3-D Finite Element Model for Analysis in the Maxilla and Zygomatic Bones Impact

Pattaraweerin Woraratsoontorn [1] and **Pitikhate Sooraksa**[2],

ABSTRACT

Concussion or head impact involves the brain injury from mild to severe. Most of the brain injury mechanism and brain lesions can be investigated from medical imaging. Alternatively, finite element model (FE model) of head impact is used for simulation and analyzing the brain concussion. This paper proposes finite element model of maxilla bone, left and right zygomatic bones impact for further analysis of effect of punch on the human skull. The finite element model is three-dimensional which solid Lagrangian elements and the explicit dynamics simulations were used for analysis. The human skull model used in this study was validated from the previous study. A foreign object impact generated contact force was used via load curve and initial velocity to estimate the contact force between the object and impacted bones. The impact force and stress distributions on the bones have been evaluated. The prediction of the bones response to the strike can be achieved using ANSYS LS-DYNA solver.

Keywords: Bone Impact; Finite Element; Head Impact; Brain Injury, Concussion

1. INTRODUCTION

Concussion or brain concussion is an injury to the brain caused by a blow to the head or by violent jarring or shaking. Sports are among the most common causes of concussion, and sports with the most physical contact, such as football, boxing, and hockey, are most likely to produce head injuries that involve concussion. Concussions can be mild or severe depending on the mechanism of injury.

The American Association of Neurological Surgeons says that 90% of the boxers sustain a brain injury. Boxing may account for fewer deaths than some other sports but a number of boxers suffering brain damage are believed to be much higher than recorded. Being hit on the head can cause fractures to the bone of the head and face and tissue damage in the brain. A punch can damage the surface of the brain, tear nerve networks, cause lesions, bleeding and sometimes produce large clots within the brain [1]. The knockout has been the goal of the boxing. It involves the death, a small sleep and a deep sleep. Jordan, who has published original research on neurological aspects of boxing, demonstrates that a cause of a knockout is rotational acceleration, a spinning of the brain. During a knockout, the brain stem doesn't move, but the spin at the top of the brain causes to lose consciousness. Professional boxers can deliver punch with such force to the movable head that the brain strike against the skull, tearing nerve fibers, the meningeal sac that supports the brain and blood vessels. The direction and power of the punch determines the severity of this tearing [2]. The recently most of the brain injury mechanism and brain lesions can be investigated from medical imaging i.e. CT and MRI.

Many researchers have been attempting to study the response of a human head to the object impact because the human head subjected to foreign object impact is one of the major causes of head injuries. Shi Wei Gong et al.[3] studied an approach for the estimation of contact force on a human head induced by a foreign-object impact. A simple head striker model to estimate the contact force between the human head and the foreign-object striker was proposed. Moreover, they analyzed the dynamic response of a human head to a foreign-object impact [4]. Chu et al.[5] studied traumatic brain injury using finite element analysis. 2D brain injury model was presented in their work. Svein Kleiven [6] developed a detailed and parameterized 3D finite element model of the human head to evaluate the effects of head size, brain size on the same acceleration impulse and impact directions on translational impulse. Gerald Krabbel and Ralph Mller [7] described the development of a 3D finite element model based on the digital data set from the head section. The fresh CT scans were used for the skull model and the MRI images for the brain model. It is expected that the model will be able to predict the risk of head injuries in a crash event. Giovanni Belingardi et al.[8] developed and validated a finite element the intracranial pressure and stress distribution due to a frontal impact. Ranganatha Rao Mulabagula [9] modeled the human skull to understand the stress distribution across the human skull using finite element analysis. An automobile chassis crash has been simulated to validate the load applied in the analysis. Xianfang Yue et al.[10] investigated the dynamic characteristics of the human skull-dura mater

Manuscript received on September 2, 2012 ; revised on October 24, 2012.

[1] Pattaraweerin Woraratsoontorn is with Department of Electrical Engineering, Faculty of Engineering, King Mongkuts Institute of Technology Ladkrabang, Bangkok, 10520, Thailand, E-mail: oaw2520@hotmail.com

[2] Pitikhate Sooraksa is with Department of Computer Engineering, Faculty of Engineering, King Mongkuts Institute of Technology Ladkrabang, Bangkok, 10520, Thailand, E-mail: kspitikh@kmitl.ac.th

system. 3D finite element model of a human skull was constructed to calculate the deformation of human skull with the intracranial pressure changing. P. Pongpanitanont et al.[11] developed software to analyze dyslexia brain fMRI images. 3D reconstruction was used for presenting an image.

Alternatively, finite element model (FE model) of skull impact is used for simulation and analyzing the brain concussion. At present, a realistic model of the skull impact to demonstrate causing a knockout has not been found. In this paper proposes the finite element model of impacted facial bones i.e. maxilla, left and right zygomatic bones for analyzing causing a knockout and investigating a maximum weak position on a human face in the future. The stress distribution on the different components of skull has been considered while a foreign object was blowing to those bones.

2. MATERIAL AND METHOD

2.1 Numerical Model

The finite element simulation of skull impact has been performed with ANSYS LS-DYNA the commercial software package.

For linear elasticity, stress given by Hook's law

$$\dot{\sigma_i} = \lambda\left(\frac{\dot{V}}{V}\right) + 2G\dot{\varepsilon}_i \tag{1}$$

$i = 1, 2, 3, \ldots$ where λ is Lame's constant and G is known as shear modulus.

The principal stresses (σ_i) can be decomposed into a hydrostatic and a deviatoric component

$$\sigma_i = -P + s_i \tag{2}$$

$$P = -\frac{1}{3}\left(\sigma_1 + \sigma_2 + \sigma_3\right)$$

$$P = K\frac{\sigma}{\sigma_0} - 1 \tag{3}$$

$$ds_i = 2G\left(d\varepsilon_i - \frac{dV}{3V}\right)$$

The explicit dynamics simulation, the equilibrium equations in dynamic analysis can be written in the form

$$[M]\left[\ddot{u}^{(i)}\right] = \left[F^{(i)}\right] - \left[I^{(i)}\right] \tag{4}$$

where $[M]$ is mass matrix, F is the vector of externally applied load and I is the vector of inertia forces. The mathematically equilibrium relation is a system of linear differential equations of second order. The solution can be obtained by finite difference expression to approximate the accelerations and velocities in terms of displacement which can be used written as

$$\ddot{u}^{(i)} = \frac{1}{\Delta t^2}\left(u^{(i+1)} - 2u^{(i)} + u^{(i-1)}\right) \tag{5}$$

The error of calculation depends on stable time increment that as relation below

$$\Delta t_{stable} = min\left(\frac{L_c}{c}\right) \tag{6}$$

where L_c is limited element edge length and c is velocity of longitudinal wave for an element is in the form

$$c = \sqrt{\frac{\lambda + 2\mu}{\rho}} \tag{7}$$

λ and μ are Lame's constants can be written in terms of young's modulus and poisson's ratio following

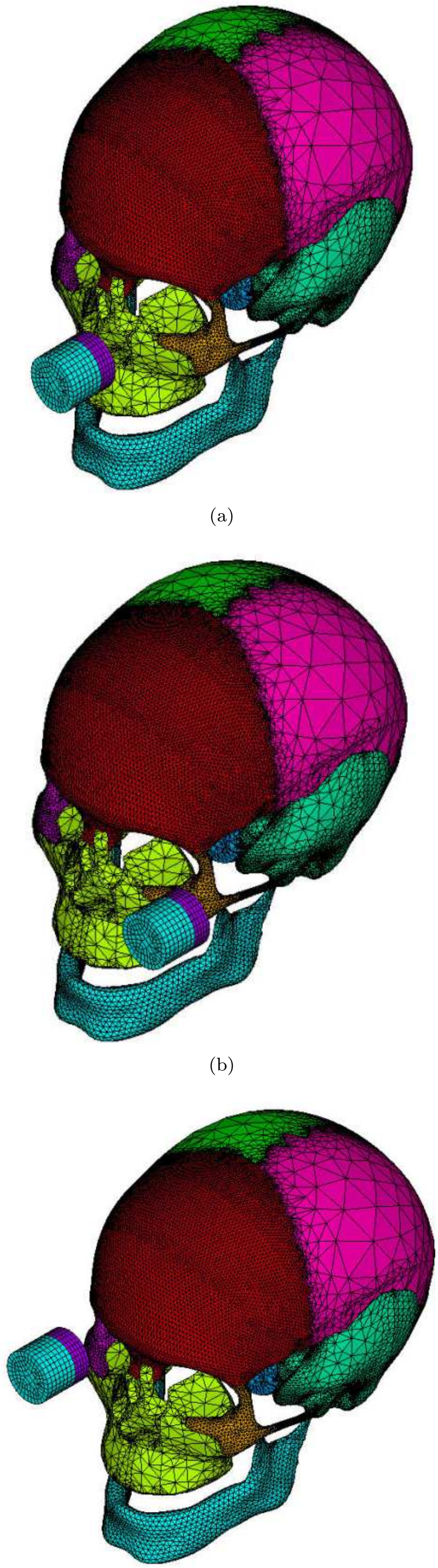
$$\lambda = \frac{\nu E}{(1+\nu)(1-2\nu)} \tag{8}$$

$$\mu = \frac{E}{2(1+\nu)} \tag{9}$$

The stable time expression mentioned for only one element in practically the ANSYS LS-DYNA solver automatically calculates the minimum time step for each element based on its characteristic length and density. The smallest of these element time steps is called the critical time step. The actual time step used during solution is the product of the current critical time step and a stability factor (usually 0.90). As elements distort during the analysis, their time steps are recalculated. However, an element's time step is calculated based on its material properties (E,ν,ρ) and characteristic length. The equation can be rearranged to find the required density of each element for a desired time step size. By adding the corresponding mass to these elements, the solution time will be reduced. This procedure is known as mass scaling and not recommend. In this paper, mass was not added to speed up run.

2.2 Finite Element Model Description

In this study, the geometrical realistic model of the skull has been finalized from native SolidWorks CAD3D commercial software. ANSYS LS-DYNA generated mesh and solved the numerical model. A three-dimensional (3-D) finite element (FE) model of facial bones impact on the different positions i.e. maxilla bone, left and right zygomatic bones are shown in figure 1. The model was composed of 364,892 elements and 87,023 nodes. The impact area and interface of bone (suture of skull) have high density of element to achieve realistic behaviour during impact. However, the human skull used in this study was evaluated from the previous study.



(a)

(b)

(c)

Fig.1:: 3-D Finite Element model of the impacted (a) maxilla bone (b) left zygomatic bone and (c) right zygomatic bone.

Table 1:: The sex, weight and height of each subject.

Bone	Material Model	ρ (kg/m^3)	E (MPa)	ν
Compact bone	Linear elastic	1800	15000	0.21
Facial bone	Linear elastic	4500	10000	0.3
Foreign object	Linear elastic	5304	210000	0.3
Covering layer	Elastic-plastic	1050	1500	0.3

2.3 Boundary Conditions

The skull considered in this study consists of two main components that were compact bones and facial bones. The frontal, parietal, temporal and occipital bones have been regarded as the first component. The zygomatic, nasal, maxilla and mandible bones were another. All of the bones have been modeled with linear-elastic behavior as proposed by Giovanni et al [8] that has been used as the reference. A foreign object was covered by a layer in the end where was surface contact to the bone. The linear-elastic behaviour can be illustrated in (3). Mechanical properties of the different components used in this FE model are shown in table 1.

The model has been considered as free in correspondence of the neck because the impact phenomenon is too fast to be influenced by neck constraints as Giovanni et al. In this facial bones impact simulation, all of the components were separated into twelve parts i.e. left and right zygomatic bones, a nasal bone, a maxilla bone, a mandible bone, a frontal bone, left and right parietal bones, left and right temporal bones, an occipital bones and a foreign object. The others tissues such as scalp, brain, ventricle, cerebro spinal fluid were not considered in this study. The object impact was defined to be mass 5.6 kg by setting up the speed at V = 7000 mm/sec (7 m/sec). All of the parts were defined as assembly contact with coefficient of friction (COF) 0.3. Contact between the object and frontal bone were identified COF = 0.2. The impact force and Von-Mises stress distributions on the different components of skull has been considered while a foreign object was blowing to maxilla bone, left and right zygomatic bones directly.

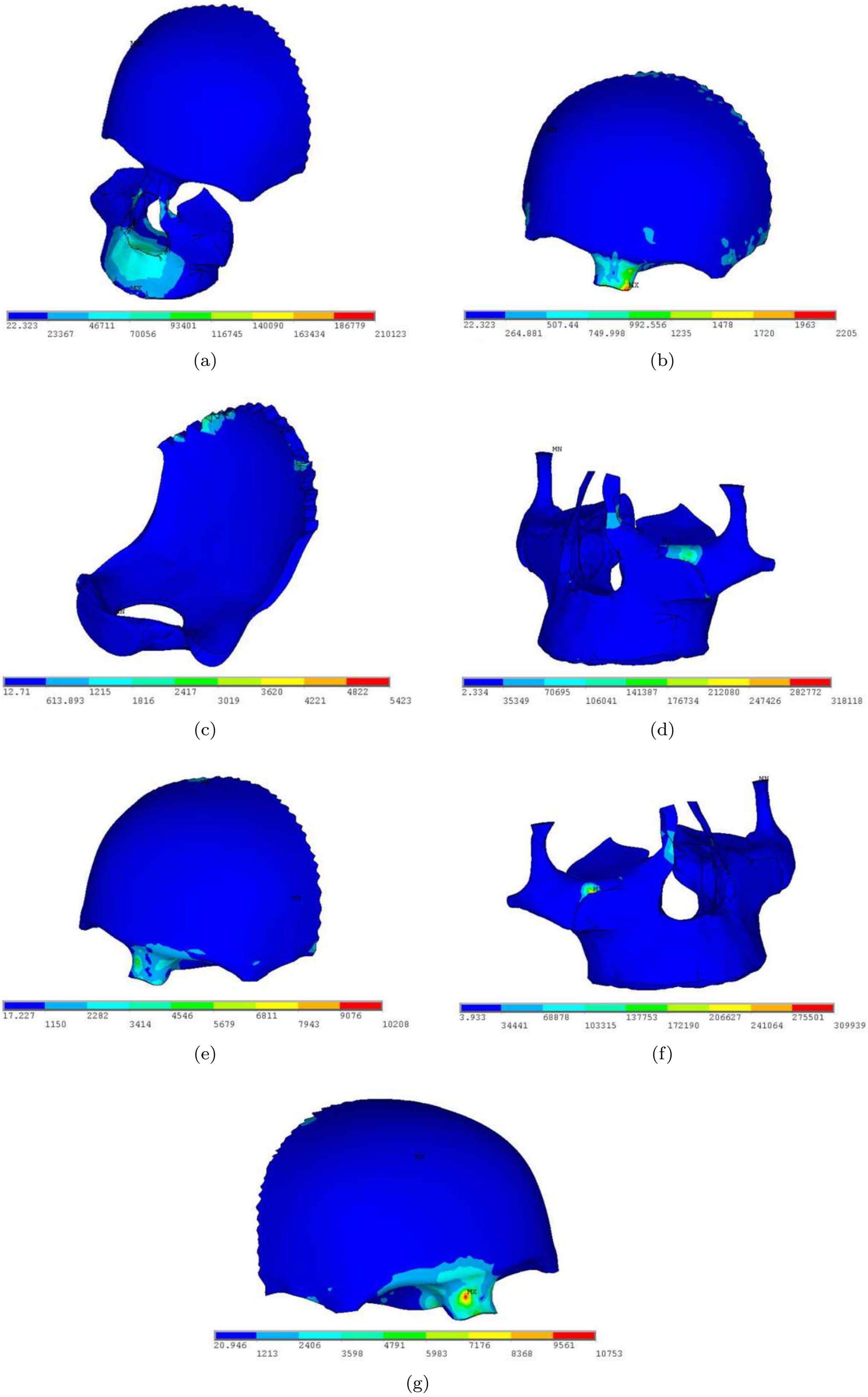
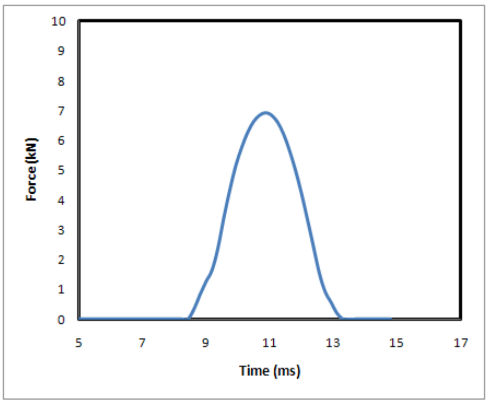
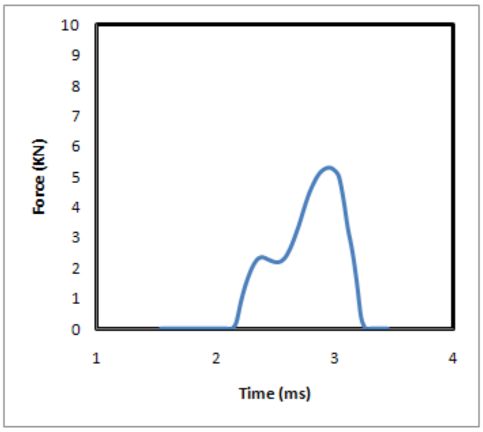


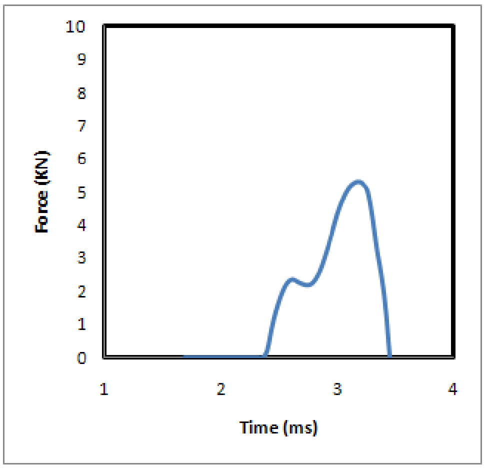
Fig.2:: Stress distribution (kPa) on the different components of skull during (a)(c) maxilla bone (d)(e) left zygomatic bone (f)(g) right zygomatic bone impact.



(a)



(b)



(c)

Fig.3:: Impact force of the impacted (a) maxilla bone (b) left zygomatic bone and (c) right zygomatic bone.

3. RESULTS

Figure 2 shows the stress distribution on the different components of skull during maxilla bone, left and right zygomatic bones impact. While an object was striking on maxilla bone, there were stress distributions on many parts of the skull i.e. maxilla, nasal, frontal, parietal and occipital bones as shown in figures 2(a)-2(c). The stresses were 210.123, 2.205 and 5.423 MPa on maxilla and nasal, frontal and parietal; and occipital bones, respectively. In addition, the stress distributions were occurred on nasal, zygomatic and frontal bones when the object was blowing left and right zygomatic bones as shown in figures 2(d)-2(g). In this case, the stresses were 318.118 and 10.208 MPa on the left zygomatic and frontal bones; and 309.939 and 10.753 MPa on the right zygomatic and frontal bones, respectively. Obviously, the maximum stress appeared on position of the component where was stroked directly. Moreover, if the maximum stress on each of the impacted position were compared at the same impact time, zygomatic bones have maximum stress.

Figure 3 shows the impact force of the impacted maxilla bone, left and right zygomatic bones. Peak of impact force on maxilla, left zygomatic and right zygomatic bones were 6.939, 5.28 and 5.29 kN, respectively. These peaks of impact force have arisen approximately within 4, 1 and 1 seconds, respectively after an object contact to the bones.

4. DISCUSSION AND CONCLUSIONS

In case of the stress distribution evaluation on the different components after maxilla bone, left and right zygomatic bones impact with the same conditions, maximum stress has occurred on zygomatic bones. This may be caused by a smaller surface area of zygomatic bone than one of maxilla bone. In this study, the stress on the left zygomatic bone was a bit different from right zygomatic at the same impact time, possibly because of setting error in initial position of foreign object.

From the FE model simulation, the impacted maxilla bone has a maximum impact force possible causing by the difference of surface contact on object. Maxilla bone may be contacted to object more complete than zygomatic bone which only contacted to the edge of the object. However, when the durations of all peaks appeared were considered, peak of impacted zygomatic bones has been taken place faster than that of another.

From the results, maybe means that zygomatic bone is a weak point on the skull. Therefore, the severity of injury of zygomatic bone is possibly most happened when struck a foreign object that had the same characteristics and behaviors on maxilla bone and zygomatic bones.

This FE model of the human skull impact built can be useful for further studying the mechanic of head impact, analyzing causing a knockout and investigating a maximum weak position on a human face. However, the boundary conditions must be changed for improvement more realistic FE model of human head with others tissues such as scalp, brain, cerebro spinal fluid etc. Furthermore, impact the foreign object to the others positions on the skull will be regarded in the future.

5. ACKNOWLEDGEMENT

This study is contributed by Department of Industrial Physics and Medical Instrumentation, King Mongkuts University of Technology North Bangkok. The simulation technique of ANSYS LS-DYNA is supported by Joompondej Bamrungwongtaree, Head Stack Assembly (HSA) process development, Seagate Technology (Thailand) Co., Ltd.

References

[1] J. Kennard, "Available: http://menshealth.about.com/od /fitness/a/boxing.htm .", *About.com: Men's health. [Online].*

[2] W. J. Hochswender, "The mechanics of a knockout punch.", *Popular Mechanics*, pp. 74–77, Jun., 1988.

[3] S. W. Gong, H. P. Lee, C. Lu, "An approach for the estimation of contact force on a human head induced by a foreign-object impact.", *IEEE Transactions on Biomedical Engineering*, vol. 54, no. 5, pp. 956–958, May. 2007.

[4] S. W. Gong, H. P. Lee, C. Lu, "Dynamic response of a human head to a foreign object impact.", *IEEE Transactions on Biomedical Engineering.*, vol. 55, no. 3, pp. 1226–1229, Mar. 2008.

[5] Y.H. Chu, M. Bottlang, "Finite element analysis of traumatic brain injury."

[6] S. Kleiven, "Finite Element Modeling of the Human Head.", *Doctoral's thesis, Royal Institute of Technology.*, Stockholm, Sweden, May. 2002.

[7] G. Krabbel, R. Müller, "Development of a Finite Element Model of the Head Using the Visible Human Data."

[8] G. Belingardi, G. Chiandussi, I. Gaviglio, "Development and validation of a new finite element model of human head."

[9] R. R. Mulabagula, "Finite Element Application to Head Injury Modeling.", *M. Sc. thesis, Ohio University*, United State, Jun. 1994.

[10] X. Yue, L. Wang, S. Sun, L. Tong, "Viscoelastic finite-element analysis of human skull-dura mater system as intracranial pressure changing.", *African Journal of Biotechnology*, vol. 7 (6), pp. 689-695, Mar. 2008.

[11] P. Pongpanitanont, W. Sittiprapaporn, W. Charoensuk, "Pattern recognition in brain FMRI for agnosia.", *International Journal of Applied Biomedical Engineering (IJABME)*, vol.3, no.1, pp. 39-44, 2010.

Pattaraweerin Woraratsoontorn received the B.Sc.(Industrial Physics and Medical Instrumentation) from King Mongkut's University of Technology North Bangkok, Thailand, in 1997 and M.Sc.(Radiological Science) from Mahidol University, Thailand, in 2000. Now, she is studying in Ph.D. at Department of Electrical Engineering, King Mongkut's Institute of Technology Ladkrabang. Her research interests are in the area of medical instrumentation and finite element modeling. Since June 2000, she has been the lecturer at the Faculty of Applied Science, King Mongkut's University of Technology North Bangkok, Thailand.

Pitikhate Sooraksa received the B.Ed.(Physics), M.Sc.(Physics) from Srinakharinwirot University, Thailand, M.Eng(Electrical Engineering) from the George Washington University, USA and Ph.D.(Electrical Engineering) from University of Houston, USA. He has been the lecturer at the Faculty of Engineering, King Mongkut's Institute of Technology Ladkrabang. His research interests are in the area of IT-automation & robotics and computer-communication-control for HDDs.