Int. J. Oral Maxillofac. Surg. 2012; 41: 66–73
doi:10.1016/j.ijom.2011.09.004, available online at http://www.sciencedirect.com

International Journal of Oral & Maxillofacial Surgery

Research Paper
Trauma

Transient finite element analysis of a traumatic fracture of the zygomatic bone caused by a head collision

A. Schaller[1], C. Voigt[2], H. Huempfner-Hierl[1], A. Hemprich[1], T. Hierl[1]

[1]Department of Oral and Maxillofacial Plastic Surgery, Biomechanical Laboratory, Leipzig University Hospital, Germany; [2]Department of Orthopedics, Biomechanical Laboratory, Leipzig University Hospital, Germany

A. Schaller, C. Voigt, H. Huempfner-Hierl, A. Hemprich, T. Hierl: Transient finite element analysis of a traumatic fracture of the zygomatic bone caused by a head collision. Int. J. Oral Maxillofac. Surg. 2012; 41: 66–73.

Abstract. Midfacial fractures rank first concerning maxillofacial traumatology. Collisions of two heads or head to object are the main causes for these fractures. An investigation based on a transient simulation using the finite element method was performed. A biomechanical head model was created and tested. A transient collision of two heads was simulated. The results were compared to a typical real patient case. This comparison revealed an identical fracture pattern, which can be interpreted as a clinical match of the simulation. The results of this study show the validity of biomechanical investigations, which may serve as a method to better understand maxillofacial fracture patterns. These results will be used for the optimization of fracture therapy or trauma prevention in the future.

Keywords: Midfacial trauma; finite element analysis; transient simulation; material assignment; von Mises stress; yield criteria; orbital floor fracture; zygomatic fracture..

Accepted for publication 15 September 2011
Available online 12 October 2011

Owing to their high incidence, midfacial fractures rank first amongst all maxillofacial fractures, demonstrating their high clinical importance[29]. Collision of two heads or of a head and an object is a frequent cause of midfacial trauma. Typical fracture patterns occur in sports such as football, handball or in violence-related trauma[9,19]. Scientific investigations in these trauma situations can be performed in two ways: cadaver studies; and through simulation in a virtual environment using finite element analysis (FEA). FEA is a numeric method used to solve partial derivatives in order to simulate the behaviour of solid bodies according to certain boundary conditions using an approximation method, is becoming increasingly popular as it can be adopted in a wide variety of investigations if an appropriate model is available. Cadaver studies can only deal with single questions and are limited by the availability of adequate cadaver specimens and the necessary equipment. FEA has been used in investigations on biomechanical simulation of isolated orbital wall fractures using defined objects to hit the infraorbital rim[15,16]. The results of these studies are supported by the findings of several biomechanical experiments concerning orbital wall fractures[1,2,27,28] and are confirmed by a case study[6]. According to the high incidence of head collisions[29] and the importance of the orbitozygomatic complex in complex midfacial and panfacial fractures[20], trauma studies on these fracture patterns are an interesting object of investigation. In contrast to their clinical importance, there have been no finite element studies regarding the collision of two heads.

This investigation aimed to design a more detailed and, in biomechanical respects, more appropriate model, compare it to earlier approaches and test it in the special trauma setting of head collision. The results would then be reviewed in a clinical context to decide whether the fracture patterns seemed realistic. If so, the next steps would be simulating further trauma modalities, investigating fracture patterns with respect to the probable impact, and considering different osteosynthesis strategies. The results could be used for trauma prevention, for example in the design of appropriate helmets as has been done in automobile safety studies or military investigations.

In order to gain further knowledge on the consequences following this particular trauma FEA can be used to calculate local stresses. These calculations could allow a better understanding of fracture genesis and permit statements on the requirements of osteosynthesis systems and reconstruction techniques. FEA allows clinicians to predict the biomechanical and anatomic consequences of any injury of the head more easily and cost-effectively than experimental methods and makes it possible to estimate a specific impact load according to an existing fracture pattern if an adequate model is available. Therefore the current finite element study with a collision of two heads was launched to investigate a computer simulation of a zygomatic trauma with respect to the occurring fracture pattern. These patterns are compared to the findings of a clinical case.

In order to simulate this trauma a transient analysis was conducted (i.e. simulating the whole trauma with smooth transition of all steps), which allows simulating the entire incident without additional calculation to determine the impact force and impact time[16,27]. This type of analysis is more suitable for representing the realistic behaviour of the skull during trauma, as the effect of the viscoelastic response of bone due to a time dependent load has already been investigated for the upper limb in an experimental study[22]. To assess the approach of this study, the biomechanical model was compared in two simulations with data taken from earlier investigations, which were concerned with geometrically defined impactors and test-bed settings[1,2,15]. A specially defined load scenario was performed in a transient dynamic study of a head-to-head collision to understand the appearance of a typical maxillofacial fracture pattern; thus three simulations were performed.

Materials and methods

FE modelling

The central idea of FEA is to describe the mechanical behaviour of any given object by cutting it into small areas, the reactions of which can be individually computed. The individual reactions of these cells are later combined to calculate the reactions of the whole object. In order to do this some material properties are required (Young's modulus and Poisson's ratio), which are assigned to the individual cells. The great advantage is that FEA can be applied to almost any form or material, ranging from automobile design to surgical questions. In the medical fields, the main problem is to describe the detail required and decide which individual material parameters should be assigned. It is obvious that simple models like attributing one parameter to a complex anatomic structure such as the skull, as done in earlier work, is a very simplified approach and could lead to inappropriate results.

The authors describe their way of establishing a better model with respect to anatomic detail and biomechanical behaviour. In order to build the finite element model for the study a patient's CT dataset (Siemens Volume Zoom Plus, 1 mm contiguous slicing) of a head without any pathological anomalies or any traumatic injury was used and segmented in the imaging software Vworks™ 4.0*Surgery* (Cybermed Co., Seoul, Korea). During segmentation only bony structures are kept and soft tissue is discharged. The segmented bone was manually adjusted and holes in thin cortical bone areas were filled according to anatomical respects[18]. The result was a very detailed 3D surface mesh. This optimized file was exported into VRML format, which is an exchange file format for 3D data and then imported into the finite element meshing module of the finite element software package ANSYS ICEM CFD 12.0.1 (ANSYS Inc., Canonsburg, PA, USA) (Fig. 1A). In this module a finite element volume mesh with tetrahedral shaped 10-node elements was created according to the segmented 3D surface mesh of the bone with 736,934 volume elements (Fig. 1B). As the main focus lay on the midface, the mandible was omitted. For the two comparison studies, finite element models of impactors were built: one with a volume of 0.0481 cm^3 according to Waterhouse et al.[28] and Nagasao et al.[15]; and another with a volume of 0.0288 cm^3 based on Ahmad et al.[1,2] (Fig. 2A and B). In the head collision study, a simplified finite element skull model with a less dense volume mesh of 86,187 elements was created in ICEM. This model was used as an impactor to simulate a realistic collision of two heads (Fig. 2C). All finite element models created in ICEM were transferred to ANSYS Classic® v12.0.1 (ANSYS Inc.) (ANSYS) in order to carry out a transient nonlinear solution.

Material properties

The finite element model of the investigated skull of the collision study was assigned material parameters based on

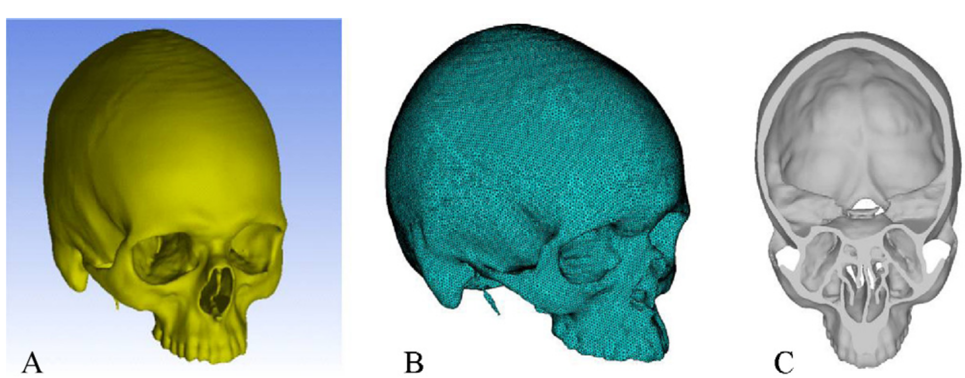

Fig. 1. (A) Representation of the surface triangulation of the segmented and optimized skull in the meshing module ICEM CFD 12.0.1 of the FEM software package ANSYS. (B) Representation of the final finite element volume mesh with 736,934 elements. (C) Cross section of the finite element skull model showing the details of the midface without the outlines of the finite elements.