International Journal of Natural and Engineering Sciences 10 (1): 36-41, 2016
ISSN: 1307-1149, E-ISSN: 2146-0086, www.nobel.gen.tr

Stress-Strain Distribution at Bone-Implant Interface Using 3D Finite Element Analysis

Durmuş Ali BIRCAN*
Çukurova University, Mechanical Engineering Depth. 01330 Balcalı, Adana, Turkey

*Corresponding Author:
E-mail:abircan@cu.edu.tr

Received: March 03, 2016
Accepted: May 26, 2016

Abstract

Dental implant surgery is a rational alternative in most clinical operations. The complex biomechanical aspects of a tooth-implant system are derived from the mobility between the osseointegrated implant and the tooth. The attachment of the abutment to the implant, the loosening of the fixation screw, and its fracture may arise from a poor distribution of occlusal loads in the bone-implant interface. Analyzing force transfer at the bone-implant interface determines the success or failure of an implant. Implant features causing overload can cause bone resorption or fatigue failure of the implant whilst underload may lead to disuse atrophy and to subsequent bone loss. The success of a dental implant depends on a variety of biomechanical factors including the design and position of the implant, implant-abutment connection, length and diameter of implants, implant shape, surface roughness, bone quality and type, depth of insertion, arch configuration, the nature of bone implant interface, and occlusal conditions. The Finite Element Method (FEM) has been applied to the prognosis of stress distribution in both the implant and its interface with the adjacent bone for a comparison of several geometries and applied loads. The study of stresses using the FEM is a virtual simulation of three-dimensional mathematical models in which all biological and material structures involved can be discretized, subdivided into smaller structures. The aim of this study was to evaluate the performance of the bone-implant system during the implantation process and the healing and maintenance phases of osseointegration for various implant thread geometries with different implant materials under fixed forces using 3D Finite Element Analysis (FEA) technique in customized mandible.

Keywords: Computer Aided Design, Implant, Finite Element Analysis, FEA, Mandible, Biomechanics

INTRODUCTION

The implantation operation requires an optimum stress distribution in order to maintain a strong and healthy bone: a stress that is too high may cause irreversible damage to the mandible; one that is too low may fail to stimulate the mandible sufficiently for satisfactory healing and thus, for osseointegration. Elevated compressive stress during implant insertion can block blood supply and damage the cells, particularly for dense bone would affect osseointegration. On the other hand, inadequate stress cannot stimulate an optimal degree of bone remodelling, thus reducing long term implant stability [1]. Without the correct implantation technique an implant may fail shortly after insertion. The method by which the implant is placed into the jawbone plays a critical role for it to be biologically accepted by the surrounding bone. An optimal stress level and distribution must be chased during and after implantation for the bone to remain strong and healthy, and to prevent bone resorption.

The major factors influencing an optimal implant design are the stress distribution within the bone, and implant stability which then favours osseointegration. Load transfer is influenced by several factors as loading type, length and diameter of the implant, implant thread form, structure of the implant surface, bone quality and quantity, and implant material have all been shown to have effects on the stress level and distribution on the bone [2]. It is essential for the dentists to have a thorough understanding of the stress characteristics within the bone to guarantee a successful implantation.

An in-depth understanding of stress profiles encountered by the implant, and more importantly in the surrounding bone can be gained through effective use of FEM. It is prominent that the results obtained through the FEA can only be used with confidence once it is validated by in vivo and/or in-vitro experiments. The increase in knowledge of stress distributions and magnitudes within the implant and surrounding bone will aid the optimisation of the implant design and material.

In the previous studies, many modellings and analysis have been made using FEA methods [3-5]. The stress distribution that occurs in the mandible was investigated for different implant geometries [6]. The effect of implant diameter and implant length on the stress distribution was examined [7]. The effect of different abutment designs on the implant systems was also studied [8]. However, rectangular structures have been considered as the mandible generally instead of the specific mandible model in the majority of these studies.

The implant diameter and length have a significant influence on the stress distribution within the bone (Huang et al., 2007 [9]; Ivanoff et al., 1999 [10]; Iplikçioğlu and Akça, 2002 [11]; Pierrisnard et al., 2003 [12]; Akça et al., 2003 [13]; Kong et al., 2006-2008 [14-15]; Sun, 2007 [16]; Eskitascioglu et al., 2004 [17]). FEM can be used to determine the optimum diameter and length that would best dissipate stresses induced by the implantation. Himmlova et al. (2004) [18] computed values of von Mises stress at the bone/implant interface for a number of variations in diameter and length. Maximum stress areas were identified to be located around the implant neck.

In this study, different implant thread forms are designed on the mandible model. In the end of the study, dentist will decide which type implant design is optimum for the patient based on bone density in patient's mandible. To prefer optimum thread design will increase the percentage of long-term success of the implant. In this study, 14,25 mm in length and 4.3 mm diameter three different 'square', 'buttress' and 'round' thread forms of implants with three different materials as titanium, zirconium and cadmium were inserted on the customized mandible model and were analyzed using FEM.

METHODOLOGY

This study was accomplished in three stages as: creating 3D solid model of the mandible, design of the implant and abutment models, and creating finite element models. Then, the analyses were done for the process of load transfer and stress distribution. Firstly, CT images of a patient's mandible will be ripped and processed in a special programme by the method of Reverse Engineering (RE). Thirty five years old male patient's mandible was scanned using a computed tomography device. The resulting data stored in the DICOM format that is medical image format of Siemens and has been transferred to the medical image processing program [19]. Scanned 2D DICOM files are converted into two different point cloud formats as cancellous and cortical bone layers by means of Mimics program. 3D models in point cloud format are not appropriate mathematical models to be used in FEA method [20]. So, the resulting two different 3D point cloud model are transferred into 3D CAD program CATIA and converted into the 3D solid model. Obtained cancellous and cortical bone layers in the form of solid models are combined and 3D model of the mandible has been obtained as can be seen in Fig. 1.

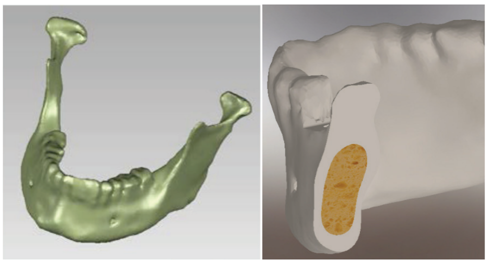

Figure 1. Solid model of mandible

After creating the mandible model, implants with a diameter of 4.3 mm and an overall length of 14.25 mm were modelled by using CAD software. All implant forms are set as 0.8 mm pitch and 0.6 mm depth of thread. These values are determined by the commercially used dental implants [21]. Full osseointegration is considered between implant and bone. Implants which have square, buttress (Fig.2) and round thread forms with three different materials as titanium, zirconium and cadmium were inserted on the customized mandible model and were analyzed using FEM.

Created geometric models are transferred to the ANSYS program for FEA. Selection of element type on the mathematical model, creating the mesh form, determining the contact areas, boundary conditions, environment and material properties and the type of analysis have been made in the program interface.

Tetrahedron element is used as shown in Fig. 3. After the meshing operation, 235,175 points and 139,420 elements were obtained for nine different models. Some physical properties are submitted for each design combination in the model for the analysis after meshing process. These properties are modulus of elasticity (E) and Poisson's ratio (γ) which are the mechanical properties of materials.

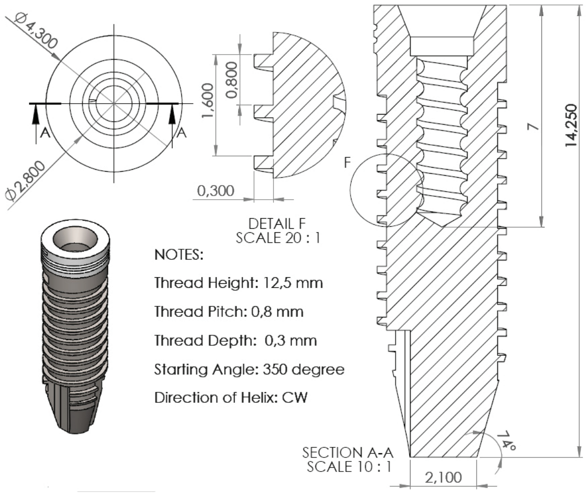

Figure 2. Buttress Thread Implant

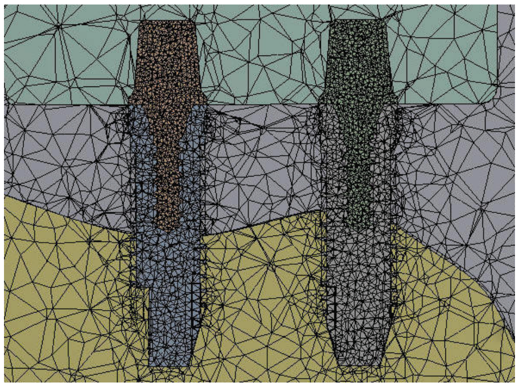

Figure 3. Meshing of implant, abutment and mandible

Bone is divided into two layers, cortical and cancellous, depending on the density of the bone. Cancellous bone layer located under a layer of cortical bone is a porous structure, less dense and less rigid than the cortical layer [22]. Hence, the two bone layers have different mechanical properties. In this study, modulus of elasticity was taken as 15 GPa for the cortical bone and 1 GPa for the cancellous bone. Poisson's ratio is taken as 0.3 for the cortical and cancellous bone. Titanium, zirconium and cadmium have been selected as three different materials for the implants and abutments. Porcelain is used as teeth material (Table 1).

Table 1. Mechanical properties of the materials used in this study

Material	Modulus of Elasticity (GPa)	Poisson Ratio
Cortical bone	15	0,3
Cancellous bone	1	0,3
Titanium	110	0,3
Zirconium	210	0,25
Cadmium	50	0,3
Porcelain	70	0,19

An important criterion for simulating a realistic implantation process is the loading and boundary conditions. When determining the boundary conditions, ramus section of the lower mandible remains stationary in x, y and z directions, all the elements of the model in this region are given zero degrees of freedom. Two different occlusal forces, 150 N in the horizontal direction and of 300 N in the oblique direction that is the resultant of vertical forces, were applied to the created model as the parallel to the long axis of the implant. The magnitude of the forces was given by considering the maximum masticatory forces in the mouth [23].

RESULTS and DISCUSSIONS

Most of the studies regarding to FEA method was reported that the Von Mises stress criterion which calculates numerically the stress condition is sufficient [24]. In this study, the solutions were compared according to this energy criterion.

During implant insertion, the von Mises stresses are detailed under increased torque to offer a better understanding of the stresses experienced by specific regions of the mandible where a stress concentration occurs. As the implant is 14.25 mm long and 4.3 mm diameter, there are nine different models to be analyzed for three different thread forms and implant materials. When the implantation process is initiated the implant is assumed to be inserted into the cortical bone so replicating the implant tip being pushed slightly into the top surface of the bone prior to the application of any torque. As the depth of the implant into the mandible increases, bone interface increases which drastically alters the von Mises stresses within the mandible (Fig.4-6). As the implant is being screwed into the mandible, the surface area contact between the implant and surrounding bone increases leading to a higher degree of resistance to the implant entering the mandible. The von Mises stresses recorded at locations on cancellous and cortical bones are plotted the along the implant length indicated in the figures. The corresponding stress characteristics are described in Fig. 7-9.

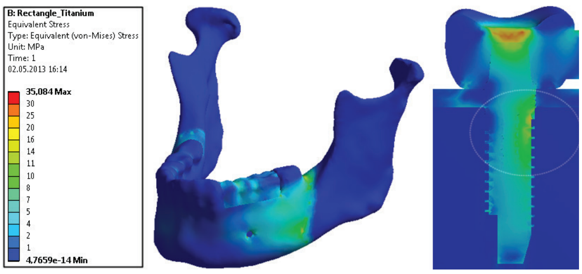

Figure 4. Stress distribution on the bone-implant interface for square thread form

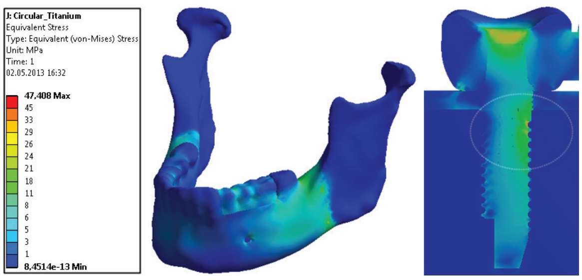

Figure 5. Stress distribution on the bone-implant interface for round thread form

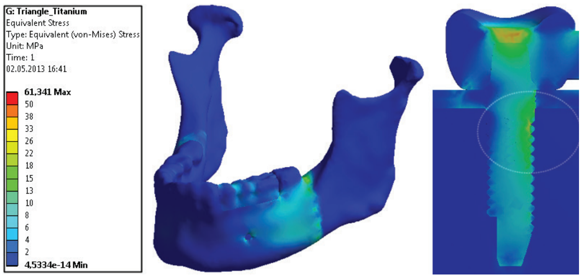

Figure 6. Stress distribution on the bone-implant interface for buttress thread form

At the start of the thread, the stress magnitude is relatively low because there is no direct contact between the implant and cancellous bone. The bottom thread comes into direct contact with the cancellous bone leading to a sudden increase in the stress. A gradual increase in stress is due to the increase in implant surface area that is in contact with the cortical bone. The stress increases noticeably and more rapidly because the cancellous bone is in contact with the bottom threads. However the cortical bone comes into contact with the interface hence increasing the stress within the cancellous bone. More surface area contact between the implant and cancellous bone leads to a gradual decrease in the stress level for each stage of insertion. An increased surface area contact between the implant and cancellous bone are responsible for the increase in stress levels. The increase in stress is due to the implant neck being in direct contact with the cortical bone. Finally, a further decrease in stress is observed at the last points of implant length because the cortical bone absorbs a higher stress. A decrease in stress from 12 mm to 14 mm of implant length is due to the fact that more implant surface area is in direct contact with cancellous bone.

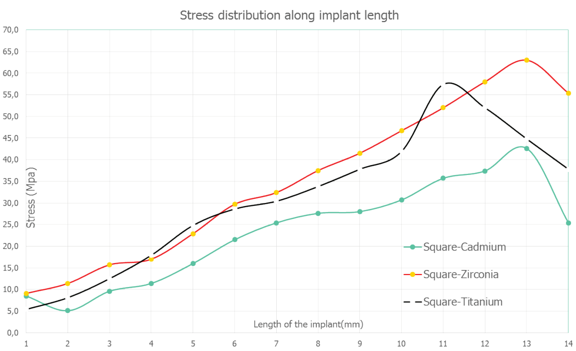

Figure 7. Stress distribution along the implant length for square thread form and materials

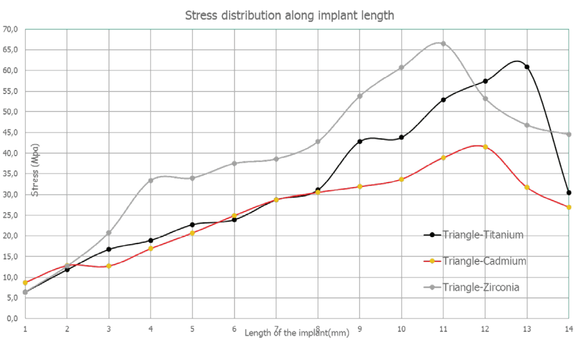

Figure 8. Stress distribution along the implant length for buttress (triangle) thread form and materials

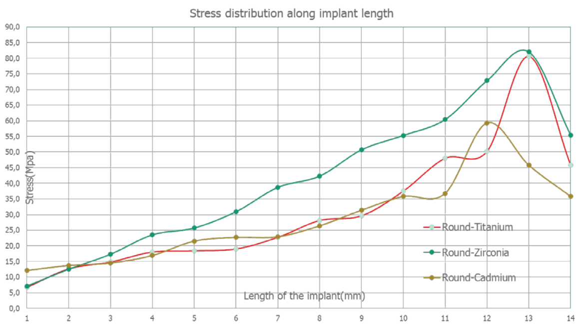

Figure 9. Stress distribution along the implant length for round thread form and materials

The results were summarized that; maximum stress value is observed on the cortical bone surface, especially first contacted side with the implant, implant geometry with 'square' thread type causes least stress. There is not a significant difference between the stress values created on the bone by the implants which have the same geometry but different materials, and stress accumulation is occurred on the hills of thread of used all of dental implants.

CONCLUSIONS

With rapid improvements and developments of computer technology, the FEA has become a powerful technique in dental implant biomechanics because of its versatility in calculating stress distributions within complex structures. By understanding the basic theory, method, application, and limitations of FEA in implant dentistry, the clinician will be better equipped to interpret results of FEA studies and extrapolate these results to clinical situations. Thus, it is a helpful tool to evaluate the influence of model parameter variations once a basic model is correctly defined. Additional research should be centeedr in analyzing stress distributions under different loading conditions of mastication, which would better mimic the actual clinical situation.

In this study, thread forms and materials of implants are taken as variable parameters and to compare these parameters using 3D finite element stress analysis method is used. Stress distribution on the cortical, cancellous bone and implant is analyzed. In this way optimal implant system is determined for patient who will be performed dental implant treatment

The main advantage of performed computer simulations is that it is fast, efficient and cheap. A comparison of the estimated stress concentrated regions with simulated stress concentrated regions data displayed the accuracy and reliability of the computer simulation method. The proposed method for estimating stress behavior can be applied to other dental implant and crown bridge designs to improve design process. New implants can be designed and justified in computer simulation using FEA before they are produced to save time and money.

Acknowledgement

This study is supported by Çukurova University Research Fund [FED-2016-7131].

REFERENCES

[1] Misch C.E., 2007. Dental Implant Prosthetics. 3rd ed. Philadelphia.

[2] Holmgren E.P., Seckinger R.J., Kilgren L.M., Mante F., 1998. Evaluating parameters of osseointegrated dental implants using finite element analysis-a two – dimensional comparative study examining the effects of implant diameter, implant shape, and load direction. Journal of Oral Implantology. 24(2):80–88.

[3] Li T., Kong L., Wang Y., Hu K., Song L., Liu B., Li D., Shao J., Ding Y., 2009. Selection of optimal dental implant diameter and length in type IV bone: a three-dimensional finite element analysis. International Journal of Oral & Maxillofacial Surgery. 38: 1077-1083.

[4] Borie E., Orsi A.A., De Araujo C.P.R. 2015. The influence of the connection, length and diameter of an implant on bone biomechanics. Acta Odontologica Scandinavica. 73: 321–329

[5] Tada S., Stegariou R., Kitamura E., Miyakawa O., Kusakari H., 2003. Influence of implant design and bone quality on stress/strain distribution in bone around implants: a 3-dimensional finite element analysis. The International Journal of Oral & Maxillofacial Implants. 18(3):357–368.

[6] Franciosa P., Martorelli M., 2012. Stress-based performance comparison of dental implants by finite element analysis. International Journal on Interactive Design and Manufacturing. 6, 2: 123–129.

[7] Baggi L., Cappelloni I., Di Girolamo M., Maceri F., Vairo G., 2008. The influence of implant diameter and length on stress distribution of osseointegrated implants related to crestal bone geometry: A three-dimensional finite element analysis. Journal of Prosthetic Dentistry. 100:422–431.

[8] Hasan I., Röger B., Heinemann F., Keilig L., Bourauel C., 2012. Influence of abutment design on the success of immediately loaded dental implants: Experimental and numerical studies. Medical Engineering & Physics. 34, 7:817–825.

[9] Huang H.L., Chang C.H., Hsu J.T., Fallgatter A.M., Ko C.C., 2007. Comparison of implant body designs and threaded designs of dental implants: a 3-dimensional finite element analysis. The International Journal of Oral & Maxillofacial Implants. 22(4):551–562.

[10] Ivanoff C.J., Sennerby L., Johansson C., Rangert B., Lekholm U., 1997. Influence of implant diameters on the integration of screw implants. An experimental study in rabbits. International Journal of Oral & Maxillofacial Surgery. 26(2):141–148.

[11] Akca K., Iplikcioğlu H., 2002. Finite element stress analysis of the effect of short implant usage in place of cantilever extensions in mandibular posterior edentulism. Journal of Oral Rehabilitation. 29(4):350–356.

[12] Pierrisnard L., Hure G., Barquins M., Chappard D. 2002. Two Dental Implants Designed for Immediate Loading: A Finite Element Analysis. The International Journal of Oral & Maxillofacial Implants.17:353–362.

[13] Akca K., Cehreli M.C., Iplikcioğlu H., 2003. Evaluation of the mechanical characteristics of the implant abutment complex of a reduced-diameter morse-taper implant. A nonlinear finite element stress analysis. Clinical Oral Implants Research. 14(4):444–454.

[14] Kong L., Liu B., Li D., Song Y., Zhang A., Dang F., Qin X., Yang J., 2006. Comparative study of 12 thread shapes of dental implant designs: a three-dimensional finite element analysis. World Journal of Modelling and Simulation. 2, 2, 134-140.

[15] Kong L., Hu K., Li D., Song Y., Yang J., Wu Z., Liu B., 2008. Evaluation of the Cylinder Implant Thread Height and Width: A 3-dimensional Finite Element Analysis.

The International Journal of Oral & Maxillofacial Implants. 23:65–74.

[16] Sun Y., Kong L., Liu B., Song L., Yang S., Wei T., 2007. Comparative study of single-thread, double-thread, and triple-thread dental implant: a three-dimensional finite element analysis. World Journal of Modelling and Simulation. 3, 4: 310-314.

[17] Eskitascioglu G., Usumez A., Sevimay M., Soykan E., Unsal E., 2004. The influence of occlusal loading location on stresses transferred to implant-supported prostheses and supporting bone: A three dimensional finite element study. Journal of Prosthetic Dentistry. 91(2):144–150.

[18] Himmlova L., Dostalova T., Kacovsky A., Konvickova S., 2004. Influence of implant length and diameter on stress distribution: A finite element analysis. Journal of Prosthetic Dentistry. 91: 20-25.

[19] Bircan D.A., Dede D., Ekşi A.K., 2016. Development of the customised implant system using CAD/CAM/CAE tools. Advances in Materials and Processing Technologies. 2, 1: 57–65.

[20] Dilek M., Bircan D.A., Ekşi A.K., 2012. Construction of 3D finite element model of human mandible for biomechanical analyses. 2nd International Scientific Conference on Engineering Manufacturing and Advanced Technologies, MAT 2012; 2012 November 22–24; Antalya, Turkey.

[21] Nobel Biocare Product Catalog, 2011.

[22] Kurniawan D., Nor F. M., Lee H. Y., Lim Y., 2012. Finite element analysis of bone implant biomechanics: refinement through featuring various osseointegration conditions. International Journal of Oral & Maxillofacial Surgery. Sep, 41, 9:1090–1096.

[23] Richter E.J., 1989. Basic biomechanics of dental implants in prosthetic dentistry. Journal of Prosthetic Dentistry. 61:602–609.

[24] Stegaroiu R., Kusakari H., Nishiyama S., 1998. Influence of prosthesis material on stress distribution in bone and implant: a three dimensional finite element analysis. The International Journal of Oral & Maxillofacial Implants. 13:781–790.

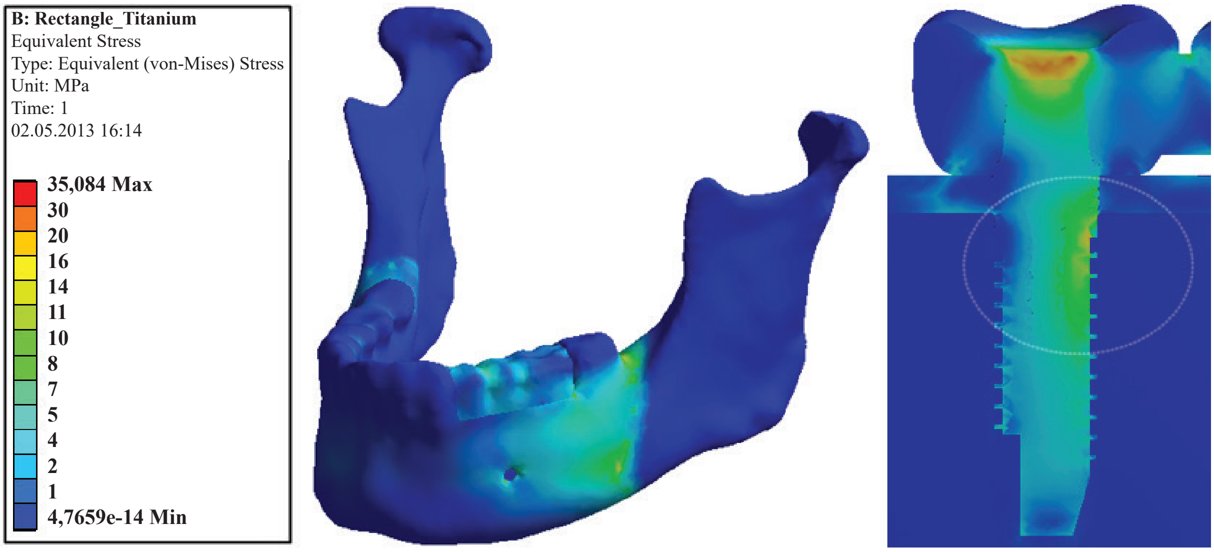

Figure 4. Stress distribution on the bone-implant interface for square thread form

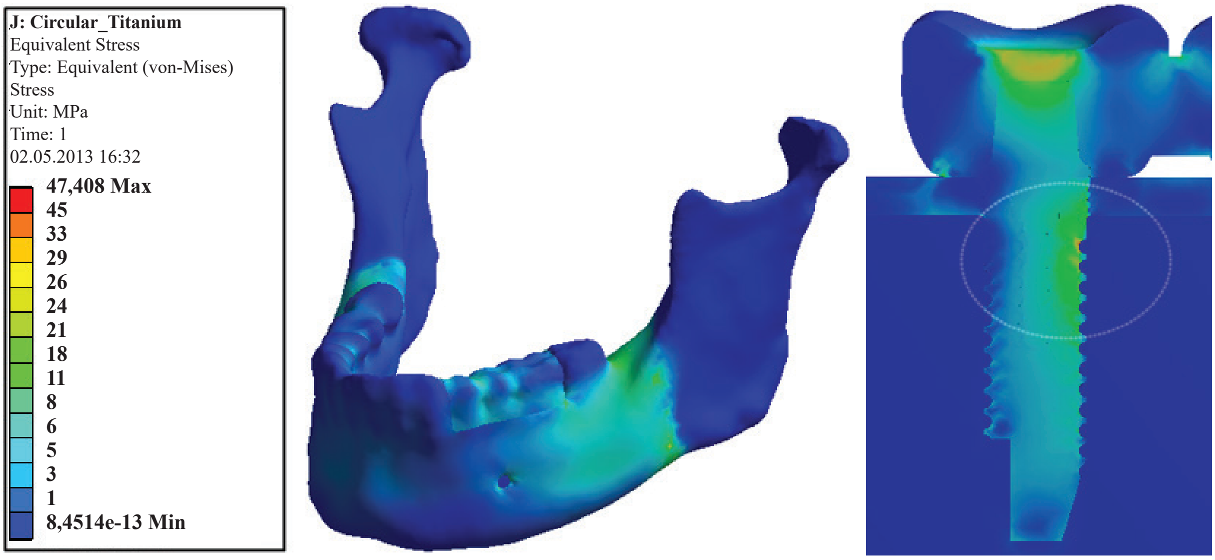

Figure 5. Stress distribution on the bone-implant interface for round thread form

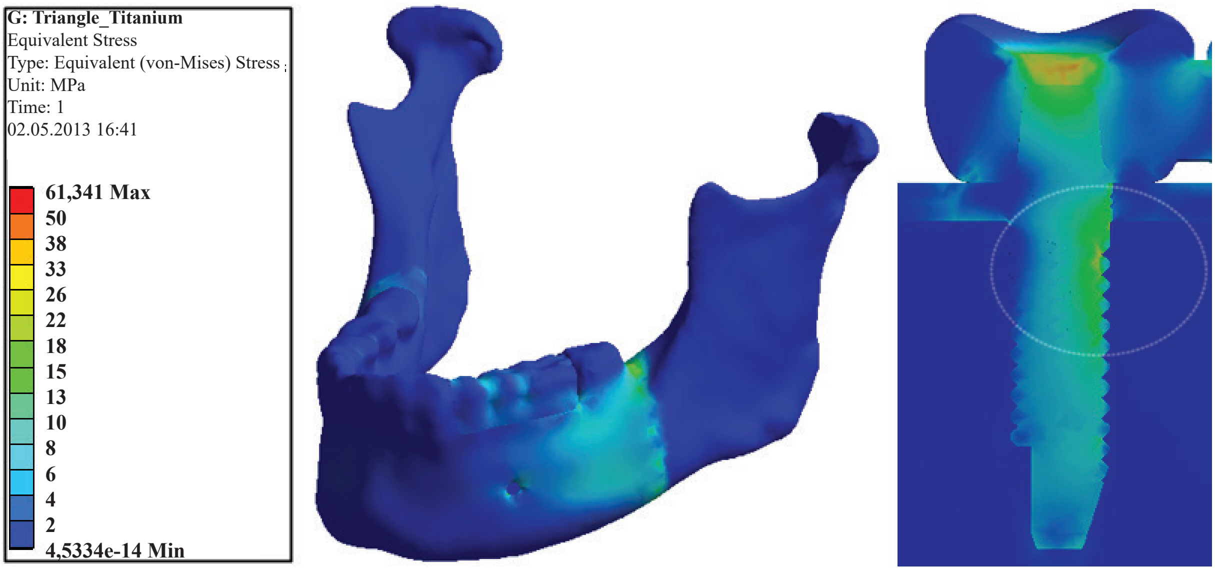

Figure 6. Stress distribution on the bone-implant interface for buttress thread form