Experimental vs. numerical investigations of stress induced by a compressive concentrated force applied to the orbital bone

J. AWREJCEWICZ[1], A. DĄBROWSKA-WOSIAK[1], J. MROZOWSKI[1], M. JARONIEK[2]

1. Technical University of Lodz, Department of Automation and Biomechanics, Lodz, Poland
2. Technical University of Lodz, Department of Strength of Materials and Structures. Lodz, Poland

Key words: *biomechanics, orbit, compression test*

1. Introduction

At present numerical simulations are essential tools in scientific research and engineering projects. One of the used methods among them is the finite element method used for example tests of the skull, eye socket and the reconstruction facial skeleton. Calculations performed by its help are always based on certain assumptions and simplifications concerning f. e.: material properties, geometry, type of loads and type of their application. Because of this advisable is a periodic revision of obtain numerical results through experiments conducted on the physical model reflects as possible faithfully the actual object of research. It is usually time-consuming and expensive procedure but giving the researcher confidence in formulating definitive conclusions.

2. Methods and materials

Modeling by finite element modeling is one of the most accurate used in the strength analysis. In this work, it is used, in the Ansys Workbench program, to model the eye socket for different type of material: bone (Table 1) and resin material used for performance, in the 3D PolyJet technology, of the real eye socket model.

Table 1. Material properties for cortical bone from various articles.

Author	Cortical		
	E (MPa)	ρ(kg/m3)	ν
Al-Bsharat [1]	12,200	2,120	0.22
Willinger [2]	15,000	1,800	0.21
Furusu[3]	11,500	2,000	0.3

Analysis of finite elements modeling showed that for the tested orbital model the biggest stresses and thus the burdened places are located near the nose. This has it's reflect in the medical research results (CT photos – DI-COM- photo). Experimental research performed on t he resin model also confirms the location of the most susceptible to damage place.

Coreresponding autor: Anna Dąbrowska-Wasiak, Technical University of Lodz, Departmaent of Automation and Biomechanics, Lodz, Poland, E-mail: anna.dabrowska-wosiak@p.lodz.pl

Fig. 1. The orbital model with strain gauges and in testing machine Instron 4485.

3. Results

Due to the fact, that the numerical analysis was static, it was necessary to ensure similar conditions in the experiment. For the purpose, it was set the speed of top traverse move of strength machine by help which the model was loaded for a value 1. Higher speed of traverse could lead to that the load could no longer be static.

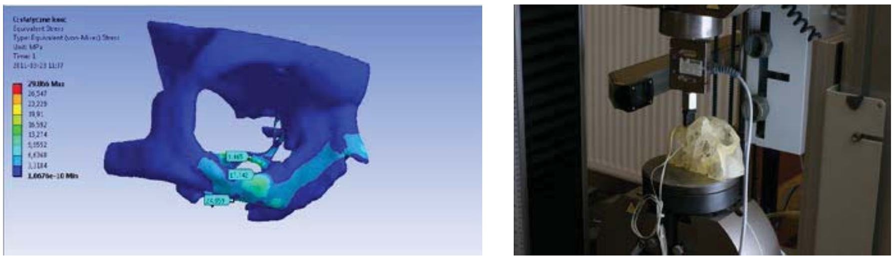

Fig. 2. The maximum stress in numerical model and confirmation in experiment.

4. Discussion

The experimental study was designed to check the correctness of obtained result from numerical analysis. The study presented two models of eye socket, numerical and made in the Rapid Prototyping technology allowing for the strength analysis in the eye socket bones. The research results of the strain gauge confirm the correctness of the stress distribution method occurring in the modeled orbit in the ANSYS program.

References

[1] AL-BSHARAT A, ET.AL., *Intercranial Pressure in the Human Head Due to Frontal Impact Based on a Finite Element Model*. Detroit, Michigan, Wayne State University.

[2] WILLINGER R., SUNG H., DIAW B., (1999) *Development and Validation of a Human Head Mechanical Model. Biomechanics,* vol. 327[IIb], 125-131. Paris, France, Academiedes Sciences/Elsevier.

[3] FURUSU, K., ET.AL. (2001) *Fundamental Study of Side Impact Analysis using the Finite Element Model of the Human Thorax*. *Japan Society of Automobile Engineers Review* **vol. 22**, 195-199. Elsevier Science BV.