A Realistic Subject-Specific Finite Element Model of Human Head-Development and Experimental Validation

Kwong Ming Tse[1, *], Long Bin Tan[1], Shu Jin Lee[2], Siak Piang Lim[1, 3], Heow Pueh Lee[1, 3, *]

[1] Department of Mechanical Engineering, National University of Singapore, Singapore
[2] Division of Plastic, Reconstructive and Aesthetic Surgery, National University Hospital, Singapore
[3] National University of Singapore (Suzhou) Research Institute, China

Abstract— **Head injury, being one of the main causes of death or permanent disability, continues to remain as a major health problem with significant socioeconomic costs. Therefore, there is a need for biomechanical studies of head injury. To assess the biomechanics of head injury mechanism, many finite element head models (FEHMs) had been built. However, in order to reduce the computation efforts, most of these FEHMs were simplified and details of complex head anatomical features are often ignored in modeling. The main purpose of the present work is to build and validate a detailed finite element model of human head in order to better predict the mechanical responses of the human head during head injury. Geometrical information of a human head is obtained from medical images of computed tomography (CT) and magnetic resonance imaging (MRI) with the use of image-processing software, for segmentation and reconstruction of a comprehensive FEHM. The head model is then validated against both intracranial pressure (ICP) data of the two experimental cadaver tests. General shape trends, magnitudes and duration of the pressure pulses in the simulation agree well with the experimental pressure pulse. Overall, there is a good correlation between the simulations and the experiments. Once being validated, this representative FEHM can be used in the assessment of the injurious effects of different loading conditions and enable the development of enhanced head injury and protection equipments through the reconstruction of the available real-world accidents information.**

Keywords— **Finite element (FE), head injury, head model, validation**

I. INTRODUCTION

Head injury, being one of the main causes of death or permanent disability, continues to remain as a major health problem with significant socioeconomic costs. Therefore, there is a need for biomechanical studies of head injury. To assess the biomechanics of head injury mechanism, many finite element head models (FEHMs) had been built, however, some of them had not been validated. Experimental validation of a numerical model is necessary in determining the degrees to which the model accurately predicts the real-world phenomena well. Only when this is achieved, the model does represent a powerful tool with which to correlate the biomechanical parameters involved in the head injury with the clinical observations. Most of the FEHMs [1-8] had been validated against the intracranial pressure (ICP) data of Nahum et al. [9]'s cadaver experiments, with some [3-5, 10] having been validated against another ICP data of Trosseille et al. [11]'s longer duration impact. However, some of these earlier idealized FEHMs, which were developed mainly for automobile related head injuries, are generally unrealistic with simplifications in the facial details. The objective of this study is to build a detailed and realistic model of subject-specific head using computed tomography (CT) and magnetic resonance imaging (MRI), and to validate it against both ICP data of the experimental cadaver tests under different impact conditions.

II. METHODS AND MATERIALS

A. Model Description

Geometrical information of the human skull was obtained from 460 computed tomography (CT) axial images of a middle-aged male with in-plane resolution of 512 by 512 pixels with a pixel size of 0.488 mm and slice thickness of 1.0 mm. As for the intracranial contents, the magnetic resonance imaging (MRI) data of the brain, with in-plane resolution of 1659 by 962 pixels with a pixel size of 0.500 mm and slice thickness of 4.0 mm, was employed for the segmentation of the brain (Fig. 1). These medical images were imported into Mimics v13.0-v14.0 (Materialise, Leuven, Belgium) for segmentation and reconstruction of the FEHM (Fig. 1). A semi-automatic meshing technique was employed in HyperMesh v10.0 (Altair HyperWorks, Troy, MI, USA) to optimize between computational efficiency and element quality, with the average element size of 1.57 mm, as well as aspect ratio of 1.61 for the model.

The FEHM consisted of a cranial skull with detailed facial bone features, teeth, cervical vertebrae, nasal septal cartilage, nasal lateral cartilages; brain components such as cerebral white matter and gray matter, cerebellum and brainstem, the cerebral peduncle (midbrain), and the entire ventricular system, as well as the cerebrospinal fluid (CSF)

J. Goh (ed.), *The 15th International Conference on Biomedical Engineering*, IFMBE Proceedings 43,
DOI: 10.1007/978-3-319-02913-9_78, © Springer International Publishing Switzerland 2014

separating the skull and the brain; lastly soft tissue overlaying the skull. The entire model is approximately 4.73 kg and consisted of 327536 nodes and 1337903 linear hexahedral elements (Fig.1).

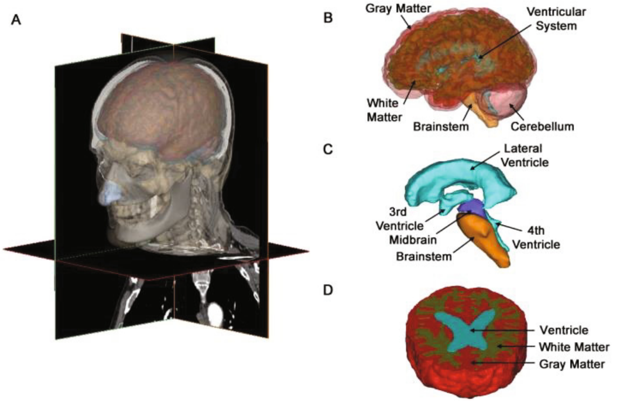


Fig. 1: The subject-specific finite element model of human head.

B. Material Properties

Like most previous FEHMs, all the skeletal tissues such as cartilages, teeth and cervical vertebrae are modeled as linear elastic, isotropic materials it has been considered to have homogeneous and isotropic behavior [12-14] (Table 1). As for the brain tissues, it has adopted a linear viscoelastic material behavior combined with the large-deformation theory [15-17] (Table 1).

Table 1: Material properties of the various parts in the head model.

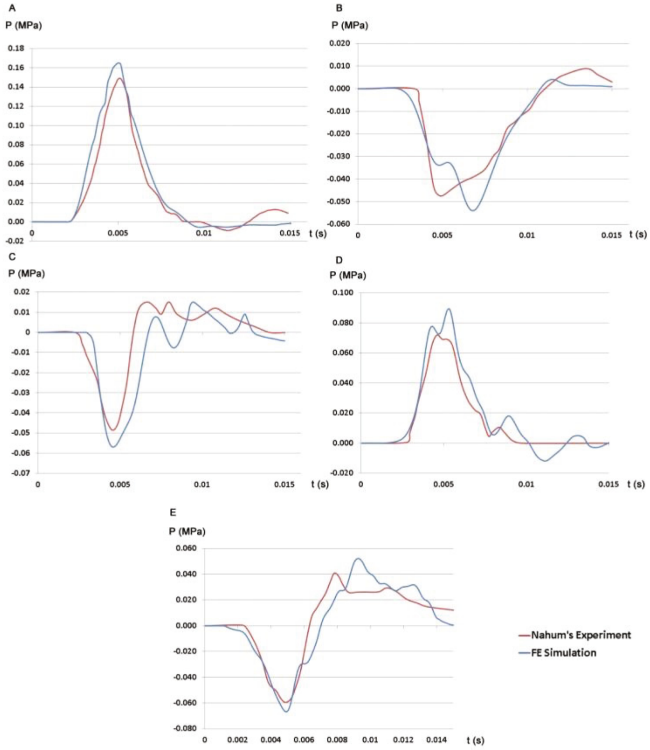
Parts	Young's Modulus, E (MPa)	Poisson's ratio, υ	Density, ρ (kg/mm^3)	Sources
Brainstem	G_0 = 0.0225 MPa, G_∞ = 0.0045 MPa, β = 80 s^{-1}	0.4996	1.06E-06	[10]
Cerebral Peduncle	G_0 = 0.0225 MPa, G_∞ = 0.0045 MPa, β = 80 s^{-1}	0.4996	1.06E-06	[10]
Cerebellum	G_0 = 0.528 MPa, G_∞ = 0.168 MPa, β = 35 s^{-1}	0.48	1.14E-06	[3, 17-20]
CSF	E = 1.314	0.4999	1.04E-06	[21, 22]
Gray Matter	G_0 = 0.034 MPa, G_∞ = 0.0064 MPa, β = 700 s^{-1}	0.4996	1.04E-06	[21, 22]
Lateral Cartilage	E = 30	0.45	1.50E-06	[23]
Septum Cartilage	E = 9	0.32	1.50E-06	[24]
Bone	E = 8000	0.22	1.21E-06	[22]
Soft Tissues	E = 16.7	0.46	1.04E-06	[22, 25]
Ventricles	G_0 = 0.101 MPa, G_∞ = 0.00101 MPa, β = 100 s^{-1}	0.49	1.08E-06	[22]
White Matter	G_0 = 0.041 MPa, G_∞ = 0.0078 MPa, β = 700 s^{1}	0.4996	1.04E-06	[21]

C. Boundary Conditions

Both the skull-CSF and brain-CSF interfaces are modeled as contact pairs with a tangential sliding boundary condition with the coefficient of friction of 0.2 [5, 25, 26] and a normal hard contact pressure-overclosure condition. All the interfaces between other intracranial contents and those between skull, cartilages and soft tissues are implemented with tie-constraints. The interaction between the head and the foreign impactor is defined by a contact algorithm, which has hard contact pressure-overclosure with default constrain enforcement method [27].

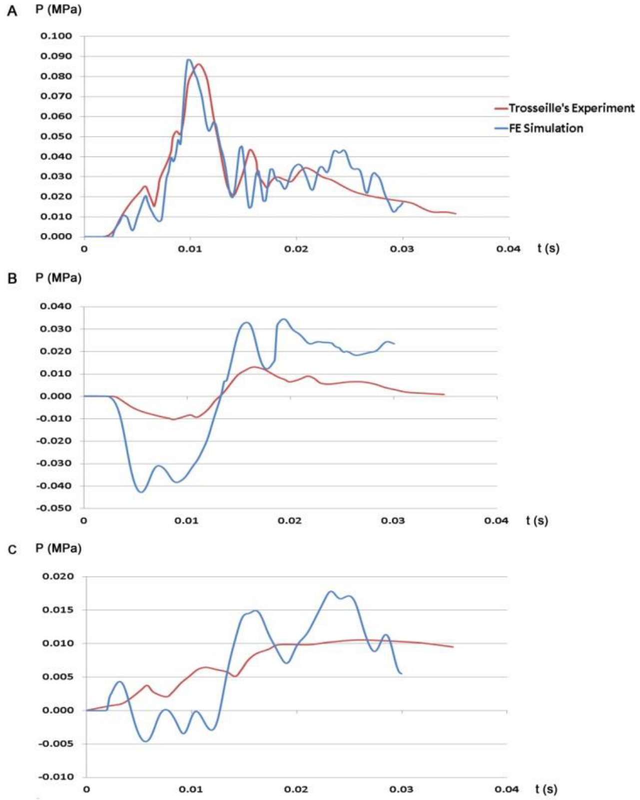
III. RESULTS AND DISCUSSIONS

The FEHM is first validated against the ICP-time histories of Nahum et al. [9]'s cadaver experiment 37, in which the FEHM is impacted by a cylindrical mass of 5.59 kg at frontal bone region and the impact velocity is 9.94 m·s^{-1}.



In comparison with Nahum et al. [9]'s experimental pressures, the simulate results have generally agree well with the experimental pressures, in terms of magnitudes and duration of the pulses, except for the great variation in bilateral occipital pressures. This could probably be due to the uneven distribution of white and gray matters in the model. Additionally, there are small oscillations in the simulated frontal and parietal pressures.

Then, the FEHM is impacted by a 23.4 kg impactor, travelling at 7 $m \cdot s^{-1}$ and hit at the nasal region, similar to the Trosseille [11]'s cadaveric experiment MS 428-2. The ICP history plots for the frontal, occipital and temporal regions, are shown in Fig. 3.



The simulated pressure at the frontal lobe agrees well with the experimental pressure. However, this is not true for the occipital pressure and temporal pressure, in particular, the occipital pressure in which its peak was overestimated by three-folds. Overall, the general trend of predicted pressure pulses matched reasonably well with the Trosseille [11]'s cadaveric experiment.

Up to date, to the authors' knowledge, there is no reported subject-specific FEHM, with such detailed features including soft tissues, being validated against these cadaveric tests. In the present study, a realistic FEHM, with detailed anatomical features, have been developed and validated against the ICP data of the two cadaver experimental tests. The comparisons of the simulated results are largely consistent and are in good agreement with the experimental measured ICP and relative displacements. Despite the fundamental differences in the numerical formulation in this FEHM as compared to others FEHMs [1-8], there exist little evident differences in the predicted ICP when comparing with the experimental ICP. This may indicate that the advancements on the details of the extracranial features and overlying soft tissue would not improve the model's predicting capabilities in brain injury as it seems that the prediction replies more on skull's mass properties and kinematics rather than its geometrical details. As a corollary, a model with the attributes of simplified extracranial features may be sufficient for modeling traumatic brain injury (TBI). Nevertheless, this newly developed FEHM, with detailed realistic anatomical features, can be used in the evaluation of either facial or brain injury.

ACKNOWLEDGMENT

The authors would like to acknowledge the support by a grant from the Swiss based CMF Clinical Priority Program of the AO Foundation under the project number C-09-2L. Special thanks to Dr. Svein Kleiven (Division of Neuronic Engineering, School of Technology and Health, Kungliga Tekniska Högskolan (KTH), Stockholm, Sweden) for sharing some technical information about the impactor padding.

REFERENCES

1. Ueno, K., Melvin, J. W., Lundquist, E. and Lee, M. C. (1989). Two-dimensional finite element analysis of human brain impact responses: application of a scaling law. *American Society of Mechanical Engineers (ASME), Applied Mechanics Division (AMD)* 106, pp. 123-124.
2. Ruan, J. S., Khatil, T. B. and King, A. I. (1993). Finite element modeling of direct head impact. In: *Proceedings of 37th Stapp Car Crash Conference*, San Antonio, USA: Society of Automotive Engineers (SAE), SAE Paper No. 933114.
3. Willinger, R., Kang, H. S. and Diaw, B. (1999). Three-dimensional human head finite-element model validation against two experimental impacts. *Annals of Biomedical Engineering* 27(3), pp. 403-410.
4. Zhang, L., Yang, K. H., Dwarampudi, R., Omori, K., Li, T., Chang, K., Hardy, W. N., Khalil, T. B. and King, A. I. (2001). Recent advances in brain injury research: a new human head model development and validation. In: *Proceedings of 45th Stapp Car Crash Conference*, San Antonio, USA, pp. 369-394: Society of Automotive Engineers (SAE), SAE Paper No. 2001-22-0017.
5. Kleiven, S. and Hardy, W. N. (2002). Correlation of an FE model of the human head with local brain motion–consequences for injury prediction. In: *Proceedings of 46th Stapp Car Crash Conference*, Ponte Vedra, USA, pp. 123-144: Society of Automotive Engineers (SAE), SAE Paper No. 2002-22-0007.
6. Takhounts, E. and Eppinger, R. (2003). On the development of the SIMon finite element head model. In: *Proceedings of 47th Stapp Car Crash Conference*, San Diego, USA, pp. 107-133: Society of Automotive Engineers (SAE), SAE Paper No. 03S-04.
7. Belingardi, G., Chiandussi, G. and Gaviglio, I. (2005). Development and Validation of a New Finite Element Model of Human Head. In:

Proceedings of 19th International Technical Conference on the Enhanced Safety of Vehicles, Washington, USA, pp. 1-9: Transportation Research Board of the National Academies (TRB), TRB Paper No. 05-0441.

8. Horgan, T. J. and Gilchrist, M. (2003). The creation of three-dimensional finite element models for simulating head impact biomechanics. *International Journal of Crashworthiness* 8(4), pp. 353-366.
9. Nahum, A. M., Smith, R. and Ward, C. C. (1977). Intracranial pressure dynamics during head impact. In: *Proceedings of 21st Stapp Car Crash Conference*, San Diego, USA, pp. 339–366: Society of Automotive Engineers (SAE), SAE Paper No. 770922.
10. Horgan, T. J. and Gilchrist, M. D. (2004). Influence of FE model variability in predicting brain motion and intracranial pressure changes in head impact simulations. *International Journal of Crashworthiness* 9(4), pp. 401-408.
11. Trosseille, X., Tarriere, C. and Lavaste, F. (1992). Development of a FEM of the human head according to a specific test protocol. In: *Proceedings of 30th Stapp Car Crash Conference*, Warrendale, USA, pp. 235-253: Society of Automotive Engineers (SAE), SAE Paper No. 922527.
12. Hardy, C. H. and Marcal, P. V. (1971). Elastic analysis of a skull. *American Society of Mechanical Engineers (ASME) Transaction*, pp. 838-842.
13. Nickell, R. and Marcal, P. (1974). In vacuo model dynamic response of the human skull. *Journal of Engineering Industry* 4, pp. 490-194.
14. Ward, C. C. and Thompson, R. B. (1975). The development of a detailed finite element brain model. In: *Proceedings of 19th Stapp Car Crash Conference*, New York, USA, pp. 641-674: Society of Automotive Engineers (SAE), SAE Paper No. 751163.
15. DiMasi, F., Marcus, J. H. and Eppinger, R. H. (1991). 3D anatomic brain for relating cortical strains to automobile crash loading. In: *Proceedings of 13th International Technical Conference on Experimental Safety Vehicles*: Paper No. 91–S8–O–11.
16. Ruan, J. S. (1994). *Impact biomechanics of head Injury by mathematical modeling.* PhD Dissertation. Detroit: Wayne State University.
17. Turquier, F., Kang, H. S., Trosseille, X., Willinger, R., Trosseille, X., Lavaste, F., Tarriere, C. and Domont, A. (1996). Validation study of a 3D finite element head model against experimental data. In: *Proceedings of 40th Stapp Car Crash Conference*, Albuquerque, USA, pp. 283-293: Society of Automotive Engineers (SAE), SAE Paper No. 962431.
18. Stalnaker, R. L. (1969). *Mechanical properties of the head.* PhD Dissertation. Morgantown: West Virginia University.
19. Shuck, L. Z. and Advani, S. H. (1972). Rheological response of human brain tissue in shearing. *J. Basic Eng* 94, pp. 905-911.
20. Yoganandan, N., Li, J., Zhang, J. and Pintar, F. A. (2009). Role of falx on brain stress-strain responses. In: Kamkim, A. and Kiseleva, I. (Eds.) *Mechanosensitivity of the Nervous System*. pp. 281-297: Springer Netherlands.
21. Al-Bsharat, A. S. (2000). *Computational analysis of brain injury.* PhD Dissertation. Detroit: Wayne State University.
22. Zhang, L., Yang, K. H. and King, A. I. (2001). Comparison of brain responses between frontal and lateral impacts by finite element modeling. *J Neurotrauma* 18(1), pp. 21-30.
23. Westreich, R. W., Courtland, H. W., Nasser, P., Jepsen, K. and Lawson, W. (2007). Defining nasal cartilage elasticity: biomechanical testing of the tripod theory based on a cantilevered model. *Arch Facial Plast Surg* 9(4), pp. 264-270.
24. Grellmann, W., Berghaus, A., Haberland, E. J., Jamali, Y., Holweg, K., Reincke, K. and Bierögel, C. (2006). Determination of strength and deformation behavior of human cartilage for the definition of significant parameters. *Journal of Biomedical Materials Research Part A* 78A(1), pp. 168-174.
25. Kleiven, S. (2006). Evaluation of head injury criteria using a finite element model validated against experiments on localized brain motion, intracerebral acceleration, and intracranial pressure. *International Journal of Crashworthiness* 11(1), pp. 65-79.
26. Miller, R. T., Margulies, S. S., Leoni, M., Nonaka, M., Chen, X., Smith, D. H. and Meaney, D. F. (1998). Finite element modeling approaches for predicting injury in an experimental model of severe diffuse axonal injury. In: *Proceedings of 42nd Stapp Car Crash Conference*, Tempe, USA, pp. 155-166: Society of Automotive Engineers (SAE), SAE Paper No. 983154.
27. Abaqus (2010). *Abaqus Analysis User's Manual*. Version: 6.10: Dassault Systèmes Simulia Corp.

Author: Kwong Ming Tse
Institute: National University of Singapore
Street: 1 Engineering Drive 2, Singapore 117576
City: Singapore
Country: Singapore
Email: tsekm@nus.edu.sg / tsekm.research@yahoo.com (Tse, KM) / mpeleehp@nus.edu.sg (Lee, HP)