid and shell elements were six (three translational and three rotational) each. Similarly the dof for the ANSYS-LS-DYNA® solid and shell elements were nine (three translational, three velocity, three acceleration) and twelve (three translational, three rotational, three velocity, three acceleration) respectively. The pelvic FE model without the car door structure (figure 1d–f) contained 5,820 shell elements and 13,070 tetrahedral elements. Hence the total 18,890 elements were connected through the 3,704 nodes. The FE model of the car door structure contained 9,897 tetrahedral elements. Hence in case of the pelvic model with car door (figure 1g–i), the total 28,787 elements were connected through 5,824 nodes.

The material properties of the pelvic bone were assumed to be isotropic and the material distribution was assumed to be homogeneous throughout the pelvic model. The same was considered for the layers of the car door structure. All the properties required for this analysis are given in table 1.

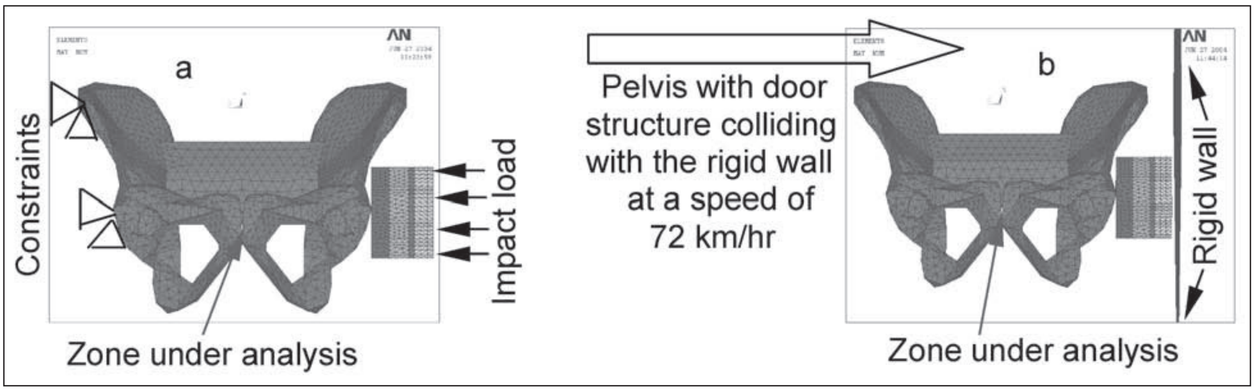

Fig. 3: Impact load cases for analysis with (a) ANSYS®, (b) ANSYS-LS-DYNA® software

Impact Loading

To simulate the motor vehicle side impact situation on a finite element model, one needs to know the loading data during the impact between two objects (motor vehicle and rigid wall for example). This may be applied in two ways: Case 1: in the form of impact load and impact duration; Case 2: in the form of velocity and acceleration. The first case was simulated with the ANSYS® software and the second case with the ANSYS-LS-DYNA® software.

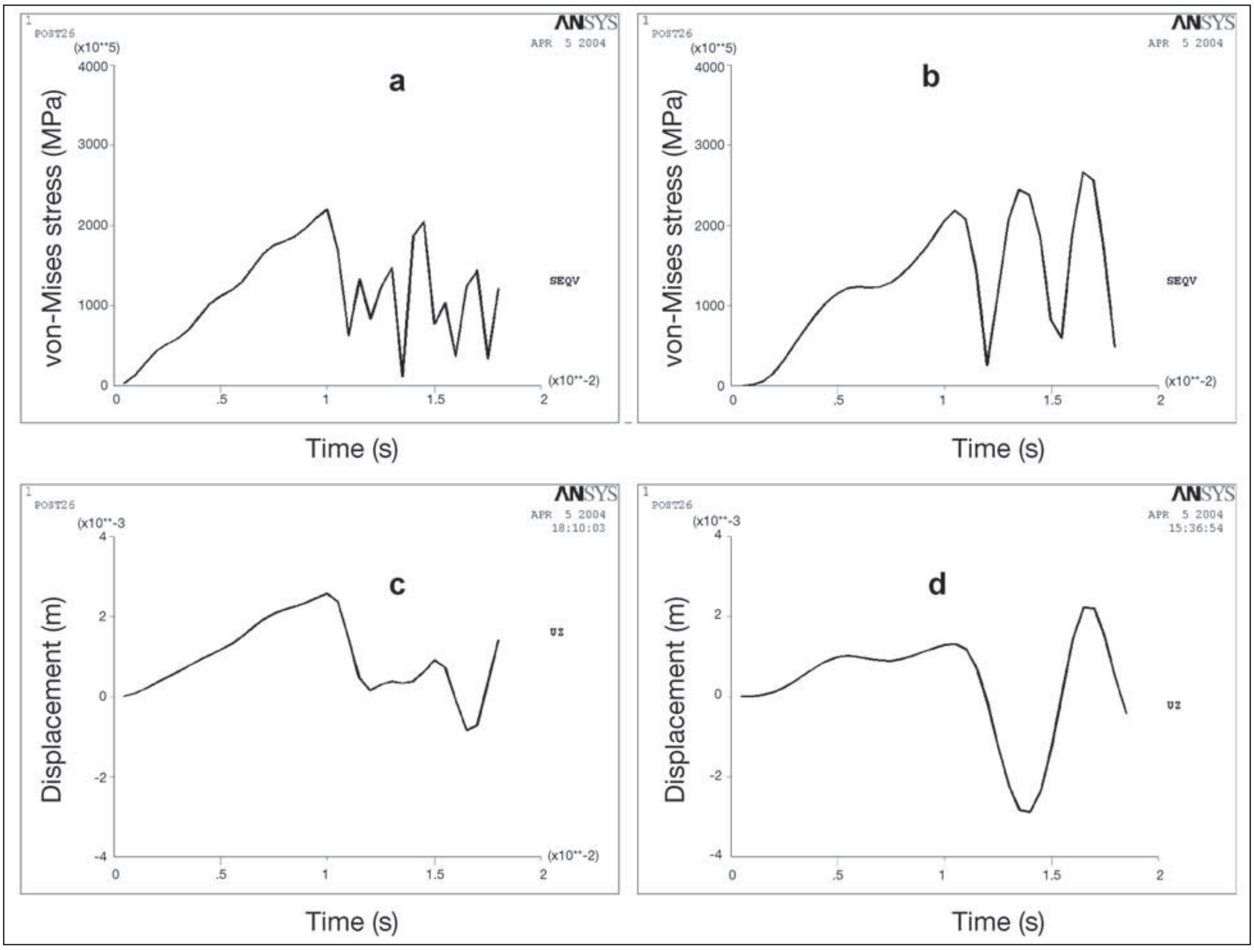

Fig. 4: Von-Mises stress plot and displacement (in the direction of impact) plot with impact duration for pelvis without car door and padding (a and c respectively) (from MAJUMDER et al. [11]) and for pelvis with car door and padding (b and d respectively) (present study), analyzed by ASIS® software

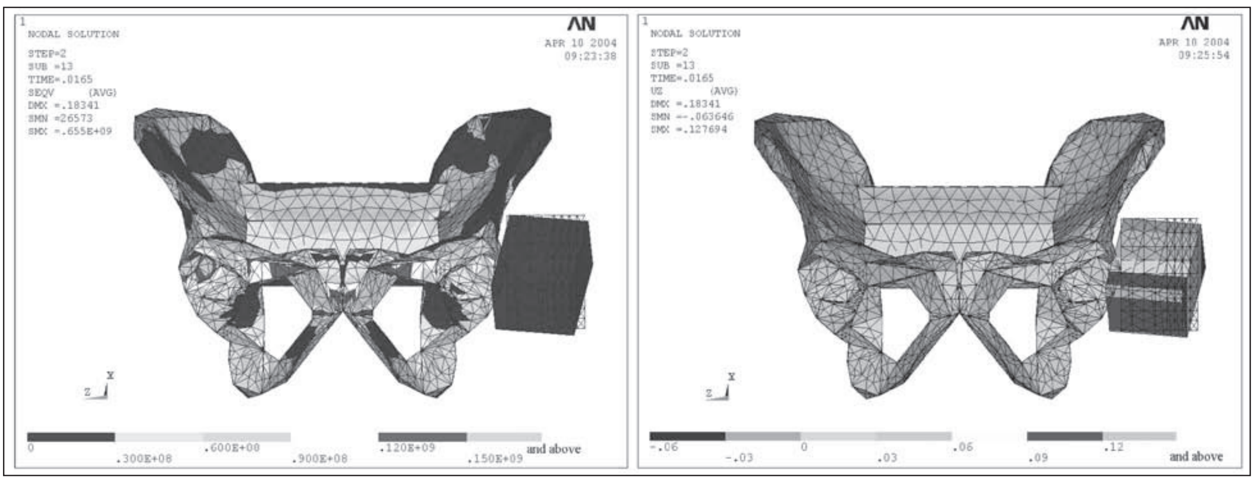

Fig. 5: (a) von-Mises stress (MPa) and (b) displacement (m) contour in the direction of impact, for the pelvis with car door structure and padding, at 16.5ms, for the first load case, analyzed with ®ANSYS software

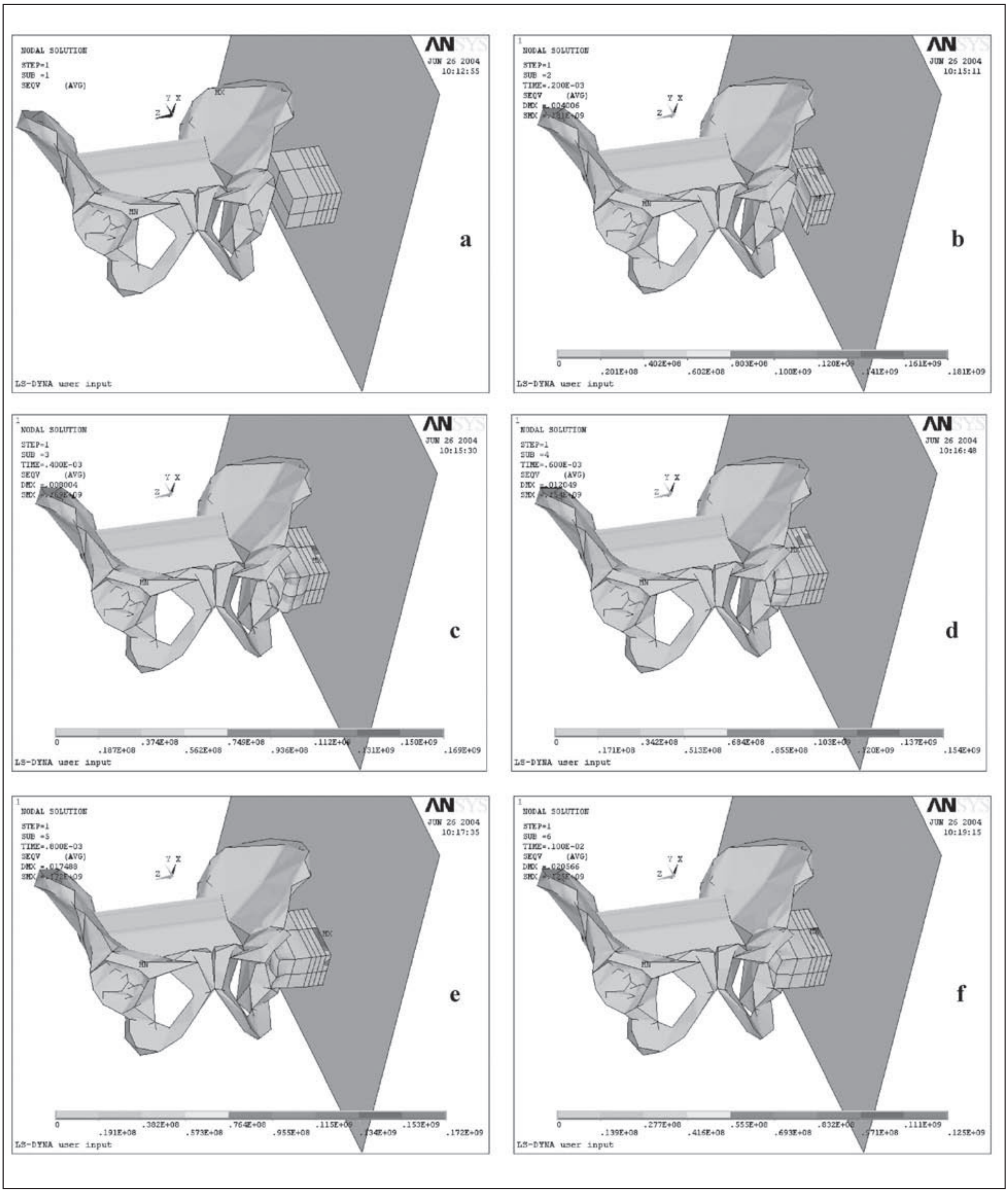

Fig. 6: (a-f) von-Mises stress contour (Pa) during different sub-time steps for the pelvis with car door and padding, for the second load case, analyzed with ANSYS-LS-DYNA® software

Load Case 1

An impact load of 5kN was applied on the outer surface (figure 3a) of the last layer (steel) for a duration of 18ms with the peak load occurrence at 10ms. As the impact load was applied to the right side in a direction from left to right, the right iliac and right acetabulum zone were constrained to no displacement situation. This case is comparable to our previous investigation [11] without the equivalent car door structure. This case was solved with the ANSYS® software.

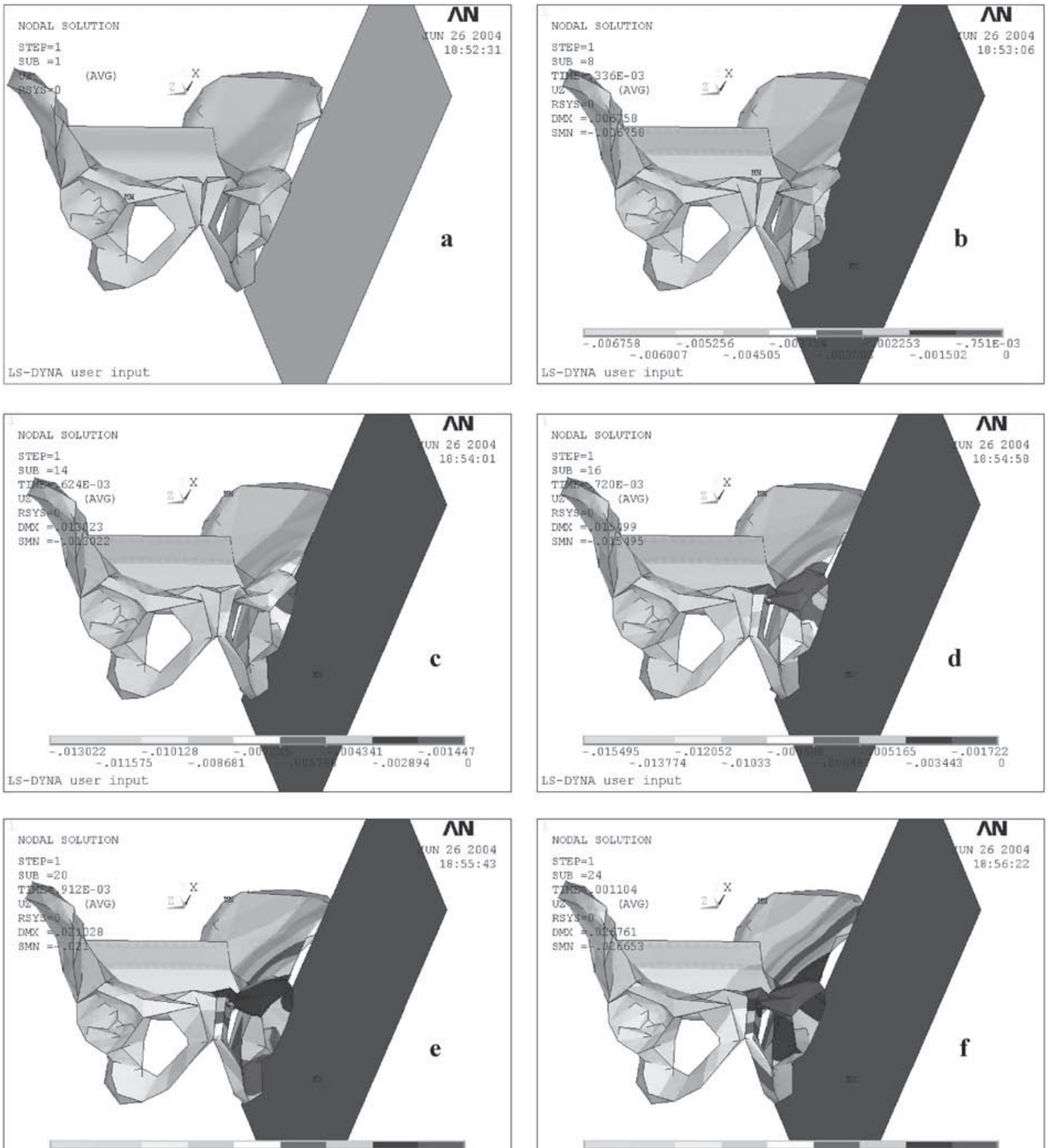

Fig. 7: (a-f) Displacement contour (m in the direction of impact, during different sub-time steps for the pelvis withhout car door, for the second load case, analyzed with ANSYS-LS-DYNA® software

Load Case 2

It was considered that the motor vehicle, with a velocity of 72km/hr collided with a rigid wall. The same condition was applied to the pelvic model with the equivalent car door structure (fig. 3b). To reduce the CPU time, the distance between the door outer surface (steel layer) and rigid wall was kept very low (6mm), so that the door along with the pelvis came into contact with the rigid wall within a very short duration. The impact duration was taken as 1.2ms. Another analysis was done with the pelvic model, without the car door structure, for the same impact condition. These cases were solved with the ANSYS-LS-DYNA® software.

Results and Diskussion

For both the load cases, the von-Mises stress and displacement (in the direction of impact) criterion were considered. For the first load case (figure 3a), analyzed by ANSYS®, it was observed that the maximum von-Mises stress (figure 4b) for a particular zone, i.e. pubic symphysis, was exceeding the compressive strength (200MPa) [16] under 5kN load case. At the end (10ms) of peak impact load, stress was 205MPa and it went on increasing up to a value of 26MPa at 16.5ms. This gradual increase depicted actual load transfer and impact absorption through car door, as compared to our earlier study, without car door and padding [11], where the highest stress (266MPa) (figure 4a) occurred at the end (10ms) of peak impact load of 5kN. The same trend was observed from the displacement pattern (figure 4c, d). The displacement in the direction of impact (figure 3a) was low at 10ms (figure 4d), as compared to our earlier findings, without padding (figure 4c). These results were also similar to the findings of the previous experimental investigations [3, 7, 13, 21]. Though the peak values were not reduced to that level during impact, the attainment of the peak values was delayed. These were due to the impact absorbing capabilities of the padding materials. The von-Mises stress and displacement contour (in the direction of impact) at 16.5ms ware given in figure 5a and 5b respectively for the first load case.

For the second load case (analyzed by ANSYS-LS-DYNA®), the von-Mises stress contours, for the pelvis with car door and padding ware shown in figure 6a-f. Displacement contours in the direction of impact, for the pelvis without car door ware shown in figure 7a-f respectively. From these contours it was observed that the padding system had reduced the impact transmission effect on the pelvis by to some extent, as compared to the case of pelvis without car door. For the case without the car door, to maintain a distance of 6mm between the greater trochanter and the rigid wall, the height of rigid wall was reduced (figure 7), as compared to the rigid wall in the case of the pelvis with car door (figure 6). Hence the rigid wall did not interfere with the superior ilium and the acetabulum came into contact first with the rigid wall in both the cases.

Conclusion

Car manufacturers are becoming more and more concerned with the protection of the occupants in lateral impacts. But there are many knowledge gaps regarding the behavior of various regions of the pelvis and its biomechanical tolerance, under dynamic loading such as heavy impact due to motor vehicle accidents. This knowledge is essential in order to optimize protection devices and car structures with regard to the security of the occupants. This knowledge is also important for designing improved crash dummies or mathematical models of the car occupants. Current research in car door padding and side air bag technology have been greatly focused on the side impact force and stress distribution on the pelvis. Hence to study the behavior of the human pelvis under impact through car door, the three-dimensional finite element model with the sacrum bone and equivalent car door structure with padding, attached to pelvis was developed, with 22,967 tetrahedral and 5,820 shell elements through 5,824 nodes. From two load cases, analyzed with ANSYS® and ANSYS-LS-DYNA®, it was concluded that for the cases with padding, the occurrence of peak stress and displacement was delayed and peak values were reduced, as compared to the case without padding

Acknowledgements

The authors would like to thank Dr. S. L. DELP, Associate Professor, Biomechanical Engineering Division, Mechanical Engineering Department, Stanford University, Stanford, USA, for providing the bone surface data of pelvis without sacrum. The authors would also like to acknowledge Mr. Debjit CHAKARBORTY for providing the technical expertise regarding ANSYS-LS-DYNA®.

References

[1] BASF Aktiengesellschaft: Ludwigshafen, Germany, http://www.basf.de

[2] D. CESARI, M. RAMET, P. Y. CLAIR: Evaluation of pelvic fracture tolerance in side impact. Proceedings of the 24th Stapp Car Crash Conference, Troy, MI, 1980, 229-253

[3] D. CESARI, M. RAMET: Pelvic tolerance and protection criteria in side impact. Proceedings of the 26th Stapp Car Crash Conference, Ann Arbor, MI, 1982, 145-154

[4] 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 117, 1995, 272-278

[5] J. M. DAWAON, B. V. KHMELNIKER, M. P. McANDREW: Analysis of the structural behavior of the pelvis during lateral impacy using the finite element method. Accident Analysis and Prevention 31, 1999, 109-119

[6] S. L. DELP: Surgery simulation: computer graphics system to analyze and design musculoskeletal reconstruction of the lower limb. PhD dissertation, Standford University, Standford, 1990

[7] H. GUILLEMOT, C. GOT, B. BESNAULT, F. LAVASTE, S. ROBIN, S. J. Y. LE COZ, J. P. LASSAU: Pelvic behavior in side collisions: static and dynamic tests on isolated pelvic bones. Proceedings of the 16th International Technical Conference on the Enhanced Safety of Vehicles, Windsor, ON, Canada, 1998, 1412-1424

[8] A. A. HOFMANN, C. P. DAHL, R. W. WYATT: Experience with acetabular fractures. Journal of Trauma 24, 1984, 750-752

[9] P. R. LEWIS Jr., F. J. MOLZ, S. Z. SCHMIDTKE, M. W. BIDEZ, P. R. LEWIS: A NASS-based investigation of pelvic injury within the motor vehicle crash environment. Proceedings of the 40th STAPP Car Crash Conference, Alburquerque, NM, 1996, 123-129

[10] S. MAJUMDER, A. ROYCHOWDHURY, S. PAL: Variations of stress in pelvic bone during normal working, considering all active muscles. Trends in Biomaterials & Artificial Organs 17, 2004a, 48-53

[11] S. MAJUMDER, A. ROYCHOWDHURY, S. PAL: Dynamic response of the pelvis under side impact load – a three-dimensional finite element approach. International Journal of Crashworthiness 9, 2004b, 89-103

[12] F. L. MOLZ, P. D. GEORGE, L. S. GO, M. W. BIDEZ, A. I. KING, J. ALONSO: Simulated automotive side impact on the isolated human pelvis. Phase I: development of a containment device. Phase II: analysis of pubic symphysis motion and overall pelvic compression. Proceedings of the 41st Stapp Car Crash Conference, Orlando, FL, 1997, 75-89

[13] G. S. NUSHOLTZ, P. S. KAIKER: Pelvic stress. Journal of Biomechanics 19, 1986, 1003-1014.

[14] J. F. PERRY: Pelvic open fractures. Clinical Orthopaedics 151, 1980, 41

[15] 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. Proceedings of the 40th Stapp Car Crash Conference, Alburquerque, NM, 1996, 17-28

[16] D. T. REILLY, A. H. BURSTEIN: The mechanical properties of cortical bone. Journal of Bone and Joint Surgery 56A, 1974, 1001-1022

[17] F. RENAUDIN, H. GUILLEMOT, F. LAVASTE, W. SKALLI, F. LESAGE, C. PECHEUX: A 3D finite element model of pelvis in side impact. Proceedings of the 37th Stapp Car Crash Conference, Washington, DC, 1993, 249-259

[18] A. ROYCHOWDHURY: Stress & tribological analysis of artificial human knee joint. PhD dissertation, Jadavpur University, Kolkata, India, 1996

[19] J. D. SPENCER, T. LALANADHAM: The mortality of patients with minor fractures of the pelvis. Injury 16, 1985, 321-323

[20] M. TILE: Fractures of the pelvis and acetabulum. Williams & Wilkins, Baltimore, 1995

[21] J. Y. ZHU, J. M. CAVANAUGH, A. I. KING: Pelvic biomechanical response and padding benefits in side impact based on a cadaveric test series. Proceedings of the 37th Stapp Car Crash Conference, Washington, DC, 1993, 223-233