Modeling of human calvarium under static loading

G. Wróblewski

Warsaw University of Technology, Department of Engineering Production, Narbutta, 85, 02-524 Warsaw

1. Introduction

There are many papers on head modeling in literature [1]. Generally, the topic of most of them is designing and modeling devices for head (especially brain) protection, e.g. helmets. On the other side, most papers are about modeling of whole part of top body (a torso and a head). The main aim of these researches is simulation of the backbone behavior under external load applied to the head. The cranium or calvarium bones simulation is very rare is presented in scientific literature [2].

The main aim of this paper is to present results of simulations of cranium injures. In fact, real injures observed in cranium bones are caused by time-depend (dynamic) loads (e.g. communication accidents, shots). But in this work the initial results are presented. In the assumed models the static (or quasi-static) loads were applied on different areas the frontal part of the cranium in order to observe stress state. Thus, it would be possible to find place where bone tissue may crack.

2. Methods

To perform numerical calculation the geometrical model of the human cranium should be generated. This model was created on the basis of the data acquired during computer tomography (CT). A patient KD (43 years, without bone pathology) was examined in Medical University in Warsaw. As a result, 85 images (slices) were obtained (3 mm distance between slices).

Then, the obtained set of data was transferred into MIMICS system and the virtual geometrical model was created. All the surfaces of the virtual model (outer and inside) were described by triangular surfaces (STL format). A virtual model was optimized, i.e. the number of triangles was reduced and their shape were modified) and final geometrical model was generated.

The geometrical model was transferred into ABAQUS system in order to perform some numerical simulations. In these investigations seven different external loadings were assumed. These loadings were applied onto different regions of the model and three different values of loadings (for each case of loadings) were assumed. The direction of applied forces were changed, too. In this paper, the results of loading applied to wide part of human face are presented, as is shown on Fig. 1. Calculations were performed for three different values of external loadings (1500, 4500 and 9500N), the direction of the load – perpendicular to coronal plane. (Fig. 1b). Values of external loads were assumed from literature [4].

Fig. 1 A geometrical model generated in MIMICS (a), FEM model with external loading (b).

The geometrical model was transferred into ABAQUS system and finite element method mesh was generated. The mesh consists of 50274 tetrahedral elements. In a whole model cortical bone material was assumed, material data were taken from [5]. The simplification is caused by the fact that it was impossible to discern in the CT images a clear separation border between cortical bone and trabecular bone. In addition the orthotropic material model was assumed. Then the boundary conditions were assumed and simulations were performed.
3. Results
As a result of numerical simulations stress, strain as well as displacements distributions were obtained. Some stress distributions after external loading 9500N applied on model are shown on Fig. 2. The Fig. 2a shows stress distribution in front al view. It is shown that high stresses are concentrated in nasal bone (os nasale), malar bones (os zygomaticum) and in the region of anterior nasal aperture (apertura nasalis anterior). The observation stresses in the frontal view does not give a full picture of stress state in calculated model. The bottom view (on base of scull) shows more complicated stress state – as is shown on Fig. 2b. High stresses are observed in malar arches (from its internal side) and especially in the region of a vomer. The values of these stresses are higher than those observed in frontal view.

Fig. 1 Von Misses stress state generated in simulated model: a) front view, b) bottom view.

4. Discussion
In this work results of the first stage of researches are presented. Up to now some numerical calculations were done. Different places of applied forces were taken in this calculations, as well as its different values and directions. Conclusions after the first calculations:
- It is necessity to prepare a new geometrical model of human scull on the basis of computer tomography. Tomography examinations must be done in assumption that distance between slices must be smaller than 3mm – in some regions (e.g. nasal parts of scull) must be as small as possible. In effect more appropriate geometrical model of humen calvarium will be done.
- A new model may reflect different materials of the cranium. The division on cortical, spongy bone as well as tooth material may be assumed in this model. The next calculation may assume that the real forces applied on the model are time – dependant.

References