Volume: 25, Issue:3, July 2014 pp: 175-182

ORIGINAL ARTICLE / ORJİNAL MAKALE

ANALYSIS OF AXIS TRAUMATIC SPONDYLOLISTHESIS BY THE FINITE ELEMENT METHOD

AKSİS TRAVMATİK SPONDİLOLİSTEZİSİNİN SONLU ELEMANLAR YÖNTEMİ İLE ANALİZİ

Doğan Güçlühan GÜÇLÜ[1],
Ömür GÜNALDI[2],
Uzay ERDOĞAN[2],
Ali Ender OFLUOĞLU[2],
Lütfi Şinasi POSTALCI[2],
Ümit KEPOĞLU[2],
Haydar ÇELİK[3]

[1]Neurosurgeon, Bakırköy Dr. Sadi Konuk Education and Research Hospital.
[2]Neurosurgeon, Bakırköy Prof. Dr. Mazhar Osman Mental Health and Neurological Diseases Education and Research Hospital
[3]Neurosurgeon, Ankara Education and Research Hospital, Ankara.

Address: Op. Dr. Uzay Erdoğan, Beyin ve Sinir Cerrahisi Kliniği, Bakırköy Prof. Dr. Mazhar Osman Ruh Sağlığı ve Sinir Hastalıkları Eğitim Araştırma Hastanesi, Bakırköy / İstanbul.
Tel.: 0212 4091515
Fax.: 0212 5703325
Gsm: 0533 4242699
E-mail: uzayerdogan@gmail.com
Received: 15th April, 2014
Accepted: 17th June, 2014

SUMMARY

Aim: Hangman's fracture is fracture of the second cervical vertebra that separates the anterior and posterior elements of the vertebrae. There are very important biomechanical consequences of a hangman's fracture. The finite element method is the most frequently used mathematical method for modeling the cervical vertebrae.

Materials and Methods: In this study, a simulation of clinically-seen hangman's fracture was created and studied using the finite element method. The mechanism and localization of the hangman's fracture was studied by linear static analysis, and migration and tensile analyses were done from the atlantodental joint surface, parallel to the axial plane of the dens axis, considering that there is a 1000 N bending force transmitted from the atlas.

Results: The results of our study are in accordance with the results of experimental cadaveric studies. Thus, this study shows the accuracy of analyses by quantitative methods, as well as the limitation conditions, loadings, and selected element types.

Conclusions: Instead of using an expensive cadaveric method or methods with differences in quality, with the finite element method, it is possible to examine trauma in detail and with high sensitivity.

Key words: Axis, Traumatic spondilolisthesis, Hangman's fracture, Finite element method

Level of Evidence: Experimental study, Level II

ÖZET

Amaç: Hangman kırığı vertebranın ön ve arka elemanlarını ayıran 2. servikal vertebra kırığıdır. Hangman kırığı ile ilişkili çok önemli biomekanik değerlendirmeler vardır. Servikal omurganın matematik modellerinin en sık kullanılanı, sonlu eleman metodudur.

Gereçler ve yöntem: Bu çalışma kapsamındaki simulasyonda, klinikte görülen asılmış adam kırığına daha net bir yorum getirebilmek için asılmış adam kırığının lokalizasyonu ve mekanizması lineer statik analiz ile incelenmeye çalışılmış, yer değişimi ve gerilme analizi gerçekleştirilmiştir. Atlantodental eklem yüzeyinden dens aksisin aksiyal düzlemine paralel olacak şekilde, atlas kemiğinden aktarılan 1000 N'luk bir eğme kuvveti geldiği varsayılarak ve sonlu elemanlar yöntemi kullanıldı. Sonuçlar: Çalışmamamız kadavra çalışmalarının bilinen deneysel sonuçlarına uyum göstermiştir. Sayısal yöntem ile yapılan analizde kabullerin, sınır şartlarının, yüklemelerin ve seçilen eleman tiplerinin doğruluğunu kanıtlar nitelikte olduğu görülmüştür.

Sonuç: Bu yöntemde, travma modeli ile ilgili araştırmaların yapılmasında, kadavra modeli gibi masraflı ve birbirinden nitelik olarak farklı yöntemlerin yapılması yerine sonlu elemanlar yöntemi kullanılarak, travma, ayrıntılarıyla ve hassasiyetle incelenebilmektedir.

Anahtar kelimeler: Aksis, Travmatik spondilolistezis, Asılmış adam kırığı, Sonlu elemanlar yöntemi

Kanıt Düzeyi: Deneysel çalışma, Düzey II

INTRODUCTION:

Although there have been developments in patient care, surgical techniques, and the prevention of complications and rehabilitation in the last few years, cervical region injuries are still important causes of disability and death. Generally, traumatic spondylolisthesis of the axis occurs due to traffic accidents, diving and falls, representing 4–8% of all cervical injuries[5,7]. A hangman's fracture is a fracture of the second cervical vertebra[2,13].

There are very important biomechanical considerations associated with hangman's fracture. Models that have been used in cervical biomechanical experiments are cadaveric, or often mathematical. The finite element method is the most frequently used type of mathematical model of the cervical vertebrae[16,17].

In this study, a simulation of hangman's fracture seen clinically was studied. The mechanism and localization of the hangman's fracture was studied by linear static analysis, and migration and tensile analyses were done.

MATERIALS AND METHODS:

To obtain the necessary initial surface geometry for a model of the axis, the axis bone of a 30-year-old healthy man was scanned in 1 mm sequenced sections using Siemens Somatom Plus-S axial tomography. Sections obtained from the tomography were arranged using the Mimics 7.0 software, and the first geometries obtained provided the necessary surface to begin the design of the C2 and dot cloud. The geometries obtained are shown in Figure-1.

As a result of the scan, these data were evaluated into Initial Graphics Exchange Specification (IGES) format using the Mimics 7.0 program, to make it possible to change the information into other formats to use with different software. The IGES formatted data was used as the initial data to obtain a finite element network by unlocking with the CATIA V5R9 software. Firstly, the geometry was arranged, as surfaces obtained by the Mimics 7.0 software cause problems while obtaining the finite element network.

Arranging the Geometry (CAD) (Computer-aided design):

To arrange the geometry, the CATIA V5R9 software was used. This is multifunctional, computer-aided design, production and engineering (CAD/CAM/CAE) software. The surface modeling skills of CATIA V5R9 helped to remodel and correct the defective surface of the axis.

ANALYSIS:

Element types of the C2 Model:

In the C2 model, elements with linear structure functions were used. This was because functionally high grade elements cause problems during network occurrence because of their complex geometry, and a great amount of the structure is equipped with a discontinuous and irregular finite element network. Particular specifications of the elements used in the analysis are given below.

Cortical Regions – Shell 63 Element: Determination of the cortical bone was performed by the Shell 63 element, that has both bending and membrane sufficiency capabilities.

Trabecular Bone – Solid 45 Element: This is used for 3D modeling of solid structures. The element is determined with eight loops that each have three degrees free (translation in the X, Y and Z directions).

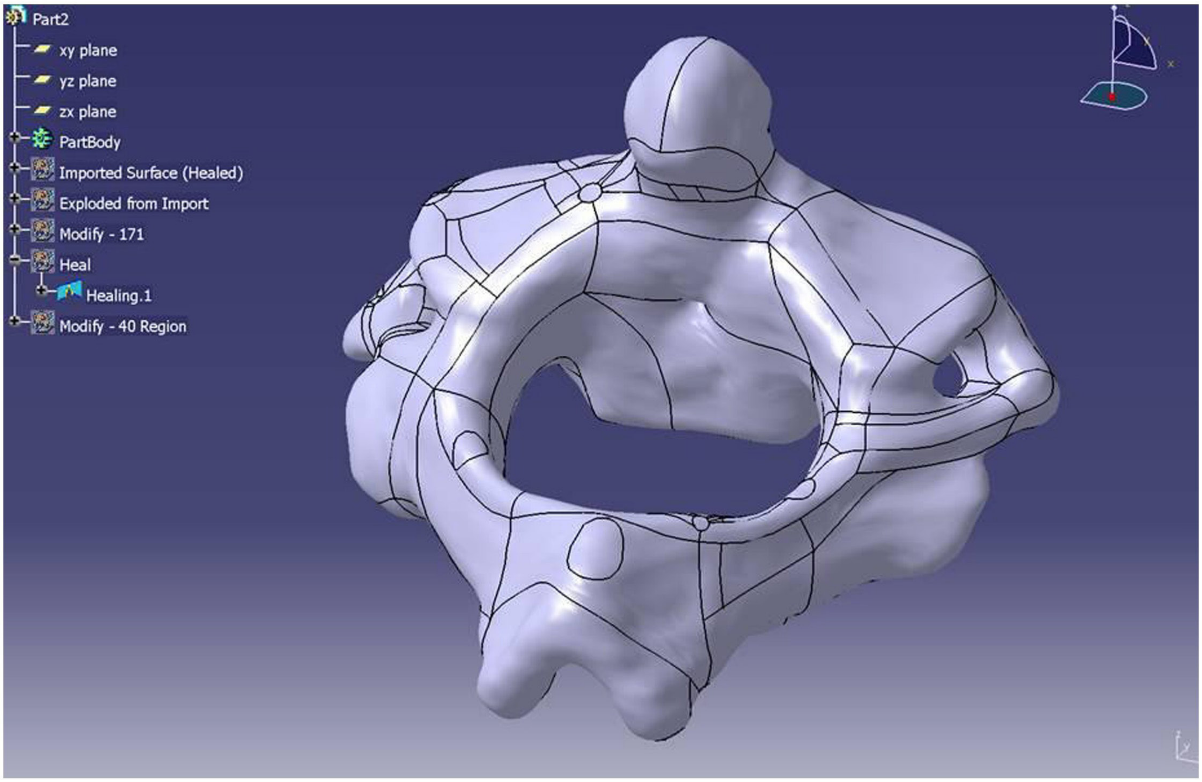


Figure-1. By arranging the defects of axis geometry, an indoor volume has been generated, and with the help of this volume, a 3D model of the axis has been obtained. The model obtained is shown in the figure.

The element has plasticity, stress stability, fluctuation, penetration, wide deflection and wide straining qualities. The element can be determined with eight loops and orthotropic material properties.

Qualifications of the Analysis and the Admittances: To solve the problems with the finite element method, some data needed to be entered into the computer, including coordinates that generated the geometry of the inspected material, convenient element type selection, segmentation of the geometry into elements, properties of the model generating materials, external factors applied to the model (applied force and direction), limitation conditions of the geometry (support coordinates and direction), type of analysis (dynamic, static, electromagnetic), and geometry, evaluated into the right structure for the analysis media from CATIA. Shell 63 and Solid 45 were chosen aselement types.

In this study, in the simulation, the localization and mechanism of a hangman's fracture were studied with linear static analysis. In linear static analysis, it is accepted that the tensile-structure changing linkages of the materials are linear, all behavior during the analysis occurs in the elastic zone, the properties of the material do not change over time, the transposition amount is very small, the applied weights are slow enough to not have an inertial force, and the tensile state is time-independent.

As a result, by examining the tensile state, the initiation and progression pathways of the fracture were studied.

The thickness variations of the cortical bone were ignored and, determining a 1 mm thickness on the surface of all bones, a shell finite network was obtained from the Shell 63 elements. Since the geometry is complex, all shell elements were generated in triangular form in an attempt to optimize the network convenience and to minimize the local errors. The trabecular bone was meshed with tetrahedron Solid 45 elements. Finite element network generating procedures, analyses and visualization steps of the results were performed in an ANSYS environment.

Limitation Conditions and Applied Forces:

Examining the anatomy of the C2 bone despite the hangman's state, the bearing regions and force stream ways were studied. According to this state, since the surfaces of the facet joints lean and stabilize on the C3 bone facets, actions at every free degree and at all loop points on these surfaces were blocked (Embedded Bearing). Since the inferior side of the spinous process of C2 leans on and stabilizes the C3 bone, this region was modeled with embedded bearing.

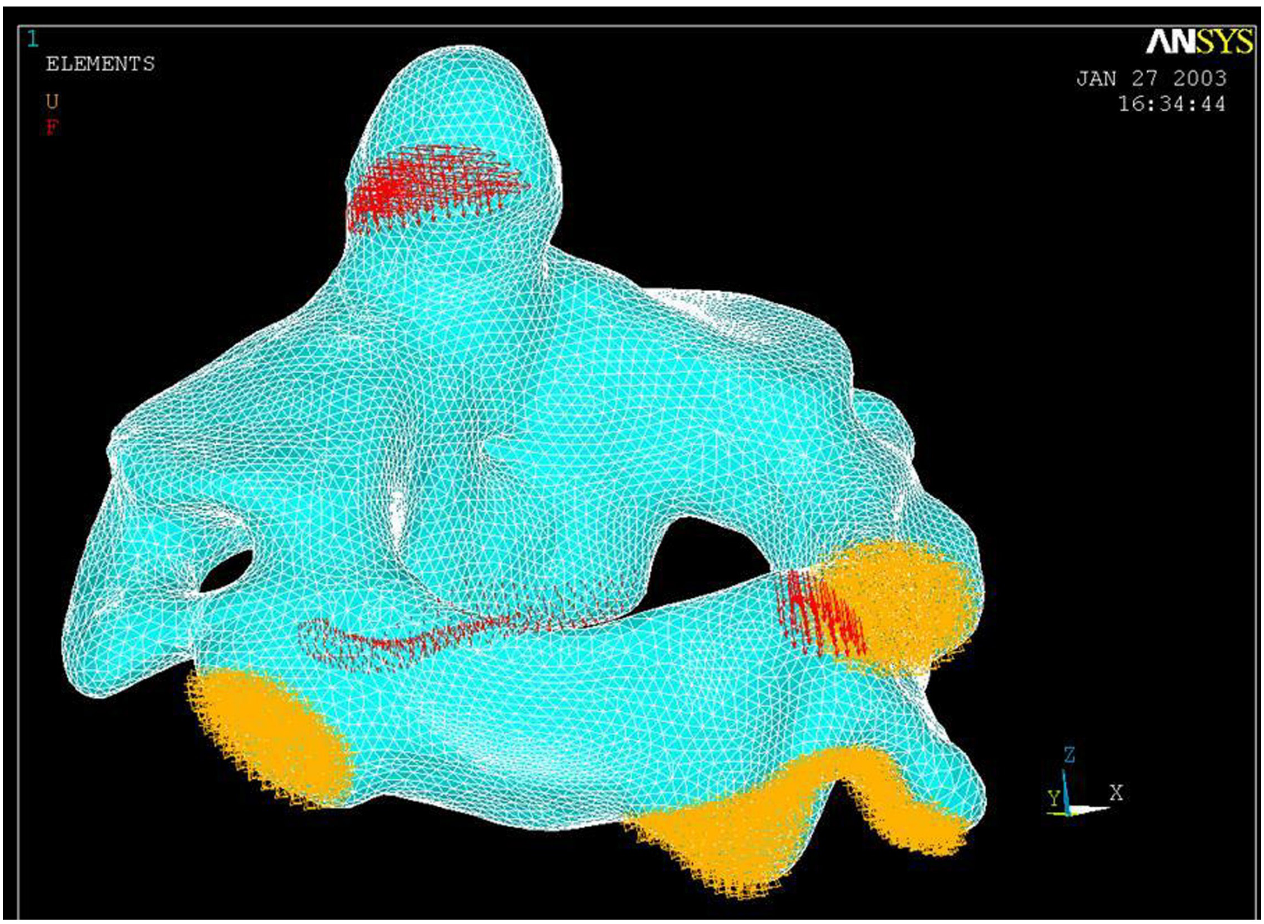


Figure-2. View of applied forces and comparative limitation conditions of finite element networks generated by 2.0 mm and 1.0 mm elements.

Simulating lineages of the inferior surface of the corpus of C2 to C3, and according to the literature, a close resistant force value of 340 N was distributed to the nodes on this surface. Parallel from the atlantodental joint surface to the axial plane of the axis, the C1 bone was considered as generating 1000 N of force as a bending force. As a result of lineages coming from the atlas, the top of the odontoid process was charged with 340 N, which is an average binding resistance. The charging state view is given in Figure-2.

The determination of 1000 N of force was consistent with the literature (the bond tensile force at C2 was shown to be 1000 N in an experimental article) with these admittances[12]. On the other hand, for a person who falls from a particular height, the falling speed is $V = \sqrt{2gh}$ and the approximate force at the moment of hitting is $F = (\Delta t/\Delta V)m$. 1000 N is an force consistent with analysis of hangman's physiology for an approximately 80 kg person who falls from a height of 0.7 m, with rope resilience reducing velocity rapidly from 0.3 sec to 0.

The material properties of the bone, consistent with the literature (Young's modulus, Poisson rate and bone density) were considered. The bone was considered completely with its cortical and spongious bone qualifications, which cause it to differ from trabecular bone. Spongious bone, because of its porous and irregular properties, is anisotropic.

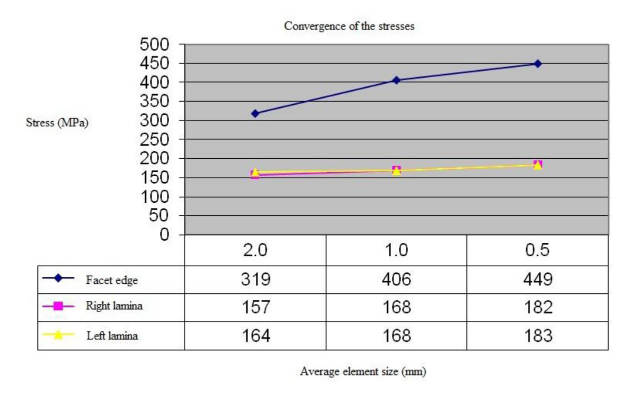
Studies have shown that the anisotropy of spongious bone can be ignored. Thus, according to the results of our study, spongious bone may be accepted as a homogeneous material.

According to the approach of linear attitude aspect of materials, for cortical and trabecular bone, the material properties are accepted as shown in Table-1.

Table-1. The material characteristics below were accepted for cortical and trabecular bone, according to their linear behavior.

	Cortical Bone	Trabecular Bone
Module of Elasticity (E)	10 GPa	1 GPa
Poisson Ratio (γ)	0.3	0.3
Density (σ)	1300 kg/m³	130 kg/m³

Table-2. Table showing loop node numbers and amount of elements for each element size for cortical and trabecular bone.



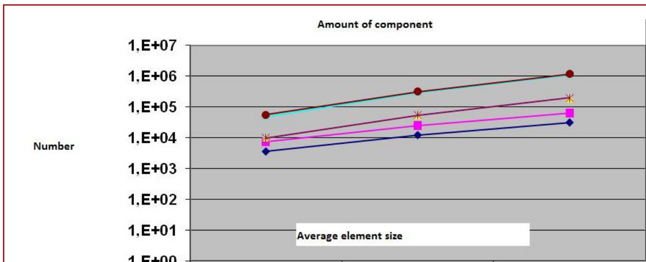
RESULTS:

In finite element solutions, one of the most important criteria is to consider the sensitivity of the obtained solutions to the finite element network's average element dimensions. This is termed convergence. The ideal element dimensions must be chosen to be as small as possible. However, in a real situation, optimization is necessary because of the increased computer source needs, and also an increase in the number of elements with numerical errors. Sensitivity to the extents, especially in linear elements 160, has increasingly vital importance, because the derivation values that have been used in the calculation of this type of tensile model show no continuous state.

At the same limitation conditions and with the same material, the observed C2 model meshed with three differently sized elements was analyzed separately. The loop node numbers for each element size and amount of elements are given for cortical and trabecular bone in Table-2.

The maximum tensile strength values obtained from these analyses, and high tensions at certain points, is given for each element size in Table-3 and Figures-3,4. The total sum of the motions of the bone and the variation of the element sizes at the intersections is given in Figure-5.

Table-3. The figure demonstrates variations according to element size of maximum tensile strength values and high tensions at certain points obtained from these analyses.



	2.0	1.0	0.5
Shell 63 Nodes	3664	12369	31475
Shell 63 Elements	7336	24746	62958
Solid 45 Nodes	9899	54298	196185
Solid 45 Elements	48129	289412	1085084
Total Nodes	9899	54298	196185
Total Elements	55465	314158	1148042

It is seen that obtaining a finite element network with a 2 mm average element size is not successful at simulating the real state. It is observed that 1 mm meshes are best to use, as 0.5 mm meshes cause a calculation problem and there is also a memory issue with the necessity for numerous elements. Analyzed results of the consistency of a 1 mm sized finite element network are given in Figure-6.

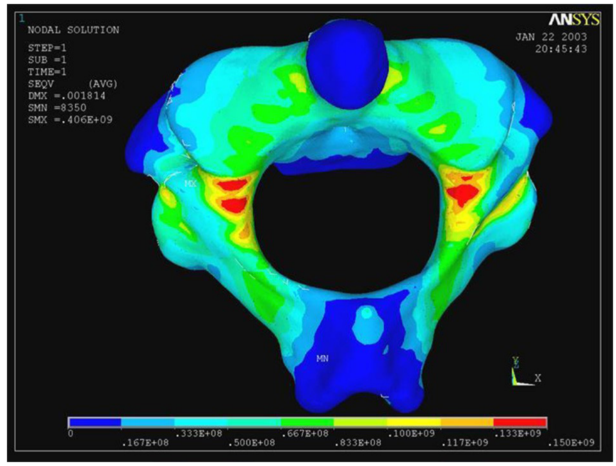


Figure-3. Finite Element Solution generated by 1.0 mm elements – Tensile Contours (top view).

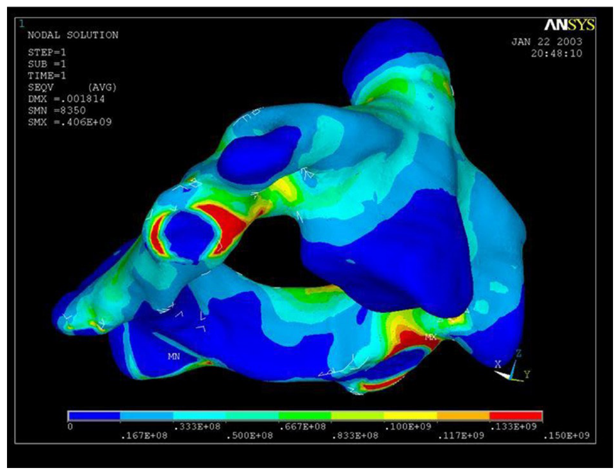


Figure-4. Finite Element Solution generated by 1.0 mm elements – Tensile Contours (bottom view).

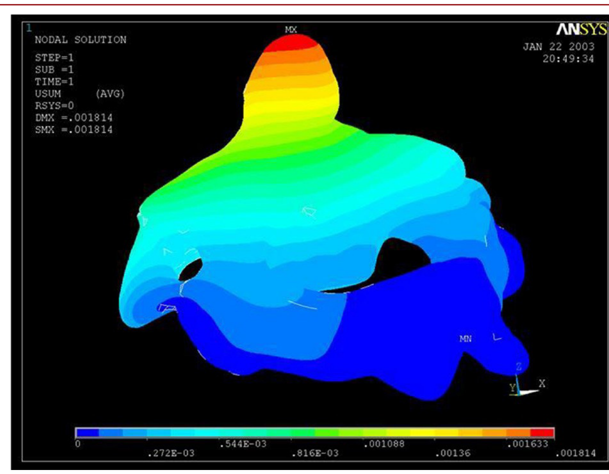


Figure-5. Finite Element Solution generated by 1.0 mm elements – Total Displacement Contours (oblique view).

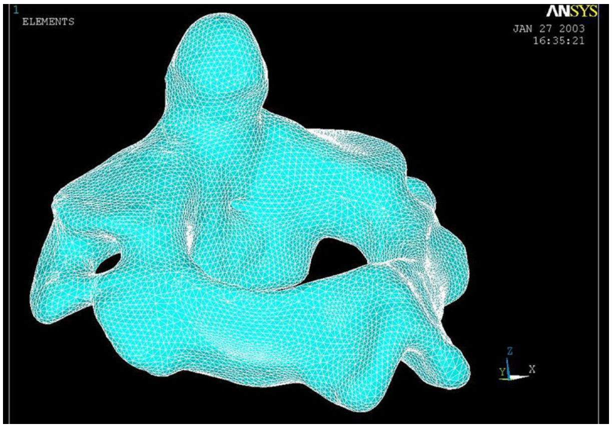


Figure-6. Comparative views of finite element networks generated by 1.0 mm elements.

DISCUSSION:

Wood-Jones first determined the cervical fracture dislocation mechanism in 1913 [14]. Garber first coined the term 'axis traumatic spondylolisthesis' in 1964 [4]. Schneider was the first to use the term 'hangman's fracture[10]. A hangman's fracture is a second cervical vertebra fracture that separates the anterior and posterior elements of the vertebra. The fracture is either at the anterior side of the lateral masses or at the pedicle area of the vertebra[2,13].

In nominal profit and car accident trauma, the weight mostly affects the pars interarticularis. Since the bone section is very small, this region is the weakest and the most sensitive to hangman's fracture[3].

According to the literature, cadaveric and mathematical methods are common methods for cervical biomechanical experiments. Human cadaveric models have geometric and structural advantages. In vitro experimental results are the most acceptable results for biomechanical approaches. These models can be developed with surgical techniques. Biological variations, differences and expenditures are the limitations of these models. To form a statistical model, it is necessary to have numerous samples[15]. In spinal analytic studies, mathematical models can be classified according to the geometry of the model used, the applied force, the analysis, and the application. The geometry of the model can be a complete system (such as the column or complete cervical), or only a part of the system (such as a functional spinal unit). The force can be statistically loaded by ignoring time and velocity, or it can be dynamically loaded at a certain time and at a certain velocity. Mechanical analysis of the material can be done as a linear analysis with previously known geometry, deformation, and limitation and loading conditions, or by changing these conditions it can be done as a non-linear analysis. Mathematical models can be discussed according to the strength of the material or according to trauma mechanisms that have been observed in clinics. A computer simulation model can be generated from a group of mathematical equations related to the physical and geometric properties of the figurative structure.

The most common mathematical model for the cervical vertebra is a finite element method. Kinematic (intervertebral motions), mechanic (motion as an effect of the applied force), internal tensile and replacement can be the subjects of the study. The finite element method represents some ideal probabilities at checkpoints, such as the abilities to repeat, change any parameter, and convert the quantity and quality of secondary effects on the diversity of certain parameters[1,8]. The effects of instrumentation and surgical attempts during the application of physiological and traumatic loadings were studied. Determination of various anatomical elements can be discussed with this method[15].

The use of mathematical models in a biomechanical area began in 1969 with 2D analysis by Orne and Liu, and then 3D models, followed by the finite element method, were developed[9]. The 3D anatomy of cervical vertebrae was determined by defragmentation of 1 mm periodic BT sections by Yoganannan et al. for the first time in 1996 [9,15].

A similar study to ours was carried out to determinate the mechanism of atlas fractures by Teo et al. in 2001, and with this method, results that were most consistent with the original results were shown[11]. Graham et al.[6] examined the mechanism of odontoid process fractures by this method, and also showed results that were the most consistent with the original data.

In conclusion, the results of our study support the conventional experimental results shown in Teo's cadaveric study; therefore, this study shows the accuracy of the assumptions in the analysis by a quantitative method, the limitation conditions, loadings and selected element types. Although nowadays nominal profit is vanished, there is a new factor in the etiology of hangman's fracture, and it has become a current issue due to airbag accidents. Instead of using expensive cadaveric methods and methods of varying quality, by using the finite element method, it is possible to examine the trauma in detail with a high sensitivity.

REFERENCES:

1. Bozkus H, Ames CP, Chamberlain RH, Nottmeier EW, Sonntag VK, Papadopoulos SM, Crawford NR. Biomechanical analysis of rigid stabilization techniques for three-column injury in the lower cervical spine. *Spine* 2005;30(8): 915-922.
2. Brasher HR, Venters GC, Preston ET. Fractures of the neural arch of the axis. A report of twenty-nine cases. *J Bone Joint Surg* 1975; 57-A: 879-987.
3. Fielding JW, Francis WR Jr, Hawkins RJ, Pepin J, Hensinger R. Traumatic spondylolisthesis of the axis. *Clin Orthop Relat Res* 1989; 239:47-52.
4. Garber JN. Abnormalities of the atlas and axis vertebrae-congenital and traumatic. *J Bone Joint Surg1964*; 46-A: 1782-1791.
5. Gehweiler JA Jr, Clark WM, Schaaf RE, Powers B, Miller MD. Cervical spine trauma: the common combined conditions. *Radiology* 1979; 130(1): 77-86.
6. Graham RS, Oberlander EK, Stewart JE, Griffiths DJ. Validation and use of a finite element model of C-2 for determination of stress and fracture patterns of anterior odontoid loads. *J Neurosurg* 2000; 93: 117-125.
7. Hardley MN, Sonntag VKH, Grahm TW, Masferrer R, Browner C. Axis fractures resulting from motor vehicle accidents. *Spine*1986; 11: 861-864.
8. Panjabi MM. Cervical spine models for biomechanical research. *Spine* 1998; ;23(24): 2684-2700.
9. Saito T, Yamamuro T, Shikata J, Oka M, Tsutsumi S. Analysis and prevention of spinal column deformity following cervical laminectomy. I. Pathogenetic analysis of postlaminectomy deformities. *Spine* 1991; 16(5): 494- 502.
10. Schneider RC, Livingstone KE, Cave AJ, Hamilton G. "Hangman's 307 fracture" of the cervical spine. *J Neurosurg* 1965; 22: 141-154.
11. Teo EC, Ng HW. First cervical vertebra (atlas) fracture mechanism studies using finite element method. *J Biomech* 2001; 34(1): 13-21.
12. Teo EC, Paul JP, Evans JH, Ng HW. Experimental investigation of failure load and fracture patterns of C2 (axis). *J Biomech* 2001; 34(8): 1005-1010.
13. Willams TG. Hangman's fracture. *J Bone Joint Surg* 1975; 57-B: 82-88.
14. Wood-Jones F. The ideal lesion produced by judicial hanging. *Lancet* 1913; 1: 53.
15. Yoganandan N, Kumaresan S, Voo L, Pintar FA. Finite element applications in human cervical spine modeling. *Spine* 1996; 21(15): 1824-1834.
16. Yoganandan N, Kumaresan S, Voo L, Pintar FA. Finite element model of the human lower cervical spine: parametric analysis of the C4-C6 unit. *J Biomech Eng* 1997; 119(1): 87-92.
17. Yoganandan N, Myklebust JB, Ray G, Sances A Jr. Mathematical and finite element analysis of spine injuries. *Crit Rev Biomed Eng* 1987; 15(1): 29-93.