Ege Journal of Medicine / *Ege Tıp Dergisi* 2016;55(3): 117-121

Analysis of plate-screw fixation by finite element method in transverse fractures of the tibia diaphysis

Tibia diafiz transvers kırığının plak-vida ile tespitinin sonlu elemanlar yöntemi ile analizi

Emre Bayramoğlu[1] Yılmaz Tomak[2] Alper Çıraklı[3] Hasan Göçer[2] Birol Gülman[2]

[1]Echomar Private Hospital, Clinic of Orthopedics and Traumatology, Zonguldak, Turkey
[2]Ondokuz Mayıs University Faculty of Medicine, Department of Orthopedics and Traumatology, Samsun, Turkey
[3]Kayseri Training and Research Hospital, Clinic of Orthopedics and Traumatology, Kayseri, Turkey

Abstract

Aim: Using the Finite Element Method, it was aimed to select the most appropriate implant for a patient with a transverse tibia diaphysis fracture and to determine the effect of sudden temperature changes on the selected implant.

Materials and Methods: After separating the diaphyseal area into transverse pieces at 1 cm regular intervals, the anterior and lateral views of the cross-sections enabled the generation of a computer model of tubular bone with 3 cortices. An 8-hole plate with 8 screws was placed over the lateral cortex of the tibia on the model. After creating a transverse fracture in the exact centre of the plate, the whole model with bone-plate-screw was converted into a mathematical model (mesh). The results of the analysis were evaluated using the finite elements method.

Results: The tensile values were found to be greatest on the 2nd holes and screws from the proximal and distal of the fracture line. In the evaluation of temperature changes, the plate-screw tensile values were determined as normal in a cold environment but were seen to be high compared to those of a hot environment.

Conclusion: By finding the tensile values on plate-screw-bone with this method preoperatively, the most appropriate plate-screw to be used on the patient can be determined. In addition, as tensile force on the plate-screw increased with transfer from a hot environment to a cold one, it could be said that fractures are more likely to occur in a cold environment.

Keywords: Tibia fracture, bone model, biomechanics, analysis, finite element method.

Öz

Amaç: *Sonlu Elemanlar Yöntemi ile hasta için en uygun implantın seçilebilmesi ve seçilen implant üzerinde ani ısı değişikliklerinin etkisinin saptanması amaçlanmıştır.*

Gereç ve Yöntem: *Diafiz bölgesi 1 cm düzenli aralıklarla transvers olarak parçalara ayrıldıktan sonra ortaya çıkan kesitler, kemiğin önden ve yandan görünümleriyle bilgisayar ortamında birleştirilerek üç korteksi olan tübüler kemik modeli oluşturuldu. Model üzerinde 8 delikli plak ve 8 adet vida tibianın lateral korteksi üzerine oturacak şekilde yerleştirildi. Plağın tam ortasından transvers bir kırık oluşturulduktan sonra kemik-plak-vida ile oluşan tüm model, matematik model (örgü) haline getirildi. Analiz sonuçları sonlu elemanlar yöntemi kullanılarak değerlendirildi.*

Bulgular: *Kırık hattına proksimaldeki ve distaldeki en yakın 2. delikte ve vida üzerinde gerilme değerinin en fazla olduğu bulundu. Isı farklarını değerlendirdiğimizde de soğuk ortamdaki plak-vida gerilme değerlerinin normal, ancak sıcak ortamdaki gerilme değerlerine göre yüksek olduğu görüldü.*

Sonuç: *Bu yöntem ile cerrahi öncesi plak-vida-kemik üzerindeki gerilme değerleri bulunarak hastaya kullanılabilecek en uygun plak-vida belirlenebilir. Ayrıca sıcak ortamdan soğuk ortama geçtiğinde plak-vida üzerindeki gerilme kuvvetinde artış olduğundan kırılmaların soğuk ortamda daha fazla olduğu söylenebilir.*

Anahtar Sözcükler: *Tibia kırığı, kemik model, biomekanik, analiz, sonlu elemanlar yöntemi.*

Corresponding Author: Alper Çıraklı
Kayseri Training and Research Hospital, Clinic of Orthopedics and Traumatology, Kayseri, Turkey

Received: 20.06.2015 Accepted: 26.08.2015

Introduction

Tibia fractures are the most common long bone fractures (1). The method of treatment should be determined by taking into account the structure of the fracture, the amount of energy applied to the extremities, mechanical properties of the bone, the general condition of the patient, age, and most importantly, the soft tissue condition of the limb (2). In the treatment of these fractures a variety of options are available, including plaster or functional brace, plate-screw and intramedullary nail with open or closed reduction and internal or external fixation techniques. Although intramedullary nail is the most common technique, many complications of nailing have been reported, including malalignment of the fracture and knee pain (3).

The plates that are used are made of different alloys in different sizes depending on the shape of fracture. Today, the main problem in plate-screw fixation is the loosening and/or fracture that are formed due to the load on the plate and screw (4,5).

The aim of this study was to determine the most suitable implant for tibial diaphyseal fractures where plate-screw fixation is considered, by generating a transverse fracture on the bone-plate-screw model created in a computer environment and making calculations using the finite element method (FEM). The hypothesis was that based on these results, the most suitable implant for the patient could be determined prior to surgery. In addition, it was aimed to observe the response of the plate-screw combination to the changes in stress values due to sudden temperature changes.

Materials and Methods

Modeling Process

The bone model used was of the same dimensions as the average human tibia. After transferring the front, rear and side image data of the whole bone to the computer, a computerized model of the tibial diaphyseal cortex was obtained in 3 parts from the cross-sections created transversely by dividing the diaphyseal region of the existing artificial bone at 1 cm intervals with an electric saw. The lateral cortical thickness of the model was accepted as 5 mm, the medulla width as 13 mm and the medial cortex thickness as 5.25 mm. A transverse fracture was then created on the model and a plate 13 cm long and 4 mm thick with 8 holes and cortical screws of 4.5 mm diameter, which was usually used in orthopedic practice, was placed to fit onto the lateral cortex of the bone. At the proximal and distal of the fracture line, care was taken that the screws were applied symmetrically for fixation of the plate.

The screw tightening force at the time of fixation (also pre-load of the screw), was considered to be 50 Newton (N) and 500 N respectively. In this bone-plate-screw model it was assumed that the average human weight is 80 kg and by applying 80 kg axial load it was intended to find the stress values on the bone-plate and screw. In addition, another objective of the study was to evaluate the response of the bone-plate-screw model to sudden changes in temperature based on the changes in the stress values.

Analysis

FEM was used to perform stress analysis. The analysis was applied in a workstation configuration computer using the ANSYS (Ansys Inc. USA) program. In the first step of the study, a geometric model of the bone was created in the computer environment with the sections obtained from the artificial bone. In the second step, this model was converted into a two-dimensional model file (IGES) format and in the third step, the IGES file was transferred into the three-dimensional (3D) finite element program. With the use of this program, it is possible to apply boundary conditions (load applied to the bone, heating the plate and the screws) and meshing of the geometric model.

Maximum loading conditions were considered in the results obtained and, as in most finite element studies, the Von Mises energy criteria were used, as they have been reported to be sufficient in calculating the stress condition numerically. In this study, the results were compared according to this energy criterion.

In the model used, the critical region was the location of the maximum stress on the implant-bone interface. When the load was increased, the first region to disrupt the structural integrity was the region where the Von Mises stress was maximum and the values were classified from low to high. According to this classification, the case giving the minimum value would be the most advantageous while the case giving the highest value would cause a disadvantage due to the probability of deformation and fracture.

Under thermal analysis, the force of the steel plate used in the fixation and the tension stress on the plate and screws created by temperature differences on the bone were investigated. There are about 20 kg of bones in the human body where the weight of the plate is accepted as 200 g When the difference between their heat capacities is considered, it is expected that the plate would react faster than the bone in the case of temperature difference. At both hot and cold temperatures, the boundary conditions were accepted as the screw clamping force (screw preload) of 500 N, and sudden temperature change of 20°C in the area of location. Until the time when the metal and bone temperatures became the same, the plate and screw stress values were again calculated by FEM.

Results

Stress values were examined in both plates and it was found that they were maximum at the second nearest hole and screw to the fracture line. The reverse side of the plates was evaluated and again the maximum stress value was detected at the second nearest hole to the fracture line. The analysis results of the screw models are illustrated in Figure-1. As a result, the maximum stress was determined to be located on the 2nd closest screw to the fracture line. In addition, stresses on all of the mounted screws were found to be higher for a 500 N (50 kg) load (Figure-2).

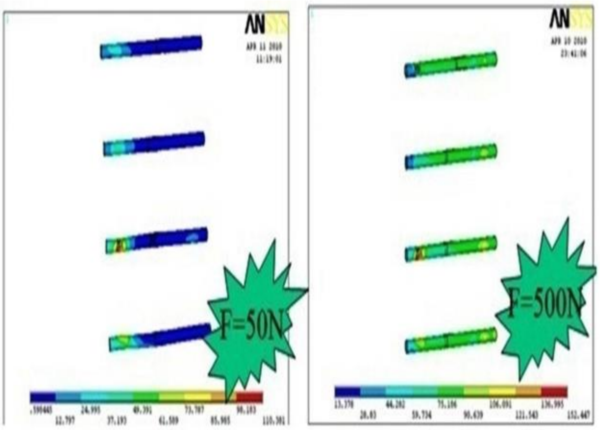

Figure-1. Stress values in screws for different (50 and 500 N) preloads (N: Newton).

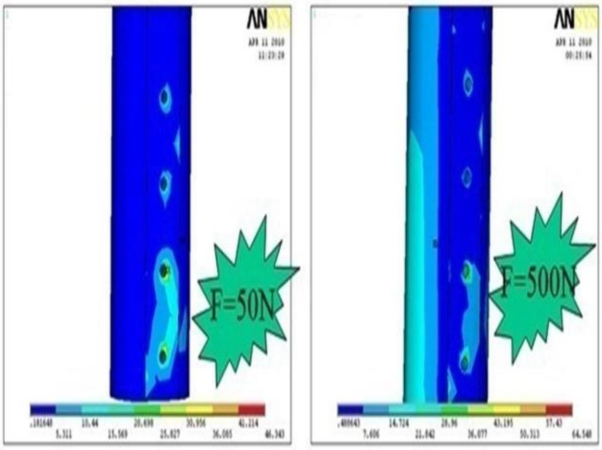

Figure-2. Stress distributions in the bone close to the fracture zone.

The tensile force on the bone inner surface was calculated as approximately 3 times higher when 500 N was applied. The stress when screws were firmly tightened was three times higher than when screws were tightened with less power. In the evaluation of the fracture line, it was observed that the maximum stress value was three times higher for preloaded cases compared with no preload. When the preload on the screws was less there was more contact between the broken ends than in preloaded case. This increases the safety of fixation (Figure-3).

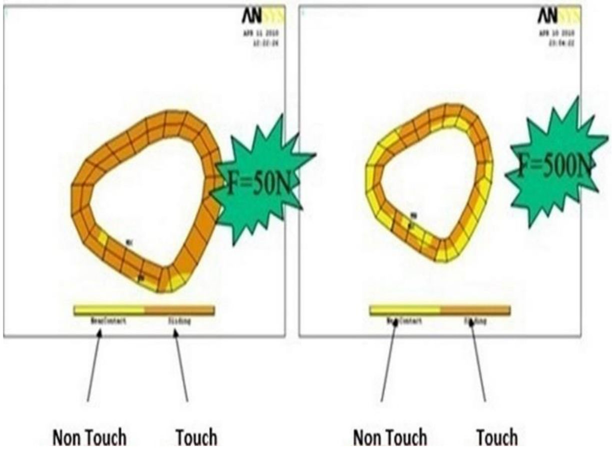

Figure-3. The change in the contact zone for different (50 and 500 N) preloads (N: Newton).

In plate-screw osteosynthesis cases in which steel and titanium plates are used, 85% of the axial load is supported by the bone, whereas 15% is supported by the plate.

The two conditions where the metal was warmer or colder than the bone were tested. When the plate was moved from a warm to a cold environment, with exposure to an environment 20°C colder, the anterior stresses around the 2nd hole became maximal as the maximum tensile force increased. Stress was determined as 152x2.5 MPa at the nominal temperature, and 165x2.5 MPa in the colder environment. The environmental models corresponding to the cases where the bone temperature was either 20°C warmer or colder than the plate temperature are shown in Figure-4.

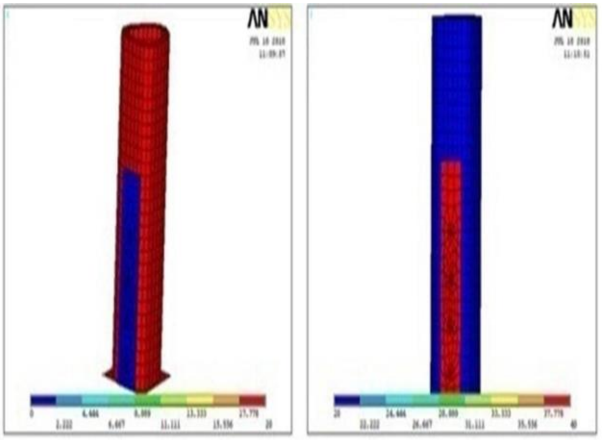

Figure-4. Temperature distributions (cold and hot temperature values) used in the finite element models are shown.

The stresses were lower on the front side of the plate when it was moved to a warmer place. The highest stress was determined at the 2nd hole, while the stress values around all the holes were close to each other. The maximal stress was located between the 1st and 2nd holes (Figure-5). It was determined that the stress on the screws was reduced in the warmer environment compared to the colder environment.

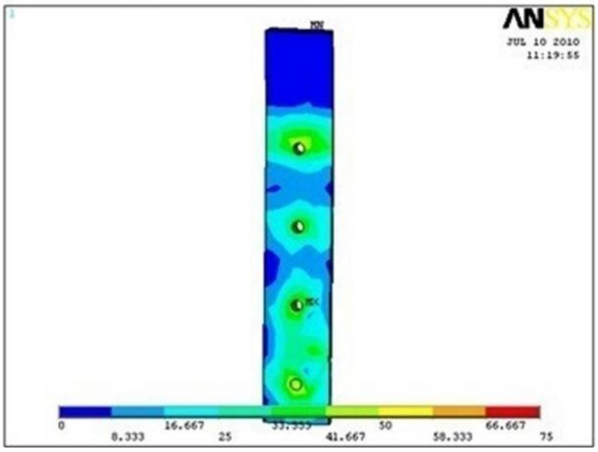

Figure-5. Stress distribution on the frontal face of the plate at 20º C.

Discussion

In recent years, computer aided analysis methods have gained importance as a result of advances in the medical sector. Therefore, many products related with the biomedical field can be analyzed before being made available to patients. Of the stress analysis methods, many studies have shown that 3D FEM in the biomedical field is superior to other methods and the results obtained are in accordance with those obtained in in-vitro studies (6).

Therefore, it was decided to use 3D FEM in this study to evaluate the analysis due to its various advantages, such as localizing the stresses on the plate, screws and in every region of the bone and realistic simulation of the plate and screws (7). The aim of stress analysis with FEM is to investigate the weak points when a structural deformation occurs on structures under a load (8,9).

This method gives information about the fracture moments on the bone, plate and the screws depending on the forces applied on the bone, plate and screws. Apart from the surgical practice and patient-related factors, the success of plate-screw osteosynthesis is based on the manner of transferring the stresses created by the plate and screw to the bone. The force transferred from the plate and the screw to the bone depends on the magnitude and the direction of the force, the interface between the bone and the plate, the length and width of the plate and the screw, the surface properties of the plate and the quality of the implanted bone fixation (10,11).

It has been reported in the literature that FEM gives more realistic results when applied to a 3D model rather than two-dimensional models (12,13). In the 1990s, Meijer et al. (14,15) compared 2D and 3D FEM and concluded that it was necessary to use 3D FEM. With the use of 3D FEM and the large number of variables in the analysis there are a great number of model types.

In the thermal analysis using FEM, boundary conditions were applied for both a cold and warm environment respectively. Evaluating the results accordingly, instantly moving to a colder place increased the stresses on the plate and the screw. Therefore, it could be inferred that the possibility of fracture increases in cold places.

In the light of that information, although FEM is an effective means of analysis in biomechanical studies, it is clear that to obtain accurate results from the analysis, the model must be as realistic as possible. Other important factors necessary for valid results are model geometry, and appropriate definition of the material properties and boundary conditions. To date, biomechanical studies conducted on animals and cadavers have been taken as reference when evaluating the reliability of different fixation materials and designs for fracture fixation (4,5). Therefore, more widespread use of FEM in biomedical studies would lead to savings in time and resources, thereby promoting the success of orthopedic trauma surgery and expanding horizons.

Conclusion

With this method, the optimal plate-screw to be applied to a patient can be determined before surgery by finding the stress values on the plate-screw-bone. In addition, when moving from a hot to a cold environment, it can be said that the probability of fractures may be increased due to the relatively higher tensile stresses on the plate and screws.

Acknowledgements

The authors thank Bülent Acar for his contributions to the study.

Conflict of interest

The authors declare that there is no conflict of interest.

Funding sources

The authors declare that there is no financial support.

References

1. Bhandari M, Guyatt G, Tornetta P, et al. Randomized trial of reamed and unreamed intramedullary nailing of tibial shaft factures. J Bone Joint Surg 2008;90(12):2567-78.
2. Chapman MW. Fractures of the tibial and fibular schafts. In: Evarts CM, editor. Surgery of the Musculoskeletal System. New York: Churchill Livingstone; 1990:3741-825.
3. Zhang J, Ebraheim N, Li M, et al. External fixation using locking plate in distal tibial fracture: A finite element analysis. Eur J Orthop Surg Traumatol 2015;25(6):1099-104.
4. Blake CA, Boudrieau RJ, Torrance BS, et al. Single cycle to failure in bending of three standart and five locking plates and plate constructs. Vet Comp Orthop Traumatol 2011;24(6):408-17.
5. Miller EI, Acquaviva AE, Eisenmann DJ, Stone RT, Kraus KH. Perpendicular pull-out force of locking versus non-locking plates in thin cortical bone using a canine mandibular ramus model. Vet Surg 2011;40(7):870-4.
6. Stegaroiu R, Kusakari H, Nishiyama S, Miyakawa O. İnfluence of prosthesis material on stress distribution in bone and implant: A three dimensional finite element analysis. Int J Oral Maksillofac Implants 1998;13(6):781-90.
7. Marie C. Strength analysis of clavicle fracture fixation devices and fixation techniques using finite element analysis with musculoskeletal force input. Med Biol Eng Comput 2015;53(1):759-69.
8. Akça K, Çehreli MC, İplikçioglu HA. Comparison of three-dimensional finite element stress analysis with in vitro strain gauge measurements on dental implants. Int J Prosthodont 2002;15(2):115-21.
9. Yildirim AO, Alemdaroğlu KB, Yuksel HY, Öken ÖF, Ucaner A. Finite element analysis of the stability of transverse acetabular fractures in standing and sitting positions by different fixation options. Injury 2015;46S(2):29-35.
10. Geng J, Tan KBC, Liu G. Application of finite element analysis in implant dentistry: A review of the literature. J Prosthet Dent 2001;85 (6):585-98.
11. Williams KR, Watson CJ, Murphy WM, Scott J, Gregory M, Sinobad D. Finite element analysis of fixed prostheses attached to osseointegrated implants. Quintessence Int 1990;21(7):563-70.
12. Menicucci G, Mosolov A, Mozzati M, Lorenzetti M, Preti G. Tooth-implant connection: Some biomechanical aspects based on finite element analyses. Clin Oral Implants Res 2002;13(3):334-41.
13. Sertgöz A, Güvener S. Finite element analysis of the effect of cantilever and implant length on stress distribution in an implant-supported fixed prosthesis. J Prosthet Dent 1996;76(1):165-9.
14. Meijer HJA, Starmans FJM, Bosman F, Steen WHA. Comparison of three finite element models of an edentulous mandible provided with implants. J Oral Rehabil 1993;20(2):147-57.
15. Meijer HJA, Starmans FJM, Steen WHA. Location of implants in the interforaminal region of the mandible and the consequences for the design of the superstructure. J Oral Rehabil 1994; 21(1):47-56.