JCAMECH

Vol. 51, No. 1, June 2020, pp 91-97
DOI: 10.22059/jcamech.2020.299796.494

An investigation of tensile strength of Ti6Al4V titanium screw inside femur bone using finite element and experimental tests

*Peyman Mashhadi Keshtiban[a, *], Milad Regbat[a], Mohsen Mashhadi Keshtiban[b]*

[a] *Faculty of Mechanical Engineering, Urmia University of Technology, Urmia, Iran*
[b] *Department of Mechanical Engineering, Tarbiat Modares University, Tehran, Iran*

ARTICLE INFO

Article history:
Received: 22 March 2020
Accepted: 30 May 2020

Keywords:
Finite Element
Tensile Test
Femur Bone

ABSTRACT

The geometric optimization of orthopedic screws can considerably increase their orthopedic efficiency. Due to the high geometric parameters of orthopedic screws, a finite element simulation is an effective tool for analyzing and forecasting the effect of the parameters on the load-bearing capacity of different types of screws and bones. Thus, in the present study, the tensile strength of a typical cortical titanium screw was investigated by the finite element method, and experimental tests confirmed the obtained results. The behavior of the screw in the tensile test was discussed in terms of stress, force, and displacement. The maximum force results show a 14% difference between simulation and experimental works in tensile type loading. Moreover, it was suggested that the trend of force curves in both the experimental test and numerical simulation shows high similarity, and FEM predicts the process with acceptable accuracy. Furthermore, it was concluded that the stress values are higher while moving toward the top surface of the bone.

1. Introduction

Extensive ranges of orthopedic screws with different specifications and material properties are used to improve bone fracture. Without screws, the use of numerous fixing devices, such as bone plates, intramedullary nails, and even some common prostheses, will be less effective or impossible. In experimental situations, the subjected screws are under various types of loads, such as bending, tensile or compression, etc.

Orthopedic screws, which are employed to increase strength and resistance to failure, always play an important role in attaching orthopedic implants to the bone. In order to optimize design, which reduces destructive effects and improves the efficiency of the implant, engineers frequently use numerical simulations in combination with optimization algorithms. Therefore, one of the main objectives of the implant components is to strengthen the bone and reduce the breakdown. Although empirical studies can considerably assist in improving the screw connection results, many parameters are difficult to detect without using finite element simulations. Researchers have conducted a large number of studies to simulate bone conditions, orthopedic plaques, and screws. Kim et al. [1] used the Taguchi and finite element methods to optimize the screw and plaque geometry. They concluded that the proposed plaques and screws could provide excellent information for applying the smallest sizes in clinical applications. In another work, Hou et al. [2] used the finite element method and laboratory tests to obtain bone conservation power by screws. They concluded that the bone conservation strength by orthopedic screws could be calculated by finite element analysis, while the effect of all geometric factors can be calculated by simulation even for unspecified threaded screws. Chao et al. [3] employed mechanical testing and finite element simulation to increase the bending strength of the Tibial screw. It was concluded that the finite element is efficient in terms of time and cost and also could be used as a strong tool to achieve the strength of the screws. It was also concluded that as the thread radius and the fatigue strength of titanium screw increased compared to the steel screws. In another investigation, Haase et al. [4] used finite element simulation to obtain stress and strain profiles in orthopedic screws. Wu et al. [5] used the finite element method to evaluate lubricant effect on the reliability of the dental implant screw. The result reveals that the lubricated screw closes with less torque, resulting in a loosening of the connection. Therefore, the lubricant application in this type of fitting is not appropriate. A study was conducted by Ketata et al. [6] on bicortical orthopedic screws both experimentally and numerically. 3D finite element analysis was performed on the pull-out tests. It was indicated that self-tapping screws ensure tighter bony contact leading to higher pull-out strength; however, non-self-tapping screws improves the stiffness of the fixation. Mau et al. [7] designed anterior cruciate ligament interference screw using the finite element method. Magnesium based screws, which support bone healing, were used in the study. It was concluded that the device failure decreases, and then the surgical success rates increase at insertion. Naidubabu et al. [8] used the finite element method to obtain stress distribution in a fractured bone. Three different implant materials, i.e., magnesium, 316L steel, and titanium, were studied and compared. It was inferred that magnesium implant reduced the stress shielding effect, and

* Corresponding author. Tel.: +98-912-276-5160; fax: +98-044-319-80251; e-mail: m.keshtiban@mee.uut.ac.ir, pmkmech@gmail.com

then it was suggested as the best implant material. Yan et al. [9] investigated 3D- printed orthopedic implants. The fracture fixation was evaluated by finite element analysis of implant and bone. Young modulus and unused screw holes can be changed easily by the 3D-printing process. The subjected modifications were made to discover the related effects on fracture fixation. Li et al. [10] studied the neck fracture using finite element analysis. Three different neck fracture fixation implants were compared in terms of stress distributions, rotational angles, and stress peaks. Tetteh and McCullough [11] focused on the thread profile and the effect of shape on the rate of stress transfer in bone using finite element analysis. It was concluded that the trapezoidal thread shape of magnesium screw transfers the highest stress. Zain et al. [12] studied the stress distribution of orthopedic implants screwed in the trabecular bone, which is a highly porous biological tissue. The results of the finite element analysis revealed that the screw head yielded as the porosity of trabecular bone.

In the present study, to confirm the tensile strength of titanium orthopedic screws, the finite element method was used, and the results were confirmed by empirical tests. The femur bone is the bone section of the human thigh with similar properties of the bone section of cattle thigh; thus, the bovine femur bone and titanium screw bone Ti6Al4V have been used in the present paper. The behavior of the screw in the tensile test is discussed in terms of stress, force, and displacement.

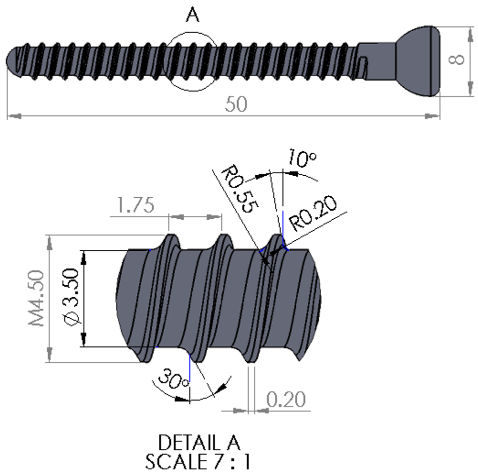
2. Materials and Methods

2.1. Experimental Works

A bone screw with a diameter of 4.5 mm, a length of 50 mm, and the pitch length of 1.75 mm are the geometric dimensions of the cortical screw used in this study. The shape of the screw, the teeth profile, and its details are presented in Fig. (1). As can be seen, the angles of the screw thread on the front and back of thread are equal to 30 and 10 degrees, respectively. The radius of the thread in the front and the rear parts are equal to 0.55 and 0.20 mm, respectively.

The geometric dimensions of the bone are exactly in accordance with the screw and without any leaping. In order to make a thread in the bone, firstly, a bore with a hole diameter of 3 mm was produced by the drilling process in the middle part of the prepared bone in a way that both upper and lower parts of the bone could be drilled in the same direction. Then, the screw was fastened in both holes. As shown in Fig. (2b), the screw is bumped into the bone at the lower part by 4 mm in addition to being passed from the top. In order to appropriately clamp during the tensile test, the additional part was designed and fabricated, as shown in Fig. (2c). The entire experimental setup is given in Fig. (2a). The bone width is equal to 10 mm, the position of the screw inside the bone is exactly in the middle of the bone, and the screw is 90 degrees towards the bone surface.

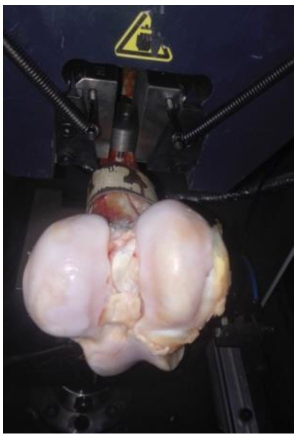
a



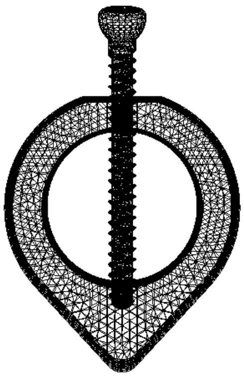
b

Figure 1. a) The shape, teeth profile and dimensional details of the screw and b) cortical screw.

a



b



c

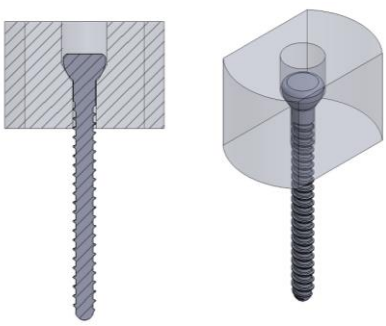


Figure 2. a) Tensile setup, b) Fastened screw and c) Holder of the screw cap.

2.2. Material Properties and Finite Element Model

The bovine femur bone was used in the present study due to possessing similar properties in terms of internal structures and material properties with human femur bone [13]. It should be noted that the freshness of bone makes the mechanical and thermophysical properties and density of bone tissue to be near to the living tissue under surgery. The screw is composed of Ti6Al4V, the characteristics of titanium screws, and femur bone, as shown in Table [13]. The friction coefficient between screw and bone teeth is [14]. The properties of bone and screw materials are considered as homogeneous and isotropic.

Table 1. Mechanical properties of bone and screw.

Human bone	Young Modulus (Mpa)	10-17
	Poisson's Ratio	0.4
	Density (kg/m^3)	1800-2000
	Tensile Strength (Mpa)	130-200
Bovine bone	Young Modulus (Mpa)	10-22
	Poisson's Ratio	0.33
	Density (kg/m^3)	1950-2100
	Tensile Strength (Mpa)	140-250
Screw	Young Modulus (Mpa)	110000
	Poisson's Ratio	0.33

The commercial finite element software ABAQUS was used considering a three-dimensional FE model for bone and screw. The mesh type for both bone and screw parts were selected to be the same as C3D10 (A 10-node quadratic tetrahedron). As depicted in Fig. (3), the bone was fully constrained in combined rotational and translational displacements, and then the screw was pulled out with the speed of 0.05 mm/s. The number of elements in the screw and bone was approximately equal to 20,772 and 23,248 elements, respectively, and stress, force, and displacement were the output results of the finite element tensile analysis.

3. Results and Discussion

Stress contour was depicted for different areas of both screw and bone. Maximum stress values occurred on the upper parts of the screw and bone (Fig. (4a)). The threaded area effectively reduces the stress values of lower parts (Fig. (4b)). Then, the left out area for the screw and the upper threaded part of the bone is the critical zone. As demonstrated in Fig. (4a), the blue rectangle shows the stress rates of the screw that are similarly distributed among threads except the first and the last ones. Due to the connection to the stronger zones, the first and the last threads bear lower rates of stress. The edge of the first thread of the bone shows the maximum stress inside the red circle. The subjected place is the thinnest part of the internal thread; thus, the related stress increased. The lower fastened zone was presented in Fig. (4b). Because of the thread discontinuity, the maximum stress of the screw occurred at the screw tip. Similar to the upper threaded part, the maximum rate of stress for bone placed at the top.

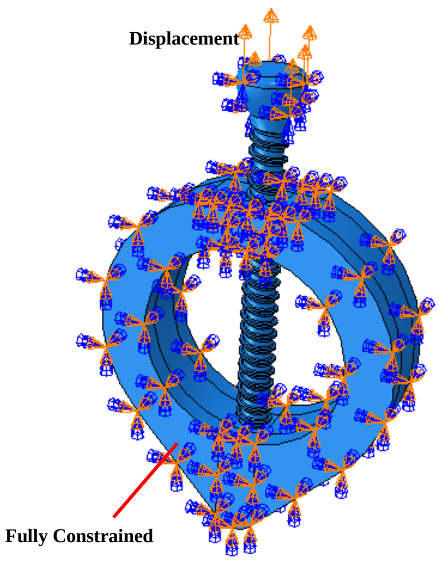


Figure 3. Loading and boundary conditions.

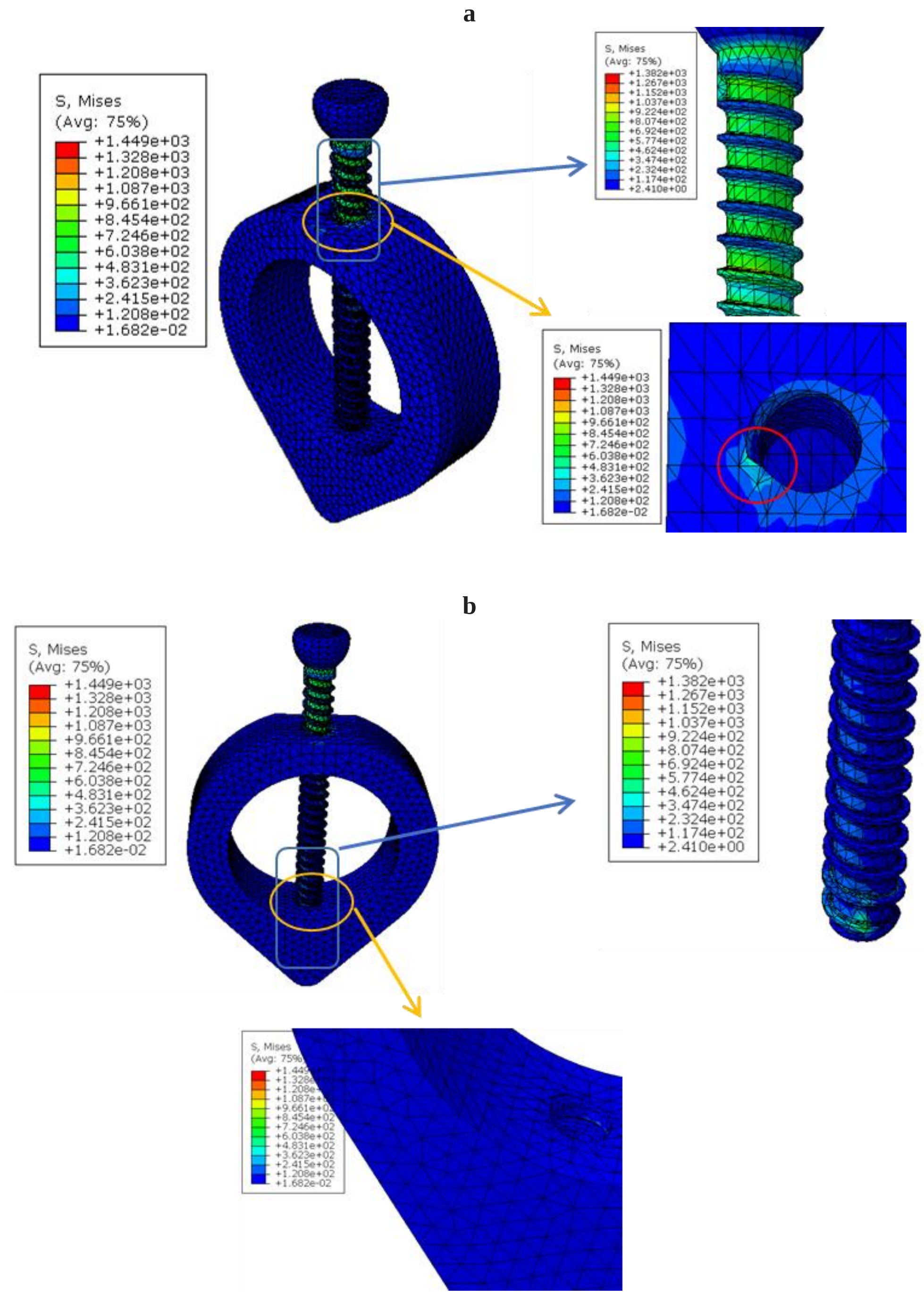


Figure 4. Stress contour on the screw and bone, a) Upper, and b) Lower bolted zone.

The stress values for the threaded bone part are presented in Fig. (5). The stress values are higher while moving toward the top surface of the bone. The main reason for the subjected occurrence decreases the threaded zone resistance against more deformation (Fig. (6)). It is evident that stress values are different in the right and left sides of the sectioned bone, which is higher at the right side. The main reason is that the tip of the threaded zone leads to a higher stress concentration on the right side. The displacement profile for both screw and bone was drawn in Fig. (7a). The upper part of the screw moved more than other areas. Moreover, it can be inferred from the comparison of two threaded parts that the upper threaded zone moves more (Fig. 7a&b).

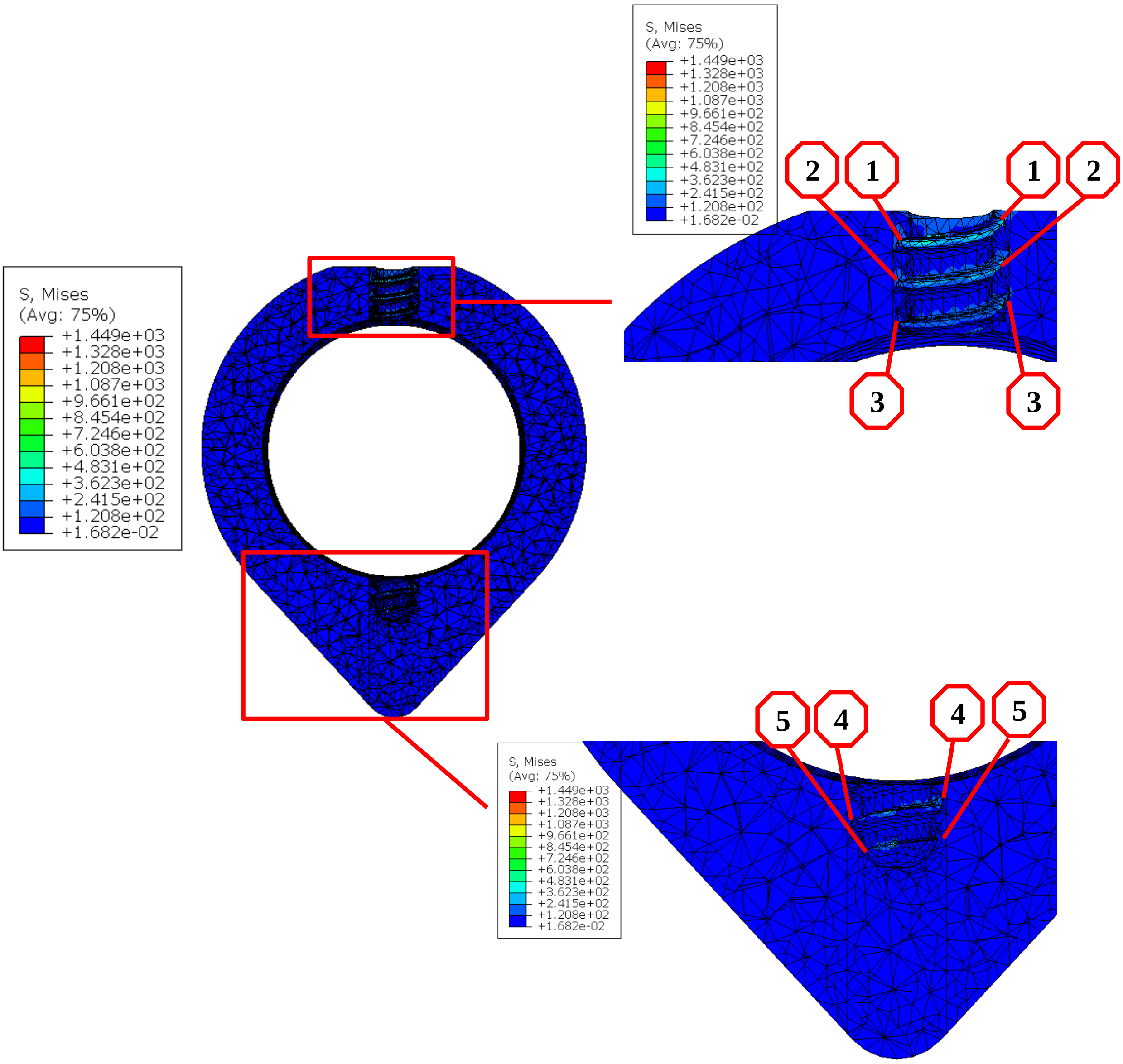


Figure 5. Stress contour of the bone thread.

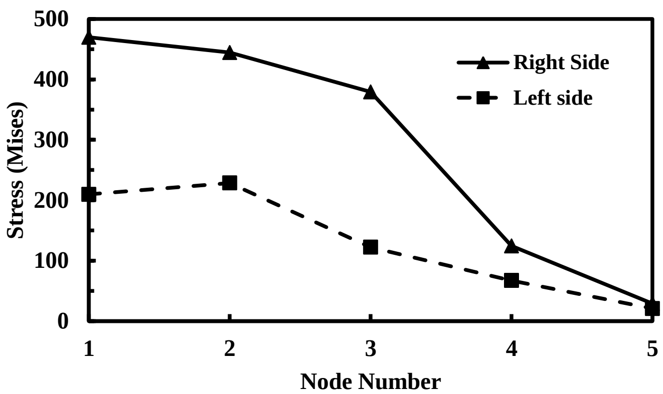


Figure 6. Stress values in the different specified node.

a

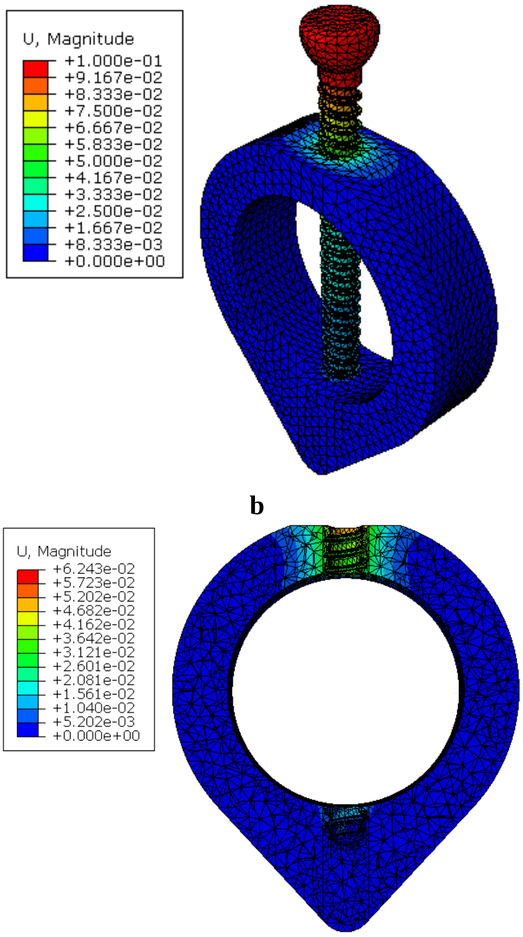


Figure 7. Displacement for (a) different assembled zones, and (b) bone cross-section.

Also, force variation curves were extracted for both numerical simulation and experimental tensile tests (Fig. (8)). The trend of both curves is similar, and the difference between curves decreases while coming toward the pick parts. It can be said that the difference between curves is the structural inhomogeneity inside the bone part. The existence of tiny fractures inside the bone and also some small differences in different bone types are the main reasons. The bone strength is higher in younger bovines. The maximum force difference between FEM and the experimental studies is equal to 14%. The maximum predicted tensile forces are 264N and 225N in FEM and the experimental works, respectively.

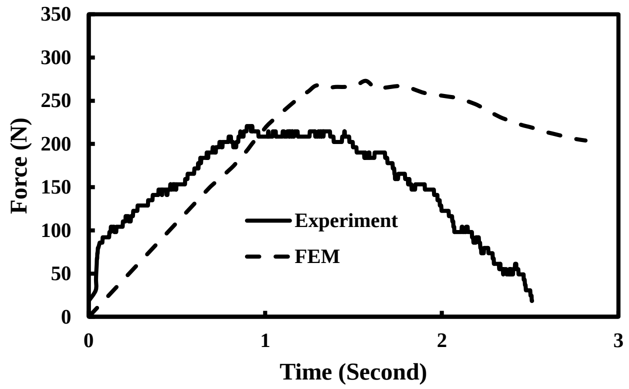


Figure 8. The force curve during tensile test obtained from both numerical simulation and experimental test.

4. Conclusion

The study was performed to determine the pulling-out strength of bone fastened by an orthopedic screw. The research was made both numerically by FEM and experimentally by the tensile test machine. The obtained results were satisfactory with the explanation that although there were higher differences between simulation and experiment in the first and last part of the test, the subjected difference decreases to 14% while approaching the maximum values of forces. Since the maximum strength of the bone against pull0out force is the objective of the present work and the force curve increases in the middle periods of the test, the force at the middle period is more important, and as mentioned earlier, the differences between simulation and experiment are within the acceptable range. Furthermore, it was concluded that the stress values are higher while moving toward the top surface of the bone. The main reason for the subjected occurrence is the reduction of the threaded zone resistance against higher deformation.

References

[1] J.-D. Kim, N.-S. Kim, C.-S. Hong, C.-Y. Oh, Design optimization of a xenogeneic bone plate and screws using the Taguchi and finite element methods, *International Journal of Precision Engineering and Manufacturing*, Vol. 12, No. 6, pp. 1119-1124, 2011.

[2] S.-M. Hou, C.-C. Hsu, J.-L. Wang, C.-K. Chao, J. Lin, Mechanical tests and finite element models for bone holding power of tibial locking screws, *Clinical Biomechanics*, Vol. 19, No. 7, pp. 738-745, 2004.

[3] C.-K. Chao, C.-C. Hsu, J.-L. Wang, J. Lin, Increasing bending strength of tibial locking screws: mechanical tests and finite element analyses, *Clinical Biomechanics*, Vol. 22, No. 1, pp. 59-66, 2007.

[4] K. Haase, G. Rouhi, Prediction of stress shielding around an orthopedic screw: Using stress and strain energy density as mechanical stimuli, *Computers in Biology and Medicine*, Vol. 43, No. 11, pp. 1748-1757, 2013.

[5] T. Wu, H. Fan, R. Ma, H. Chen, Z. Li, H. Yu, Effect of lubricant on the reliability of dental implant abutment screw joint: an in vitro laboratory and three-dimension finite element analysis, *Materials Science and Engineering: C*, Vol. 75, pp. 297-304, 2017.

[6] H. Ketata, F. Affes, M. Kharrat, M. Dammak, A comparative study of tapped and untapped pilot holes for bicortical orthopedic screws–3D finite element analysis with an experimental test, *Biomedical Engineering/Biomedizinische Technik*, Vol. 64, No. 5, pp. 563-570, 2019.

[7] J. R. Mau, K. M. Hawkins, S. L.-Y. Woo, K. E. Kim, M. B. McCullough, Design of a new magnesium-based anterior cruciate ligament interference screw using finite element analysis, *Journal of Orthopaedic Translation*, Vol. 20, pp. 25-30, 2020.

[8] Y. Naidubabu, V. Kondaiah, R. Dumpala, B. R. Sunil, *Assessing the Material-Dependent Stress Distribution in Fractured Bone and Orthopedic Fixing Plate by Finite Element Analysis*, in: *Advances in Materials and Manufacturing Engineering*, Eds., pp. 337-342: Springer, 2020.

[9] L. Yan, J. L. Lim, J. W. Lee, C. S. H. Tia, G. K. O'Neill, D. Y. Chong, Finite element analysis of bone and implant stresses for customized 3D-printed orthopaedic implants in

fracture fixation, *Medical & Biological Engineering & Computing*, pp. 1-11, 2020.

[10] J. Li, Z. Zhao, P. Yin, L. Zhang, P. Tang, Comparison of three different internal fixation implants in treatment of femoral neck fracture—a finite element analysis, *Journal of orthopaedic surgery and research,* Vol. 14, No. 1, pp. 76, 2019.

[11] E. Tetteh, M. B. McCullough, Impact of screw thread shape on stress transfer in bone: a finite element study, *Computer Methods in Biomechanics and Biomedical Engineering*, pp. 1-6, 2020.

[12] N. Zain, R. Daud, N. Aziz, K. Ahmad, A. Ismail, B. Izzawati, Stress analysis prediction on screw orthopedic implant in trabecular bone, *Materials Today: Proceedings,* Vol. 16, pp. 1838-1845, 2019.

[13] K. Alam, A. Mitrofanov, V. V. Silberschmidt, Experimental investigations of forces and torque in conventional and ultrasonically-assisted drilling of cortical bone, *Medical engineering & physics,* Vol. 33, No. 2, pp. 234-239, 2011.

[14] S. Eberle, C. Gerber, G. Von Oldenburg, S. Hungerer, P. Augat, Type of hip fracture determines load share in intramedullary osteosynthesis, *Clinical Orthopaedics and Related Research®,* Vol. 467, No. 8, pp. 1972-1980, 2009.