Bulletin of Environment, Pharmacology and Life Sciences
Bull. Env. Pharmacol. Life Sci., Vol 4 [11] October 2015: 66-71
©2015 Academy for Environment and Life Sciences, India
Online ISSN 2277-1808
Journal's URL:http://www.bepls.com
CODEN: BEPLAD
Global Impact Factor 0.533
Universal Impact Factor 0.9804

ORIGINAL ARTICLE **OPEN ACCESS**

Finite Element Model of the Human Head in order to Assess the Situation and Location of the Impact

S. Omidian[1]*, M. Namdarian[2],V. Salamat[3],
[1]Department of Engineering, Eshragh, Institute of Higher Education, Bojnord, Iran
[2]Department of Mechanical Engineering, Islamic Azad University, Nor Branch, Iran
[3]Department of Mechanical Engineering, Islamic Azad University, Nor Branch, Iran
Email: shahabomidian@eshragh.ac.ir

ABSTRACT

When a car accident and incident occurs in sport, impact loads occur on the head. Information about loads traumatic injury mechanism is not yet complete. Mathematical models provide a overused tool to analyze the mechanism of the head injuries, in particular finite element methods are considered the structure of the mathematical model of the head, each may explain the complex geometry. The purpose of this project is to develop a two-dimensional finite element model of the human head in order to assess the situation and location of the impact. Detailed geometric models may alter the structure of the model, as well as changes in mesh brain tissue and time step may be effective on the results. In this project the dynamic response of the adverse impact shall be reviewed by using the finite element in which a two-dimensional model of the human had in coronal section was modeled by using real anatomy. The model consists of three layers of skull (internal and external layers of compact bone, the middle layer of cancellous bone) , cerebrospinal fluid, brain membranes (Falks and tentorium) and brain tissue. This model is loaded with a sinusoidal pulse and maximum pressure of 40 Kg pulse. Simulation is performed by using finite element and ANSYS software version 9. The pressure profile and pressure distribution is the area of coup is studied.

Keywords: *Head injuries, finite element, viscoelastic, frequency response, contact structure*

Received 18.08.2015 Revised 02.09.2015 Accepted 24.09.2015

INTRODUCTION

The human head has a tough membrane (skull) , it is covered by a soft layer (scalp) . The central nervous system (CNS) includes the brain and the spinal cord .The brain is separated from the skull by meanings and mid sagittal (CFS) .More details about the anatomy of the human head may be found in the books [1] . Finite element models of the human head have been developed to a certain extent. The tow-dimensional models are simulated the cross-sectional shape of the human head when the load impact occurs.

Shugar [2] developed the plane strain model from the mid sagittal of the human head that includes a layer as the skull and the brain is fluid. He got the pressure gradient in which the coup pressure is three times higher that the counter-coup pressure. The counter pressure had been prepared the distribution of the pressure in the brain. Khalili et al [3] were made the different model of finite element of the head including scalp , skull and the brain.

They have achieved a linear pressure gradient in the fluid with compaction near the point of the impact. The model includes the skull, the brain and the cerebro-spinal fluid. He tested several times to reach the conclusion that the deformation of the skull is of the great importance in the simulation of the head injuries. Lee [4] imitated Margulies [5] , he used two-dimensional model of the full cylinder and semi-cylinder . This model was tested to check the angular acceleration. The relationship among shear strain in the gel-like material that represents the brain and the linear elastic properties of the brain was studied as a parameter.

The purpose of this papers us to determine the influence of mesh .time step in the impact loading and the place of the contact effects.

THE GEOMETRIC MODELING

Two-dimensional models:

Two-dimensional model formed of a skull and intracranial structures that include the brain. To reconstruct the geometry if the human head each used the visual data of the doctors.

These data sets include images of CT scan and MRI of the man's head. CT images and MRI show the different structural geometry of the head in details. The skull is modeled as a homogeneous isotropic structure with linear behavior. External and internal layers of the skull have the characteristics if the compact bone, the middle layer has the characteristics of the cancellous bone.

Mash generation:

In analysis and mechanical design, finite element mesh in general structure is done by using the pre-processor of finite element such as ANSYS.

We should describe the head model from the MRI data and actual dimensions of the human head ,we use the reference that used for the model for the fiftieth percentile of American male . These data have been digitized to make the borders of model. Different levels of the head including the skull , brain tissue levels, levels of the cerebrospinal fluid (in the brain) and a level indicating Falks and Tentorium membranes , it separates the levels of the brain tissue are made by these borders. Dimensional properties of the human head is shown in figure 1.

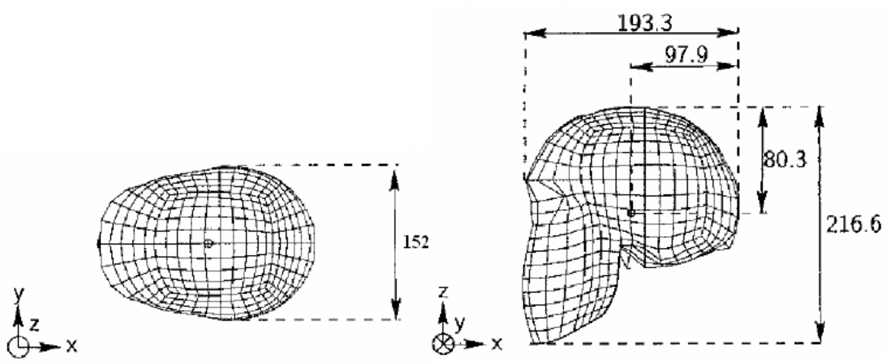


Figure 1- Geometry of the human head in mm.[13]

The model consists of three layer; skull , brain tissue , cerebro-spinal fluid levels , Falks and tentorium membranes that contact of the brain and skull shall be coupled.

Equations of the motion and contact:

Tension and strain that generated by a solid object attributable to the force, each may be expressed by the momentum balance equation:

Pb=pü (1)+Δ.σ

Which σ is stress tenso, b vector if volume force per mass unit, u relocation vector.

The finite element method starts with the new formulation of partial differential equation (2) that is equivalent to the differential equation .The formulation of the differential equation is solved in the domain Ω and the space Vh , it is measured by a simple function that depends on only limited several parameters. Since the space of the Vh function is limited an approximation of the exact solution shall be achieved . In this section finite element methods for solving the motion equation (2) are used when we assume that the behavior of materials are linear elastic.

The first step in the finite element method is to multiply the motion equation to a weighting function Ш ,

$$\int_{\Omega} \omega.(\nabla.\sigma \quad + \rho b - \rho \ddot{u})d\Omega = 0 \qquad (2)$$

An integration of the first semester with detailed method :

$$\int_{\Omega} \omega.(\rho \ddot{u})d\Omega + \int_{\Omega} (\nabla \omega)^{c} : c^{4} : \varepsilon(u)d\Omega = \int_{\Omega} \omega.(\rho b)d\Omega + \int_{\Gamma} \omega.(\sigma.n)d\Gamma \qquad (3)$$

Which Ţrepresentsthe boundary domain of Ω .

These equations are written in the weakened form. The equation includes first order derivatives against the second order derivative in equation (2) because the continuity needed to stress σ is reduced to a degree. Laterthe range Ω is divided into a few sub-region Nel .The relocation range u(x,t) in every element is obtained by interpolation between transport nodes un(t) using polynomial Nn(x) , that is called form or interpolation function .

(4)
$$u(x,t)=\sum_{n=1}^{N_{nd}^{e}} N_n(x).u_{n(t)}$$

N e nd represents the several element's nodes of e . Functions are selected in such a way that in node I, Ni (x) =1 , for other nodes , Ni(x)=0 .

To solve the equation(3) , functions are needed , that first stage derivative is non-zero and each may satisfy the continuity of the inner element. When we may use the Galerkinmethod , weighting function interpolates the same as the shift range.

$$\omega(x)=\sum_{n=1}^{N_{nd}^{e}} N_{n(x)}.\omega_n \quad (5)$$

By putting Equations (4) and (5) on (3) , calculated integral for each element , the process uses the standard finite element. The result of the process is as follows:

(6)
$$\underset{\sim}{\omega}{}_e^T\,\underline{M}_e\,\underset{\sim}{\ddot{u}}{}_e+\underset{\sim}{\omega}{}_e^T\,\underline{K}_e\,\underset{\sim}{u}{}_e=\underset{\sim}{\omega}{}_e^T\,\underset{\sim}{f}{}_e$$

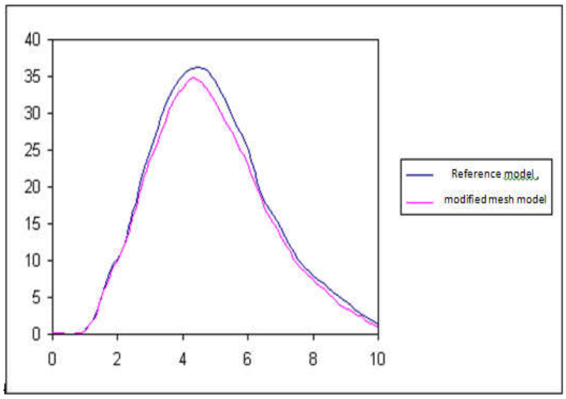
Where Me is the element mass matrix, K e element stiffness matrix and f e is the column for nodes' force in element e. This equation must be maintained for each weighting function w , and maybe w e is removed.

We repeat this approach to all element's range Ω, we may sum vectors and element matrix by general matrix , therefore the matrix equation system is given :

$$\underline{M}\,\underset{\sim}{\ddot{u}}+\underline{K}\,\underset{\sim}{u}=\underset{\sim}{f} \quad (7)$$

Checking the mesh :

The used elements have great influence in parametric studies. To find the effects of the elements on the results, we decrease the several model's elements from 6345 to 4767 and the rest of the conditions remain constant. The primary choice time step is 1.4*10 -3 and the same explained impact force is applied to model. The impacts tensions for both models with modified mesh are shown in figure (2).The measured tensions are similar to the initial tensions that means we select the right size of element.



Figure(2)-Evaluation of mesh size

Checking the time step:

In transient analysis, the minimum time step is used for improving the convergence. Two simulations are done to evaluate the effects of time step. First the time step decrease from 1.4*10 -3 to 1.4*10-4 , the second simulation is done by increasing the time step to 1.4*10-2 .

Figure 3 shows the results of the tension region that done by different time steps. In the simulation with the biggest time step negligible different of tensions in the impact area compared with the original model may be seen and the results in the smaller time step are consistent with the reference tension.

It may be concluded from this study that the initial time step must be small enough then the calculated response may be accurate.

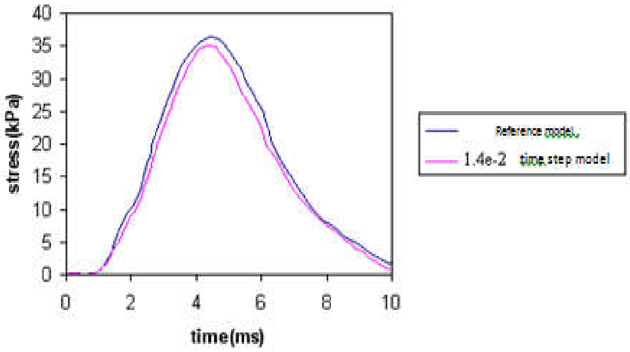


Figure 3 – checking the time step .

Simulation and comparison of the impact

Two- dimensional model has 10 milliseconds period of time .The time steps are calculated considering the relationship between the frequencies of the system.

Simulation was done with ANSYS code. In this section the results of the two-dimensional model is described and the result of the tension-time chart in the impact area was offered.

Maser von tension and main strain are used to specify the deformation in the brain caused by the stroke. The area of the calculated tension and tension-time chart are shown in Figure 4 and 5.

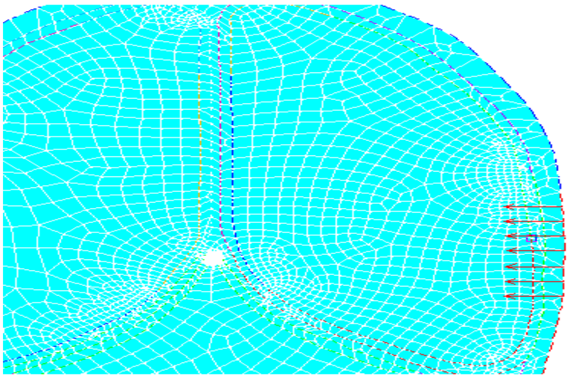


Figure 4 –Defined node in the impact area to see the results.

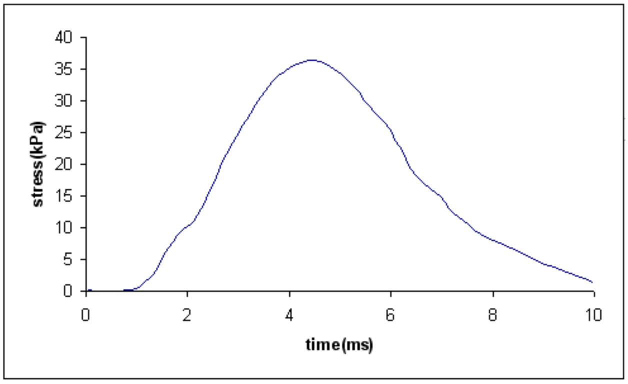
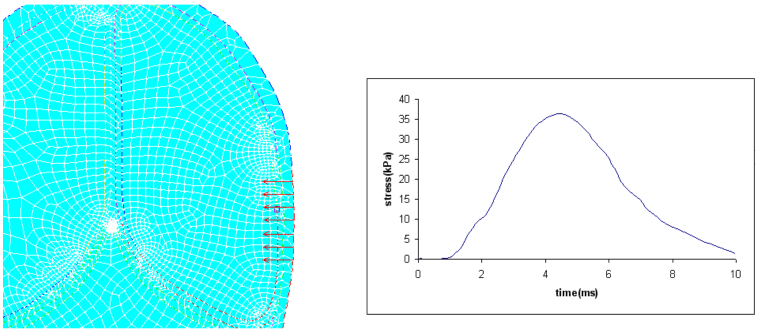


Figure 5_ Tension-time response in the impact area for the reference model.

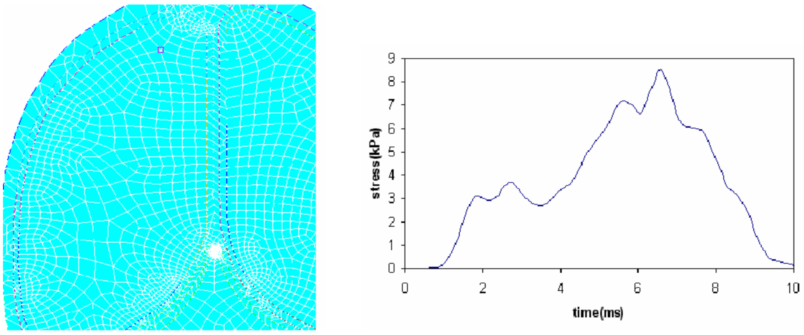
Checking the depth of the injury:

In order to evaluate the damage severity and determine the vulnerability points, the tension-time chart in the different areas of the brain tissue is studied. Although we anticipate the maximum tension occurs in the impact area, but the results indicate that the areas of the brain that has incompatible geometry are at the greatest damage.

This area in particular is located in the upper occipital lobe. Lower tensions may be seen in the frontal regions.



A-Lateral area of the right lobe of the brain.



B-Frontal area of the left lobe of the brain.

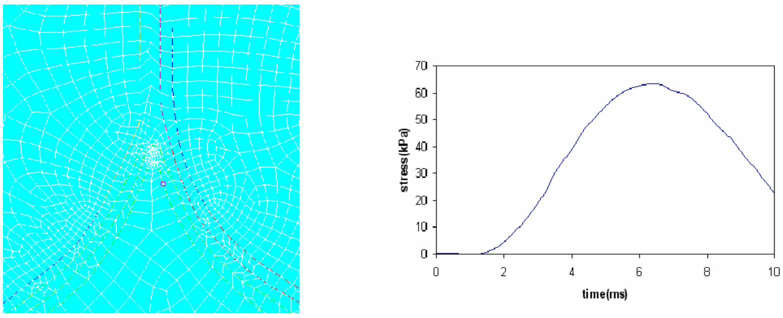


Figure 6 _Assessment of damage in the different brain areas.
A-Lateral area of the right lobe of the brain
B-Frontal area of the left lobe of the brain
C-Upper area of the occipital lobe.

CONCLUSION

Comparisons in this section are between the calculated numerical results and the articles that means they're not completely valid models but looking for suitable agreement with the results presented in other articles.

Frequency analysis results obtained from the model matched well with the values provided by Ruan [6].

Frequencies calculated in the frequency analysis of the brain are well matched with the frequencies calculated in two-dimensional models of researchers.

The time step changes show that the initial time step must be small enough.

Mesh reform represents too small improvement in results compared to the reference model. The influence of mesh size on the results indicates a decrease in the amplitude of the tension on the impact area as much as 6% compared to the reference model. The results indicate that areas of the brain that are a heterogeneous geometry are at the greatest damage. This area in particular is in the upper part of occipital lobe. The minimum tension is observed in the frontal area.

REFERENCES

1. Bogduk. N., (1986).Anatomy and pathophysiology of whiplash, Clinical Biomechanics, 1:92–101.
2. Shugar, T. A.,(1977). Finite Element Head Injury Model", Report No. DOT HS-289-3-550. TA, Vol. I.
3. Khalil, T. B. and Hubbard, R. P., (1977). Parametric Study of Head Response by Finite Element Modeling", *Journal of Biomechanics,* Vol. 10, pp. 119-132.
4. Lee, E. S., (1990). A Large-strain, Transient-dynamic Analysis of Head Injury Problems by the Finite Element Method", Ph.D. dissertation, Georgia Institute of Technology, October.

5. Margulies, S. S., (1987).Biomechanics of Traumatic Coma in the Primate", Ph.D. dissertation, University of Pennsylvania.
6. Ruan, J.S, T. Khalil and A.I. King (1991).Human head dynamic response to side impact by finite element modeling .Journal of Biomechanical Engineering, 113, 276-283

CITATION OF THIS ARTICLE

S. Omidian, M. Namdarian,V. Salamat. Finite Element Model of the Human Head in order to assess the Situation and Location of the Impact. Bull. Env. Pharmacol. Life Sci., Vol 4 [11] October 2015: 66-71