EXPERIMENTAL VALIDATION OF THE IMPACT CALCULUS MODEL FOR A BONE TISSUE

Alexander BABIA[1], Horia Alexandru PETRESCU[2], Horia Miron GHEORGHIU[3]

A numerical impact calculus model, destined for an in depth analyses of bone tissue behaviour during impact. is validated in an in vitro experiment. The obtained results can be used to determine the effect of fracture. As the criteria for an experimental validation of the model was chosen comparing the strength in the impactor transmitting energy to the bone with that of the impactor model simulated numerically and considered as deformable body. Reasonable level of error resulting from this comparison confirms the correctness of the considered model.

Keywords: impact, bone, numerical analyses, fracture.

1. Introduction

In mechanics, an impact is a high force or shock applied over a short period of time when two or more bodies collide. Such a force or acceleration usually has a greater effect than the same force applied over a proportionally longer period of time. The effect depends critically on the relative velocity of the bodies

At normal speeds, during an inelastic collision, a body struck by a projectile will suffer a deformation. This deformation will absorb most, or even all, of the force of the collision. Viewed from the perspective of the conservation of energy, the kinetic energy of the projectile is used to produce deformations witch lead to heat and sound, as a result of the vibrations induced in the struck object. However, these deformations and vibrations cannot occur instantaneously.

A high-velocity collision (an impact) does not provide sufficient time for these deformations and vibrations to occur. Thus, the struck material behaves as if it were more brittle than it is, and the majority of the applied force goes into fracturing the material [1].

Impact strength as a critical measure is one of the most important properties to consider, as well as the most difficult to quantify, involving complex problems [2], [3].

[1] Departement of Strength of Materials, University POLITEHNICA of Bucharest, Romania, E-mail: alexander.babia@yahoo.fr
[2] Departement of Strength of Materials, University POLITEHNICA of Bucharest, Romania, E-mail: alexander.babia@yahoo.fr
[3] Departement of Strength of Materials, University POLITEHNICA of Bucharest, Romania, E-mail: alexander.babia@yahoo.fr

In orthopedic medicine, injuries of bone elements, produced by applying shock loads occupy an important place. Choosing and applying the means of treating fractures depends on the proportions of injuries. These are related to the stress created by the impact, but also to the displacements it produces.

Over the last decade, the finite element method has been widely used to study the behavior of bone elements. Models were developed to study the mechanical behavior at static or dynamic loads, with specific application to different segments of bone: femur [4], pelvic bone and its components [5], clavicle [6], skull [7], [8], mandible [9], vertebrae [10], [11] and others. Generally, components with prevailing trabecular bone are studied separately, considered as non-homogeneous and anisotropic material [10], [12], [13], while in the studies about components having the strength ensured by cortical bone the material is considered homogeneous and isotropic. [14], [15]. Few studies reported the finite elements methods (FEM) analysis of the conditions that can produce open or closed fractures, when bone is protected by a tegument layer.

Checking the calculation models used in these studies is required as a necessary process, due to the complexity of the studied phenomenon influenced by multiple factors, and the extent to which the assumptions are used. Even for static cases, few of the works cited perform model checking [5], [16]. Dokko et al., for example, perform in [16], [17] a qualitative check of the calculation model by comparing the results obtained by FEM with neuropathological effects.

The aim of this paper is to present the validation criteria for a computational model of a skeletal element submitted to shock. Model description, method of implementation, its advantages and possibilities of using are not discussed here.

2. Validation procedure

The parameter chosen to validate the calculation model was the force of the impact hammer striking the bone sample and which can be measured experimentally when the shock is applied. This force was then compared with the one obtained using the finite element model that contains not only skeletal element investigated, but also the impactor, treated as deformable body.

The method was applied to the tibia bone of cow, for which a sufficient number of specimens were available for experiments.

A total of five specimens taken from such bones were tested to shock loading, obtaining thus the maximum force of the hammer impactor, for a given impact energy.

Two of the considered samples had a similar physical model, taken from the opposite leg of the same animal. They were prepared in order to obtain the geometrical model for the numerical calculation.

The model obtained from the bone specimens used in the experiment was used as a finite element model in a dynamic numerical analysis in which a shock having the same energy as the one from experiments was applied. From this analysis, the maximum stress in the impactor was determined and used for comparison with the experimentally obtained values.

3. Specimens. Model and experimental setup

A specimen used in the experimental determinations is presented in Fig. 1.

Specimens of the same shape have been chopped by saw cutting in thick slices, small enough in order to achieve the geometric model used in the numerical analysis (Fig. 2)

Fig. 1 Specimen for impact test

Fig. 2 Specimen cut

Drawings of each slice were scanned and imported into SolidWorks software where the 3D virtual model has been obtained as in Fig. 3.

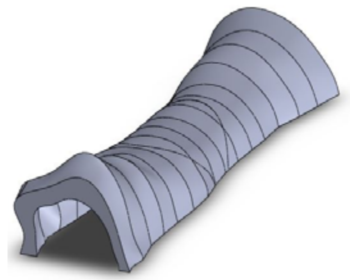

Fig.3 Geometric model

Experiments and calculation were performed for applying the shock by an impactor of mass of 4 kg, which falls freely from a height of 100mm, thus having an energy of 4 J and a speed of 1.41 m/s at the time of impact.

4. Experimental results

Fig. 4 presents the impact hammer of the INSTRON 8800 universal machine used to perform the experimental analysis and the clamping method for impact. Fig. 5 presents an impacted specimen.

Fig. 4 . The impact hammer and the clamping method for impact

Fig.5 Impacted specimen

Curve 3 from Fig. 6 presents the variation of the force in the impactor, for one of the tested specimens, as provided by the software of the testing machine, and curve 2 of the same figure represents its polynomial interpolation. Maximum force resulting from this test is 5004N. For the other tested samples, the maximum forces had similar values, as shown in Table 1.

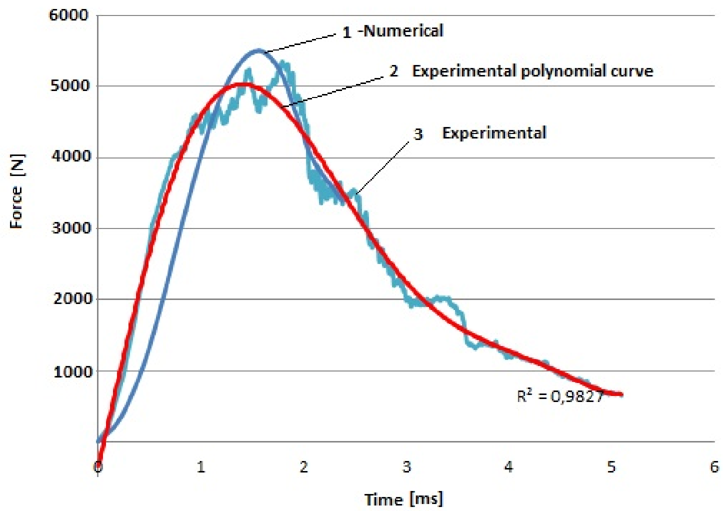

Fig. 6. Force variation over time

5. The numerical model

The finite element analysis was performed for the two bone elements almost identical to those for which experimental determinations were made(taken from the opposite leg)

Geometric model was imported into the module Explicit Dynamics of the ANSIS program [18], together with the impactor and the support plate models, both made of steel. The obtained finite element model for this assembly is shown in Fig. 7.

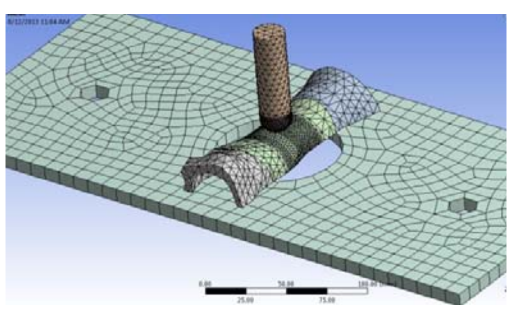

Fig.7 The numerical model

In the impact zone the mesh was refined, in order to better emphasize the steep stress concentration that occurs here.

The same boundary conditions as for the experimental determination were considered.

The elastic constants of the bone (Poisson's ratio $\nu = 0.38$ and dynamic Young's modulus $E_{dyn} = 20000$ MPa) were previously determined experimentally on the specimens made from the same investigated bone.

Because the process is followed until the destruction of the bone element, the numerical impact requires an incremental analysis process. Thus, several parameters were selected considering the results from the experimental tests. A 4.5 ms time interval was set for the program to complete the numerical calculus and also a 10^{-7} s increment was set. The speed applied to the impact hammer was set at v = 1.41 m/s and its mass to a value of 4 kg, resulting an impact energy of 4J, similar to the experimental value. The value of the mass was obtained by choosing an adequate mass density correlated with the volume of the model.

After processing, one obtains the distribution of stresses and displacements in the assembly for the entire duration of the shock. For example, in Fig. 8 is shown the map of the von Mises equivalent stress at a time t = 0.14 ms during the contact of the hammer with the specimen.

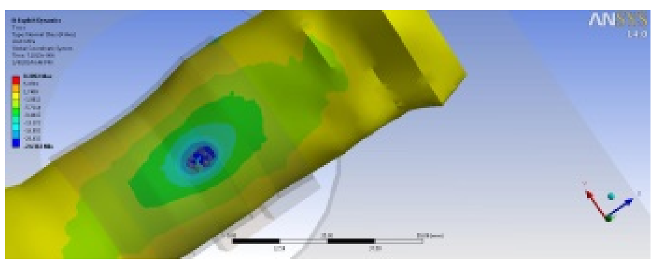

Fig. 8 Stress distribution at t = 0.14 ms

This model allows finding the stress distribution in any point of the assembly. Choosing a point in the direction of the impactor's axis and multiplying the stress in this point by the area of the cross section, we obtained the impact force. The variation of this force is represented by curve 1 in Fig. 6. The maximum force, read on this curve, is 5450 N.

For the second specimen model which was analyzed with the finite element method, similar in shape to specimen 2, from Table 1, we obtained a maximum force of 5631 N. in the impactor.

Table 1.

Comparison between the experimental and numerical values of the impact force

Specimen	Maximum force (N) (experimental)	Maximum force (N) (numerical)	Difference [%]
1	5004	5450	8.91
2	5209	5631	8.10
3	5514		
4	5541		
5	5240		

4. Conclusions

The results of numerical analyses are validated in this paper by comparing them with experimental data. The difference between the values calculated by applying the numerical model and those determined experimentally for the two investigated specimens are 8.91% and 8.1% respectively. The errors fit into the usual engineering approach of 10%

The numerical model developed for this impact calculus allows a detailed analysis of the bone tissue materials.

Due to the complexity of phenomena, numerical modeling requires experimental verification, difficult to achieve since experiments in vivo are almost prohibitive. Choosing a measurable parameter is important, given that the usual methods for experimental determination of the stresses are not easy to be applied.

Validated by the comparison between experimental and numerical impact force , the computing model can be used to study the impact on any element of bone, including tests in which the influences of the tegument layer are taken into account .

Stress distribution occurring in the bone tissue, allows studying the behavior of these type of material, both during and after an impact. The obtained information can be also used to design structures with a greater capacity to absorb impact energy regarding human safety gear.

R E F E R E N C E S

[1] *M.A. Crisfield*, Non-linear Finite Element Analysis of Solids and Structures, vol 1, Essentials, London, 2000

[2] *H.A. Petrescu, A. Hadăr*, Influence of Load Position on a Honeycomb Sandwich Using Nonlinear Analsys, Revista de Mat. Plast., vol. 48, nr. 2,Bucuresti 2011.

[3] *Audrey Auperrin.,* Caractérisation tissulaire pour la détermination du comportement de l'os crânien: essais mécaniques et imagerie médicale, PhD Thesis, L'Université de Valenciennes, 2009.

[4] *D. C. Wirtz, N. Schiffers*, T. Pandorf, K. Radermacher, D. Weichert, R Forst, Critical evaluation of known bone material properties to realize anizotropic FE simulations of the proximal femur. Journal of Biomechanics, vol. 33. USA Elsevier2000.

[5] *M. S. El-Asfoury, M. A. El-Hadek*, Static and Dynamic Three-Dimensional Finite Element Analysis of Pelvic Bone. International Journal of Mathematical, Physical, & Engineering Sciences, Vol. 3. 2009.
[6] *J. Travis*, Finite Element Analysis of Human Clavicle Bone. Ohio : Teza doctorat, The Ohio State University, 2012.
[7] *G. Belingardi, G. Chiandussi, I. Gaviglio*, Developement and validation of a new finite element model of human head. ESV Conferance. Washington,2005.
[8] *M. Claessens, F. Sauren, J. Wismans*, Modeling of the human head under impact conditions: a parametric study. 41st Stapp Car Crash Conference. Lake Buena Vista, Florida, USA, 1997.
[9] *D. Vlasceanu, Daniela Tudor, A. Hadar, H. Gheorghiu*, Static analysis using finite element method a mandible-strut plate assembly. 4th International Conference "Biomaterials, Tissue Engineering &Medical Devices. Sinaia, Romania, 2010.
[10] *M. Kasra, M.D. Grynpas*, Static and dynamic finite element analyses of an idealized structural model of vertebral trabecular bone. Journal of Biomechanical Engineering, Vol. 120,1998.
[11] *D. J. Janko, Lj. J. Miomir*, Finite Element Modeling of the Vertebra with Geometry and Material Properties Retreived from CT-Scan Data. Mechanical Engineering, Vol. 8. 2010.
[12] *G. Luo, M., S. C. Cowin, J. Kaufman,* Dynamic relationships of trabecular bone density, architecture, and strength in a computational model of osteopenia. Bone, Vol 18, New York Elsevier Inc ., 1996.
[13] *Ulrich, D., van Rietbergen, B., Weinans, H., P Ruegsegger,*. Finite element analysis of trabecular bone structure: a comparison of image-based meshing techniques. Journal of Biomechanics, Vol. 12, 1998.
[14] *M. F. Emer, P. McGarry*, Computational Study of Cortical Bone Screw Pullout using the Extended Finite Element Method (XFEM), Teza de doctorat, National University of Ireland, Galway, Ireland , 2012.
[15] *Abdel-Wahab, A. Adel, V. Silberschmidt*, Dynamic properties of cortical bone tissue: impact tests and numerical study. Mechanical and Manufacturing Engineering ,Vol. 180, Leicestershire, UK., 2011.
[16] *Y. Dokko,R. Anderson, J. Manavis, P. Blumburgs, J. McLean, L. Zhang, Yang, K., A. King,* Validation of the human FE model against pedestrian accident and its tentative application to the examination of the existing tolerence curve. Proceedings 18th International Technical Conference on the Enhanced Safety of Vehicle, 2003.
[17] *E. Budyn, T. Hoc*, Multiple scale modeling for cortical bone fracture in tension using X-FEM. PhD thesis, India, REMN, 2007.
[18] *** ANSYS 13, Finite Element System, User Guide, 2010