10.5005/jp-journals-10015-1294

REVIEW ARTICLE

3D Finite Element Technology and Its use in Craniofacial Injuries

[1]ES Shobha, [2]HP Raghuveer, [3]Suresh Nagesh, [4]Dilip Kumar Rayapati, [5]NT Prashanth, [6]Vinod Rangan

ABSTRACT

Quite a bit of interest and evolution has happened in the field of research in the recent times toward analyzing and understanding of trauma and injuries especially that on the human skull and related organs. With the development of faster and latest computers, high fidelity human models have been created to understand the biomechanics of these structures to trauma. In the present article a broad methodology of the fundamentals of model creation and analysis and fracture mechanics has been presented. In particular this paper highlights the methodology of 3D simulation of craniofacial region and an insight to the role of FEA (Finite Element Analysis) in craniofacial trauma. The paper also explains the various steps in computational simulation of the craniofacial skeleton.

Keywords: Craniofacial fractures, Finite element analysis, Finite element technology.

How to cite this article: Shobha ES, Raghuveer HP, Nagesh S, Rayapati DK, Prashanth NT, Rangan V. 3D Finite Element Technology and Its use in Craniofacial Injuries. World J Dent 2014;5(4):223-228.

Source of support: Nil

Conflict of interest: None

INTRODUCTION

The human skull is a very complex structure with different skull bones aligned to have a specific structure to protect the sense organs, cranial contents, dental and oral structures, etc. Road traffic accidents, falls, fights, etc. makes the human skull susceptible to injuries which lead to concentration of stresses in particular areas, especially sutures. The prominence of skull is due to skull bones like mandible, zygoma, nasal bones. These are more susceptible to fractures when compared to the rest of facial skeleton. The complexity of the craniofacial skeleton creates a need to study the craniofacial fractures for better understanding of injury mechanism for planning, prevention and for designing and inventing of protection devices as well as surgery planning in maxillofacial surgery.

Finite element modeling (FEM) is a numerical method of creation of an anatomically realistic generic computational model. The modeling allows reproduction of a particular loading on a structure with varying parameters and the stresses can be calculated. Computational realistic models of the human skull can be created and the structural mechanics can be studied under different impact forces. Finite Element (FE) models offer the possibility to analyze the mechanism of injuries as well as the effect of the impact forces on the mechanism of these injuries.

HISTORY

Finite element modeling was initially developed as a (Courant 1943) mathematical evaluation procedure and its wide spread use was unseen until the advent of computers. The use of FEM has spread into the field of aerospace (Levy 1953) as well as increasingly in the field of engineering and biomechanics (Huiskes and Chao 1983). It has varied usage in the field of biomechanics as well as in the field of life sciences.[1]

DISCUSSION

Finite element method (FEM) or more generally finite element analysis (FEA) is the process of numerical solutions to field problems in engineering and physics.[1] Typically the process involves solving differential equations using numerical solution techniques. In general, the process involves discretizing a given system (could be an engineering component, subcomponent or in general any structure, such as human bones, skeletons, skull, etc.) into a finite number of elements. Then one would apply the relevant material properties and boundary conditions including loading, chose the right type of solution based on the loading and the material types. These typically involve static or dynamic, linear or nonlinear analysis. Thus, in general, FEA can be applied to biomechanics applications.

Finite element modeling in summary is the process of creating a representative numerical model of the physical model followed by subjecting the model to

[1,4]Professor, [2]Professor and Head, [3]Chair Professor
[5]Associate Professor, [6]Assistant Professor

[1,2,4-6]Department of Oral and Maxillofacial Surgery, Dayananda Sagar College of Dental Sciences, Bengaluru, Karnataka, India

[3]Department of Computational Mechanics, CORI, PES University, Bengaluru, Karnataka, India

Corresponding Author: ES Shobha, Professor, Department of Oral and Maxillofacial Surgery, Dayananda Sagar College of Dental Sciences, Bengaluru-560078, Karnataka, India, e-mail: shobha_es@yahoo.com

Fig. 1: Model of a skull impacting a rigid surface

appropriate load and boundary conditions and then to validate and interpret the results.[1] Particularly, in the biomechanics domain, this would involve creation of the finite element model from a 3D CT scan in DICOM format of the craniofacial skeleton. This is one of the most time consuming phase and involves the integration of data regarding the model geometry, finite element meshing, element design, incorporating the material properties, applying loads and boundary conditions (Fig. 1). [1] The second phase or solution phase consists of calculating the stresses, strains and energy fields for the chosen problem as well as collecting data available in literature. This followed by the last phase or post processing phase which involves the interpretation of results and validation.

Finite element methodology can be used to know the biomechanical behavior of human skull. Computational mechanics has made it possible to create a finite element model of the human skull and the feasibility to virtually simulate the impact forces and to evaluate the stress and energy fields on the human skull by mathematical calculations and analysis.

Finite element models helps in understanding the injury mechanism, the biomechanical behavior of human skull and the pathological dysfunction which in turn helps in prevention and protection against injury, helps in devising protection systems for road users like helmets as well as surgical planning of maxillofacial cases which requires a knowledge of force application and bone displacement.[2]

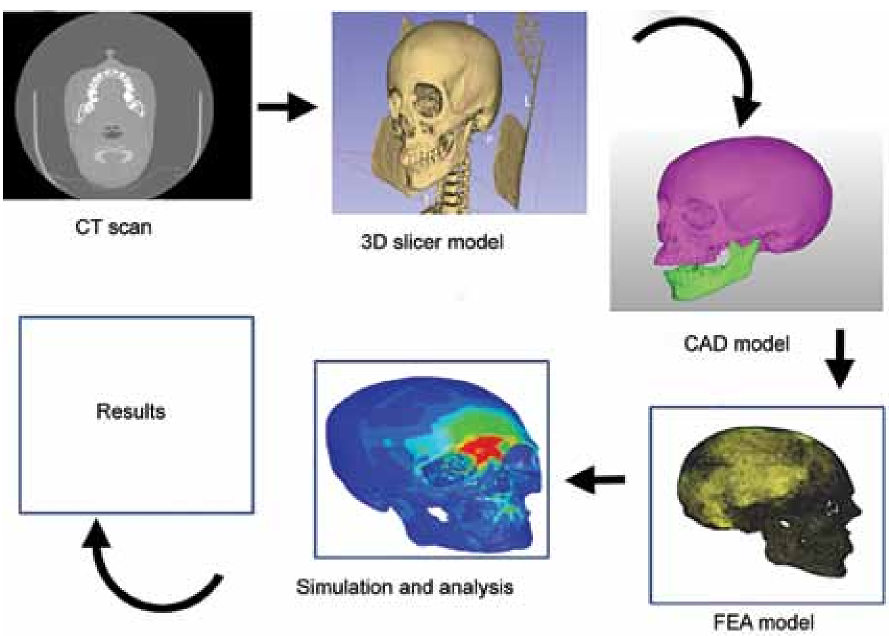
Muscle insertions have been simulated and stress concentrations in the maxillofacial regions has been assessed with or without occlusal load and it was found that the stress progressed through the maxilla following nasal, zygomatic and pterygoid route. Stress seemed to be concentrated in areas of architectural reinforcement.[3]

Forces are applied to the mandible and maxilla through the teeth, muscles of mastication and through the reaction forces at the TM joint region. Knowledge of distribution of internal forces and the deformation occurring as a consequence of these internal forces is required for the success of the treatment planned.[4,5]

Nahum et al[6] has conducted a number of cadaveric studies. He investigated two series of head impact experiments using a single impact experiment and different energy level impacts for a single specimen. FE models can be validated against the cadaveric impacts performed by Nahum.

The procedure for creation of realistic geometric model involves the inclusion of the geometry of the structure into the computer. This process involves the use of automated or manual approach. CT scans are preferred

Flow Chart 1: Process flow chart for the study



over laser scans as the automation and high resolution of CT'S make it more attractive.[1] CT scan images are then converted into CAD models which in turn are changed to FE models (Flow Chart 1).

Once model creation is completed meshing or subdividing the model into large number of geometrically simple domains called finite elements is done which helps in making the calculations of stress and strains easier. The mesh quality is improved by the edge contraction and smoothing methods. The mesh quality without any hanging nodes is created using volumetric imaging data.[7]

Once the model meshing is completed, the material properties are incorporated into the model. Studies have been conducted to know the variations of material properties of maxilla and mandible. It was found that the elastic modulus and hardness were higher in the posterior than the anterior region and the variation in the trabecular structure of bone coincides with variation in the mechanical properties.[8]

The elastic properties of the maxilla were more variable than mandible and areas of greatest consistency were the alveolar bone and frontomaxillary area. The buccal alveolar bone was thicker, less dense and stiffer and the palatal vertical bone is more or less similar to alveolar cortical bone. Incorporation of these properties into the FE models improves the accuracy and reliability of the models.[9,10]

Mechanical properties, such as young's modulus, ultimate strength, yield strength, modulus of resilience, modulus of toughness and density are the key factors required for a better understanding and proper evaluation or analysis of subjects (Juvinall 1983; Wang and Dixon 1997 a, b, Wang et al 2000).[11]

Boundary conditions are important and necessary as the boundary constraints anchor the model and enable a unique elasticity to be obtained preventing the entire model to be uniformly displaced in the frame of reference as a rigid body without affecting the elastic response of the structure (Fig. 2). Finite element computation would have been impossible without these boundary constraints.[1] The completed model is then used to obtain the nodal displacement occurring and the resulting stresses and strains are calculated.[1]

Validation of the FE model in both 2D and 3D approach[12] is important to assess the accuracy and precision of the model results. Models can be validated using experimental data from cadavers.[13] Results obtained from the FE models were also compared with the real patient case for validation.[14] Imprecise assumptions of elastic properties as well as inappropriate loads and constraints applied on the FE model may cause an unrealistic deformation of the model,[15] thus limiting the utility of FEA for analysis of the biomechanical behavior of the FE model and also will affect the validity of the model.

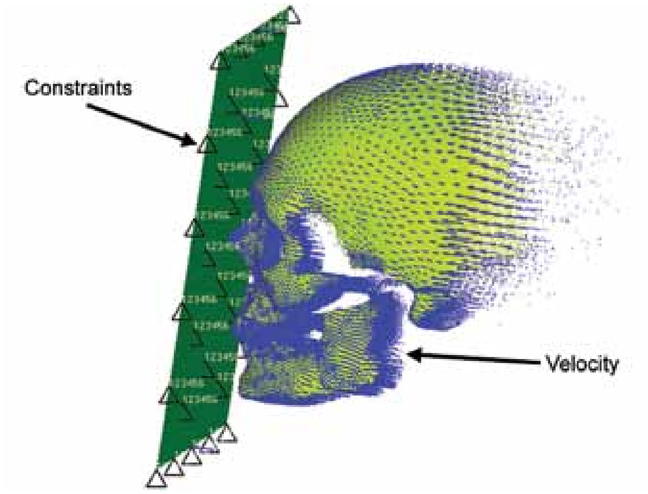


Fig. 2: Loading and boundary conditions

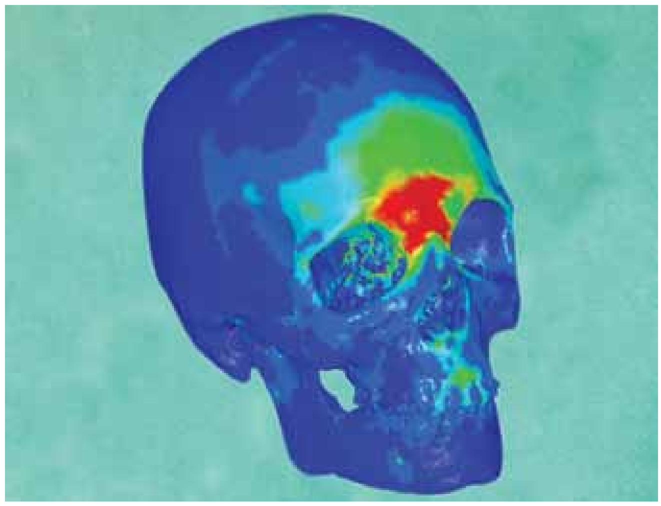


Fig. 3: Von mises stress contour on the skull after impact at t = 0.015 mS

Vollmer et al[16] compared mandibular deformation under mechanical loads to the results derived from finite element analysis and concluded that the procedure for generating an FE model is valid and precise and is also a non-invasive method to predict the complexity of different parameters in assessing the biomechanical behavior of human mandibles. Hence, knowledge and variation of the elastic properties is required for creating a valid and accurate finite element model as well as to evaluate the biomechanics of the skull.[17]

As mentioned earlier the elastic properties varies between different regions in the craniofacial skeleton variation is more in maxilla than mandible. Elastic modulus is higher in the posterior than anterior region. Studies have shown that the thickness of the cortical plates also varies with the age of the subject. Ten to nineteen-year-old subjects had thicker cortical plates with peak thickness seen in 40 to 49 years old which decreases in thickness after this period.[18] Hence, incorporating the elastic properties as well as the appropriate dimensions of the anatomical region into the model helps in obtaining a realistic, significantly precise and accurate as well as a validated model for the assessment of the biomechanical behavior required for the study (Fig. 3).

Finite element analysis has its varied application in the field of medicine. FEA helps in simulating the human birth process, the need for an elective cesarean section can be predicted when the obstetrician is in a dilemma due to cephalo-pelvic disproportion.[19]

In pediatrics mechanical cranial birth injuries can be assessed by creation of realistic models of fetal head molding occurring due to prolongation of labor wherein the forceful contractions can cause excessive displacement of skull bones which may cause bony lesions, dural membrane injury, intracranial hypertension congestion of the galenic venous system and direct injury to major intracranial vessels.[19]

In the field of maxillofacial surgery FEA can be used to predict soft tissue deformations in maxillofacial surgery planning[20,21] as well as predicting weak areas in the craniofacial skeleton.[22,23] Further FE models can be used to simulate and analyze the biomechanical behavior of bone in two standard trauma situation and to study the maximum stress concentration in the different regions of the maxillofacial skeleton.[24]

Focusing on the treatment planning in maxillofacial trauma patients, FEA can be used to formulate biomechanical justification regarding the position of different plates at the fracture site.[25,26] Using this technology we can now determine the appropriate length of the plate, the number of plates to be used in a particular fracture situation and the number of screws to be used as well.[27] On analyzing the data parameters such as the stiffness of the plates[28] and stability of the fracture fragments, desirable angulations during plate fixation required can also be determined.[29]

Researchers have tried various biomaterials for implants in maxillofacial surgery and using FEM have compared single and double miniplates for fracture stabilization. The stress distribution and the stress shielding effect around the miniplates made of different biomaterials has been studied.[30-36]

3D reconstruction mandibular models have been created with muscle forces; bite forces as well as joint forces have been simulated. The factors determining the loads across a fracture and the magnitude and direction of forces on osteosynthesis and the optimal localization and direction of osteosynthesis has been studied. It was concluded that complex loads act on the fracture and it was the surgeon's choice to decide on the ideal plate, optimal positioning and number of bone plates.[37]

Finite element analysis has been used to study the stability of the osteotomized segments in orthognathic surgery as well. Different fixation methods following Bilateral sagittal split osteotomies (BSSO) have been tried. Miniplates with monocortical screws have been compared to lag screws[38,39] and it was found that in BSSO the triangular lag screw or double miniplates provided better stability than the linear lag screw and single oblique miniplates. Another study has compared four resorbable osteosynthesis screws for stability of BSSO.

Further studies have used FEA in studying the behavior of human head with or without helmet during frontal impact.[40] Gunshot injuries were simulated and analyzed to determine the mechanism of injury and degree of damage to the mandible in humans.[41] Lag screws with or without washer were compared with anchor screws and the load distribution below them have been studied by FEA.[42,43]

Apart from adult trauma FEA has been used in pediatric trauma to simulate the pediatric skull and to analyze mechanisms leading to neurologic injuries[44] further relationship of impacted mandibular 3rd molar and angle fractures has been studied.[45]

CONCLUSION

Finite element technology has been and is an emerging technology to understand the biomechanics of structural changes that occur on application of force under a constraint. The use of such a technology in the field of medical sciences has proven its efficacy with the glorious advantages it offers to the surgeon with regard to the choice of the hardware to be applied in the fixation of any fracture in the craniofacial region. This technology gives an insight into the advantages of each fixation system over the other and helps chose the most appropriate hardware. Further, by simulating muscle forces and applying loads we are now able to determine the changes that happen to the bony fragments and also the stresses passing through the area of fixation. Although finite element technology is the need-of-the-hour with regard to achieving best results, clinical correlation is indeed required prior to delivery of the chosen treatment plan. FEA continues to help in designing newer preventive devices, focusing on better treatment options and developing better hardware for treating craniofacial injuries.

Figure 3 shows contour of representative von mises stress at a particular instant of impact. The figure shows the area of high stress concentration on the skull as it hits the rigid surface. It can be observed that the stress concentration is maximum at the surface of the skull contacting the rigid surface.

REFERENCES

1. Richmond BG, Wright BW, Grosse I, Dechow PC, Ross CF, Spencer MA, Strait DS. Finite element analysis in functional morphology. Anat Rec 2005;283A:259-274.
2. Autuori B, Bruyère-Garnier K, Morestin F, Brunet M, Verriest JP. Finite element modeling of the head skeleton with a new local quantitative assessment approach. IEEE Trans Biomed Eng 2006 Jul;53(7):1225-1232.
3. Alexandridis C, Caputo AA, Thanos CE. Distribution of stresses in the human skull: J Oral Rehabil 1985 Nov;12(6): 499-507.
4. The elastic properties of a human mandible: Ashman RB, Van Buskirk WC: See comment in PubMed Commons below. Adv Dent Res 1987 Oct;1(1):64-67.
5. van Eijden TM. Biomechanics of the mandible. Crit Rev Oral Biol Med 2000;11(1):123-136.
6. Nahum A, Smith R, Ward C. Intracranial pressure dynamics during head impact. SAE Technical paper 770922;1977.
7. Zhang Y, Bajaj C, Sohn BS. 3D finite element meshing from imaging data. Comput Methods Appl Mech Eng 2005 Nov 15;194(48-49):5083-5106.
8. van Eijden TM, van der Helm PN, van Ruijven LJ, Mulder L. Structural and mechanical properties of mandibular condylar bone: J Dent Res 2006 Jan;85(1):33-37.
9. Peterson J, Wang Q, Dechow PC. Material properties of the dentate maxilla. Anat Rec A Discov Mol Cell Evol Biol 2006 Sep;288(9):962-972.
10. Seong WJ, Kim UK, Swift JQ, Heo YC, Hodges JS, Ko. Elastic properties and apparent density of human edentulous maxilla and mandible: CC Int J Oral Maxillofac Surg 2009 Oct;38(10):1088-1093.
11. EBW Giesen, M Ding M Dalstra. Mechanical properties of cancellous bone in the human mandibular condyle are anisotropic: TMG J van Eijden. J Biomechanics 2001 June; 34(6):799-803.
12. Evans SP, Parr WC, Clausen PD, Jones A, Wroe, J Biomech. See comment in PubMed Commons below Finite element analysis of a micromechanical model of bone and a new 3D approach to validation: 2012 Oct 11;45(15):2702-2705.
13. Roth S, Raul JS, Willinger R. Finite element modelling of paediatric head impact: Global validation against experimental data: Computer Methods and Programs in Biomed 2010 July;99(1):25-33.
14. Schaller A, Voigt C, Huempfner-Hierl H, Hemprich A, Hierl T. Transient finite element analysis of a traumatic fracture of the zygomatic bone caused by a head collision. Int J Oral Maxillofac Surg 2012 Jan;41(1):66-73.
15. Strait DS, Wang Q, Dechow PC, Ross CF, Richmond BG, Spencer MA, Patel BA. Modeling elastic properties in finite-element analysis: how much precision is needed to produce an accurate model? Anat Rec A Discov Mol Cell Evol Biol 2005 Apr;283(2):275-287.
16. Vollmer D, Meyer U, Joos U, Vègh A, Piffkò J. Experimental and finite element study of a human mandible. J Craniomaxillofac Surg 2000 April;28(2):91-96.
17. Wang Q, Dechow PC. Elastic properties of external cortical bone in the craniofacial skeleton of the rhesus monkey. Am J Phys Anthropol 2006 Nov;131(3):402-415.
18. Swasty D, Lee JS, Huang JC, Maki K, Gansky SA, Hatcher D, Miller AJ. Anthropometric analysis of the human mandibular cortical bone as assessed by cone-beam computed tomography. J Oral Maxillofac Surg 2009 Mar;67(3):491-500.
19. Lapeer RJ, Prager RW. Commons below fetal head moulding: finite element analysis of a fetal skull subjected to uterine pressures during the first stage of labour. J Biomech 2001 Sep;34(9):1125-1133.
20. Keeve E, Girod S, Pfeifle P, Girod B. Anatomy-based facial tissue modeling using the finite element method: IEEE visualization, 21-28;1.
21. Mollemans W, Schutyser F, Nadjmi N, Maes F, Suetens P. Predicting soft tissue deformations for a maxillofacial surgery planning system: From computational strategies to a complete clinical validation. Medical Image Analysis 2007 June;11(3):282-301.
22. MacNeil JA, Adachi JD, Goltzman D, Josse RG, Kovacs CS, Prior JC, Olszynski W, Davison KS, Kaiser SM. Predicting fracture using 2D finite element modelling: CaMos Research Group. Med Eng Phys 2012 May;34(4):478-484.
23. Philemon C, Lu Zi, Paul R, Erik T, Jiangyue Z, Narayan Y, Frank P. Development of a generalized linear skull fracture criterion. Proceedings of the 20th International Technical Conference on the Enhanced Safety of Vehicles (ESV).
24. Gallas Torreira M, Fernandez J. A three-dimensional computer model of the human mandible in two simulated standard trauma situations. Craniomaxillofac Surg 2004 Oct; 32(5):303-307.
25. Korkmaz HH. Evaluation of different miniplates in fixation of fractured human mandible with the finite element method: Oral Surg Oral Med Oral Pathol Oral Radiol Endodontol 2007 June;103(6):e1-e13.
26. Rangan V, Raghuveer HP, Rayapati DK, Shobha ES, Prashanth NT, Sharma R. The influence of stress distribution in four different fixation systems used in treatment of mandibular angle fractures—a three-dimensional finite element analysis. Oral Surg 2013;6:186-192.
27. Wagner A1, Krach W, Schicho K, Undt G, Ploder O, Ewers R. A three-dimensional finite-element analysis investigating the biomechanical behavior of the mandible and plate osteosynthesis in cases of fractures of the condylar process. Oral Surg Oral Med Oral Pathol Oral Radiol Endod 2002 Dec;94(6): 678-686.
28. Aquilina P, Chamoli U, Parr WC, Clausen PD, Wroe S. Finite element analysis of three patterns of internal fixation of fractures of the mandibular condyle. Br J Oral Maxillofac Surg 2013 Jun;51(4):326-331.
29. Zięwbowicz A, Marciniak J. The use of miniplates in mandibular fractures—biomechanical analysis. J Materials Processing Technol 2006 June 1;175(1-3):452-456.
30. Ji B, Wang C, Liu L, Long JTian WWang H. A biomechanical analysis of titanium miniplates used for treatment of mandibular symphyseal fractures with the finite element method. J Oral Surg Oral Med Oral Pathol Oral Radiol Endod 2010 Mar;109(3):e21-27.
31. Korkmaz HH. Evaluation of different miniplates in fixation of fractured human mandible with the finite element method. J Oral Surg Oral Med Oral Pathol Oral Radiol Endod 2007 Jun;103(6):e1-13.

32. Tams J, Otten B, van Loon JP, Bos RR. A computer study of fracture mobility and strain on biodegradable plates used for fixation of mandibular fractures. J Oral Maxillofac Surg 1999 Aug;57(8):973-981.
33. Feller KU, Schneider M, Hlawitschka M, Pfeifer G, Lauer G, Eckelt. Analysis of complications in fractures of the mandibular angle—a study with finite element computation and evaluation of data of 277 patients. J Craniomaxillofac Surg 2003 Oct;31(5):290-295.
34. Tate GS, Ellis E 3rd, Throckmorton G. Bite forces in patients treated for mandibular angle fractures: implications for fixation recommendations. J Oral Maxillofac Surg 1994 Jul; 52(7):734-736.
35. Tams J, Otten B, van Loon JP, Bos RR. A computer study of fracture mobility and strain on biodegradable plates used for fixation of mandibular fractures. J Oral Maxillofac Surg 1999 Aug;57(8):973-981.
36. Rozema E, Otten E, van Willigen JD, Bos RRM. Computer-aided optimization of choice and positioning boneplates and screws used for internal fixation of mandibular fractures. Int J Oral Maxillofac Surg 1992;21:373-377.
37. Erkmen E, Simşek B, Yücel E, Kurt A. Comparison of different fixation methods following sagittal split ramus osteotomies using three-dimensional finite elements analysis. Part 1: advancement surgery-posterior loading. Int J Oral Maxillofac Surg 2005 Jul;34(5):551-558.
38. Erkmen E, Şimşek B, Yücel E, Kurt A. Three-dimensional finite element analysis used to compare methods of fixation after sagittal split ramus osteotomy: setback surgery-posterior loading. Br J Oral Maxillofac Surg 2005 April;43(2):97-104.
39. Maurer P, Holweg S, Knoll WD, Schubert J. Study by finite element method of the mechanical stress of selected biodegradable osteosynthesis screws in sagittal ramus osteotomy. Br J Oral Maxillofac Surg 2002 Feb;40(1):76-83.
40. Pinnojil PK, Mahajan P. Finite element modelling of helmeted head impact under frontal loading: Sadhana 2007 Aug;32(4):445-458.
41. Tang Z, Tu W, Zhang G, Chen Y, Lei T, Tan Y. Dynamic simulation and preliminary finite element analysis of gunshot wounds to the human mandible: Injury: 2012 May;43(5):660-665.
42. Schuller-Götzburg P, Krenkel C, Reiter TJ, Plenk H Jr. 2D-finite element analyses and histomorphology of lag screws with and without a biconcave washer. J Biomech 1999 May;32(5):511-520.
43. Terheyden H, Mühlendyck C, Sprengel M, Ludwig K, Härle F. Self-adapting washer system for lag screw fixation of mandibular fractures. Part II: In vitro mechanical characterization of 2.3 and 2.7 mm lag screw prototypes and in vivo removal torque after healing: J Craniomaxillofac Surg 1999 Aug;27(4):243-251.
44. Roth S, Vappou J, Raul JS, Willinger R. Child head injury criteria investigation through numerical simulation of real world trauma. Comput Methods Programs Biomed 2009 Jan;93(1):32-45.
45. Takada H, Abe S, Tamatsu Y, Mitarashi S, Saka H, Ide Y. Three-dimensional bone microstructures of the mandibular angle using micro-CT and finite element analysis: relationship between partially impacted mandibular third molars and angle fractures. Dent Traumatology 2006;22:18-24.