

Impact Simulation for Cranioplasty using Non-Linear Finite Element Model

[H.K.Park](#), J.H.Park, H.D.Han & J.H.Chung

*Biomedical Education Center, Department of Biomedical Engineering, School of Medicine
Kyung Hee University, Seoul, Korea*

INTRODUCTION: Understanding the mechanical reasoning of injuries can help to determine injury tolerance and develop how to protect from injury and quantify its relationship with the degree of injury. The brain movement inside the skull has a complex three-dimensional dynamic boundary condition, which means a dynamic head injury induces transient stress distribution in three dimensions. The internal biomechanical response due to brain injury is difficult to analyze by the present outcomes from experimental animal models because mechanical transducers cannot measure the transient stress changes internally due to the current limitations of the technique. The changes after cranioplasty indicated in selected cases that cranioplasty can improve cerebral circulation and protect from neuronal deterioration [1, 2] with a good cosmetic result.

In previous studies in our laboratory, we evaluated a cranioplasty that used four different implant materials to reconstruct the normal contours of the skull. The result presented the particular reference to the safety and suitability of cranioplasty and the behavior of the implanted material over secondary impact injury. In order to improve the limitation of a previous model, we designed a redefined model close to the human skull and analyzed dynamic behavior with new implant materials. In this study, large cranioplasty that used triple layer implant materials was simulated during the injury by weight drop impact.

METHODS: The CT image (1 mm slice thickness) of the male patient skull was transferred to the line data in order to create an accurate surface patch. The data transferred from the CT scan was filtered by imaging data protocol to verify the integrity of the data set for slice and volumetric viewing.

In order for any computer model to provide useful information, an accurate material with constitutive laws and material properties of the patch are essential. For the finite element analysis of the computational model of the patch, four-node quadrilateral isotropic shell element is mainly used.

RESULTS: The overall model consists of 4,740 nodes, 2,168 solid elements, and 4,736 shell elements. The average size of the element was 2 mm², and the thickness of the patch was 7 mm. Four peripheral areas were fixed by using titanium mini-plates and screws to secure the patch to the skull.

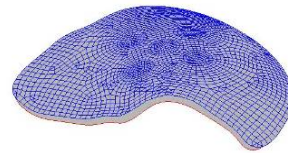


Fig.1: oblique view of inner, outer and middle layer

Table1. Comparison of the simulation results by composite implant

	Force (kN)	Displacement (mm)	Equivalent Stress (MPa)	
			Peripheral	Center
Spongy/cortical bone	2.8	2.96	29.2	47.1
HDPE/HA	5.7	1.44	64.7	172
HDPE/PMMA	1.8	4.48	25.2	100.

DISCUSSION & CONCLUSIONS: As a result of dynamic analysis, the composite structure of the implantable material was responding differently by the injury. However, when severe circumstances are considered, such as hitting by baseball or bat, the impact can be occurred at remarkably short time duration. When the impact energy is extremely high, the exact result is still unknown. It may be assumed that severe response can be reduced by change of location of the fixed point or of material properties without consideration of change of the geometric shape.

REFERENCES: ¹ Segal, D.H., et.al(1994) Neurological recovery after craniectomy, *Neurosurgery*, Vol. 34, No.4, pp. 729-731.
²Dujovny, M., et.al(1997) Post-cranioplasty cerebrospinal fluid hydrodynamic changes: Magnetic resonance imaging quantitative analysis, *Neurological Research*, Vol. 19, pp. 311-316

ACKNOWLEDGEMENTS: supported by MOCIE of Korea