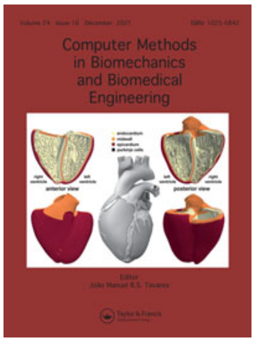

Computer Methods in Biomechanics and Biomedical Engineering

ISSN: (Print) (Online) Journal homepage: https://www.tandfonline.com/loi/gcmb20

Bone stress and damage distributions during dental implant insertion: a novel dynamic FEM analysis

Ahmet Emin Demirbas, Recep Ekici, Mustafa Karakaya & Alper Alkan

COMPUTER METHODS IN BIOMECHANICS AND BIOMEDICAL ENGINEERING
https://doi.org/10.1080/10255842.2021.2012765

Bone stress and damage distributions during dental implant insertion: a novel dynamic FEM analysis

Ahmet Emin Demirbas[a], Recep Ekici[b], Mustafa Karakaya[c] and Alper Alkan[d]

[a]Department of Oral and Maxillofacial Surgery, Faculty of Dentistry, Erciyes University, Kayseri, Turkey; [b]Department of Mechanical Engineering, Faculty of Engineering, Erciyes University, Kayseri, Turkey; [c]Depratment of Oral and Maxillofacial Surgery, Sultanbeyli Oral and Dental Health Hospital, Ministry of Health, İstanbul, Turkey; [d]Department of Oral and Maxillofacial Surgery, Faculty of Dentistry, Bezmialem Vakif University, İstanbul, Turkey

ABSTRACT
The objective of this research was to evaluate the stress and damage occurring on the bone model of D2 quality during implant insertion procedure using a novel dynamic finite element analysis (FEA) modeling. Three-dimensional finite element method was used to simulate the implant placement into the mandible. The cross-sectional model of the implant was created in SolidWorks 2007 software. The implant model was created to resemble a commercially available fine thread bone level dental implant (Bilimplant®, Turkey). 3 D bone models created with and without cortical bone drilling were specified according to D2 bone (Misch's Bone Classification) with a 1.5 mm cortical bone thickness. The stress patterns in both cancellous and cortical crestal bone were examined during implant insertion by using a novel dynamic FEA in ABACUS/Explicit (ABAQUS/Explicit version 6.14). According to the results of the dynamic FEA, it was reduced stress and damage significantly on the crestal bone region using the cortical drill before the implantation. Also, implant placement time was shorter when the cortical drill was used. The present research is a pilot study using a novel dynamic FEM to model and simulate the dental implant insertion process. This study showed that the use of cortical drills decreased the stress in the bone, especially crestal region, and shortened the whole implant insertion time.

ARTICLE HISTORY
Received 2 May 2020
Accepted 28 November 2021

KEYWORDS
Dental implant; finite element analysis (FEA); crestal bone stress; dynamic FEM analysis; cortical bone drill; implant insertion

Introduction

The rehabilitation of the edentulous jaws with the use of dental implants is a widespread and accepted treatment method that has well-documented and successful outcomes (Bozkaya et al. 2004; Schrotenboer et al. 2008). The biomechanics of dental implants play an important and major role in the long-term success of implant-supported prosthetic restorations (Hasan et al. 2014). The stress and strain occurring in the jawbone around the dental implant can be affected by some biomechanical factors such as quality and quantity of the jawbone, the loading type, macro and micro geometry of the implant, and surface properties of the implant (Staden et al. 2006). One of the main factors for the success of a dental implant is how the stresses are transferred to the surrounding jawbone (Geng et al. 2001). According to Wolff's theory, the bone's response to absorption or healing is directly related to stress in the bone (Wolff 1892). Also, according to Wolff's theory, bone remodeling is directly proportional to the forces acting on the bone (Wolff 1892; Monstaporn et al., 2020). These forces and stress especially cause crestal bone loss and are decisive in the success of the implant. The amount of bone loss in the neck area plays a role in determining the success of the implant (Wolff 1892; Ravishankar 2021; Monstaporn et al., 2020). Stresses around the dental implant can cause resorption of the jawbone. Especially, crestal bone loss around the dental implant can be observed in the short-term period after the implant insertion. Crestal bone loss occurs to a degree with all dental implant designs used clinically and it is mostly caused by excessive crestal bone stresses. This process results in the formation of a pocket around the neck region of the dental implant which can cause bacterial colonization and following tissue loss due to inflammation (Vaillancourt et al. 1995). The end of this process may result in a complete failure of the dental implant treatment. This bone resorption process which affects mainly the neck

CONTACT Ahmet Emin Demirbas aedemirbas@erciyes.edu.tr
Supplemental data for this article is available online at https://doi.org/10.1080/10255842.2021.2012765.

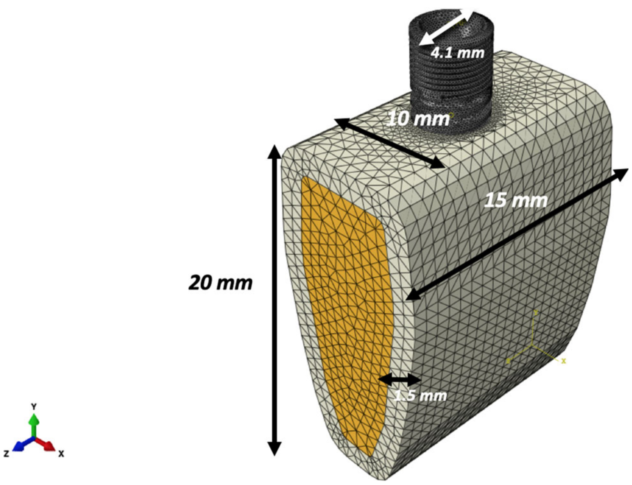

Figure 1. Details of the mandibular bone model.

region of the dental implant can be activated also by surgical trauma, bacterial infections, functional forces and overloading at the bone-implant interface (Baggi et al. 2008).

Finite element analysis (FEA) is a very popular and useful method for assessing stresses occurring on the implant and surrounding bone (Geng et al. 2001). The basic concept behind the FEA is to subdivide a body of any shape into simpler geometric shapes or elements (Rieger et al. 1989). The FEA is a computer-aided method used to estimate how a dental implant or bony structures react to some physical effects like force, vibration, and heat in the researches of the dental implant field (Vaillancourt et al. 1995). The most important advantage of the FEA is that it can non-invasively analyze many factors that can affect the success of a dental implant treatment (Gedrange et al. 2003). Since 1976, FEM analysis has been used in many studies in dentistry (Weinstein et al. 1976; Mohammed et al. 1979; Brunski 1992; Lewinstein et al. 1995). The biomechanics of the jawbone around the dental implants have been mainly analyzed with the use of the FEM in many studies in the literature (Meijer et al. 1994; DeTolla et al. 2000; Geng et al. 2001; Deck et al. 2004; Baggi et al. 2008; Staden et al. 2008; Hasan et al. 2014). However, there are very limited FEA studies that investigated the stress profile occurring in the crestal region of the jawbone during the implant insertion (Staden et al. 2008). During the implantation process, an ideal stress profile is required to protect the bone surrounding the dental implant. Excessive bone stresses occurring during the insertion can irreversibly damage the jawbone and cause implant failure. On the other hand, the insufficient stresses may fail to stimulate the jawbone for satisfactory wound healing and osseointegration (Staden et al. 2008). Therefore, it is important to control the stresses that occur during implant insertion for the long-term success of the dental implant.

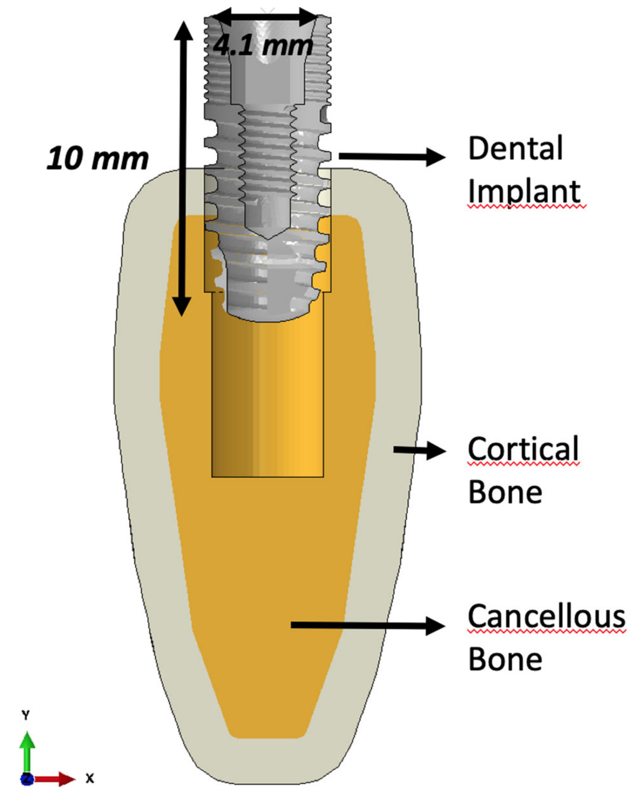

Figure 2. Details of the dental implant design and implantation site in the bone using cortical drill.

In this study, it is aimed to investigate the stress and damage distributions during implant insertion into the mandibular bone model of D2 quality in a time-dependent manner using a novel dynamic FEA modeling. For this purpose, the 3 D explicit (dynamic) non-linear finite element method was used for the simulation process. The cortical and cancellous bones were modeled as elastoplastic using Johnson-Cook plasticity and damage models. However, the dental implant was modeled as rigid. The dynamic analysis results were given comparatively in a time-dependent manner in terms of stress and damage.

Materials and methods

FEM modelling and model geometry

Three-dimensional (3 D) finite element models were built to analyze stress distributions of usage standard drill and cortical drill during the implant placement. FEM analyses of implantation process were performed by ABAQUS/Explicit (ABAQUS/Explicit version 6.14) software.

The computed tomography images of a healthy edentulous posterior human mandible were taken and

Table 1. Geometric properties of the implant and drills.

	Diameter (mm)	Length in the bone (mm)
Dental implant	4.1	10
Cortical drill	4.1	4
Standard drill	3.5	10

the bone contour was created with image processing software (ABAQUS, (ABAQUS/Explicit version 6.14)). A mandibular bone segment was extracted to be suitable size for this research, and to assume a D2 bone quality (Misch's Bone Classification), a 1.5 mm thickness cortical layer was formed along the buccal and lingual surface of the bone segment. Overall dimension of the posterior mandible bone segment was 20 mm vertical height, 15 mm mesiodistal width and 10 mm buccolingual width at ridge crest (Figure 1).

Platform switch-type implant was modelled using the design of 4.1*10 mm bone level dental implant (Bilimplant®, Turkey) (Figure 2). Two different drilling procedure were used in the preparation of the implant socket; the standard drilling and the standard drilling with the cortical bone drill. The cortical drill was modelled using the design of 4.1 mm diameter and 4 mm length and the standard drill was modelled using the design of 3.5 mm diameter and 10 mm length. Geometric properties of implant and cortical drill system are shown in Table 1.

The finite element models of the implant and jawbone are meshed using solid and shell elements, respectively. The bone model was meshed with a total number of 85,543 elements. The implant model has a total number of 31,384 elements. The element type used for the implant is R3D4, while the element type used for bone is C3D8R. In the study, a convergence test about mesh refinement was carried out to optimize solution accuracy and time. In order to more accurately estimate the local damage caused by the implant, a mesh refinement operation was performed in the drilling area. It was observed that the results obtained did not change more than 5% in the above element number. As a result, the best mesh convergence was obtained in the optimum element number above for this study.

The Poisson's ratio, the Young's modulus and density of the cortical bone and cancellous bone are listed in Table 2. The simulated elastic and plastic properties of the cancellous and cortical bone are listed in Table 2.

Johnson-Cook plasticity model was used to predict elasto-plastic behaviors of cortical and cancellous bones. According to Johnson-Cook plasticity model, the material behaves as linear elastic but when the yield stress is reached, after this point, the material deforms plastically. In addition, the ductile damage initiation criterion was used to determine the amount of damage during implant delivery (Deck et al. 2004). The Johnson-Cook plasticity model is given in Equation (1) below. This equation describes the stress evolution during plastic deformation.

Table 2. Mechanical properties of cortical and cancellous bone.

Properties	Cortical bone	Cancellous bone
Young's modulus (GPa)	14.5	1.37
Poisson's ratio (ν)	0.323	0.3
Density (kg/m^3), (ρ)	1100	270
Friction coefficient	0.35	0.35

$$\sigma = \left(\mathrm{A} + \mathrm{B}(\varepsilon_p)^n\right)\left(1 + \mathrm{C}\ln\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right) \times \left(1 - \left(\frac{\mathrm{T}-\mathrm{T}_{\mathrm{room}}}{\mathrm{T}_{\mathrm{melt}} - \mathrm{T}_{\mathrm{room}}}\right)^m\right) \tag{1}$$

where A, B, n, C, ε_p, $\dot{\varepsilon}$, $\dot{\varepsilon}_0$ and m the yield stress, the hardening modulus, the hardening exponent, the strain rate coefficient, the equivalent plastic strain, non-dimensional plastic strain rate, the reference strain rate and thermal softening coefficient, respectively. Also, $\mathrm{T}_{\mathrm{room}}$ and $\mathrm{T}_{\mathrm{melt}}$ are room and melt temperature, respectively.

The ductile damage initiation criterion considers that the equivalent plastic strain ($\bar{\varepsilon}_D^{pl}$) at the beginning of the damage is a function of the stress triaxiality and the strain rate:

$$\bar{\varepsilon}_D^{pl}(\eta, \dot{\bar{\varepsilon}}^{pl}) \tag{2}$$

where $\eta = -p/q$ is the stress triaxiality (p is the pressure stress and q is the Mises equivalent stress) and $\dot{\bar{\varepsilon}}^{pl}$ is the equivalent plastic strain rate. The Johnson-Cook dynamic damage model depends on the value of the equivalent plastic strain at element integration points, and damage is assumed to take place when the damage parameter (W_D) is more than 1. The damage parameter (W_D) is defined as [18],

$$w_D = \sum\left(\frac{\Delta\bar{\varepsilon}^{pl}}{\bar{\varepsilon}_D^{pl}(\eta, \dot{\bar{\varepsilon}}^{pl})}\right) \tag{3}$$

where $\Delta\bar{\varepsilon}^{pl}$ is an increment of the equivalent plastic strain. Fracture strain, Stress triaxiality and Strain rate values are 0.0001, 0.5 and 1 for cortical bone while Fracture strain, Stress triaxiality and Strain rate are 0.0002, 0.5 and 1 for cancellous bone.

The Johnson-Cook Material properties are showed in Table 3. It was accepted that the cortical bone and the cancellous bone were perfectly connected to each

Table 3. Johnson-Cook plasticity model constants.

	Cortical bone	Cancellous bone
Yield stress (A)	90 MPa	28
Hardening modulus (B)	0.1 MPa	0.1 MPa
Hardening exponent (n)	0.1	0.1
Thermal softening coefficient (m)	0.02	0.02
Strain rate coefficient (C)	0.03	0.015
Reference strain rate ($\dot{\varepsilon}_0$)	0.001	0.001
T_{melt} (K)	1573	
T_{room} (K)	293	

other. The General Contact Algorithm (contact interaction domain-All with self) was used to define the contact between bone and implant with tangential behavior and penalty friction formulation with friction coefficient of 0.35 (as seen Table 2) in ABAQUS/Explicit. Although The General Contact Algorithm can be used only with three-dimensional surfaces and in mechanical finite-sliding contact analyses, it allows very simple definitions of contact with very few restrictions on the types of surfaces involved.

Simulation technique

Dynamic explicit finite element analysis was performed during implant placement. A finite element model has been created and simulated to enhance the implantation process. Figure 3 shows that the implant is delivered with a speed of 80 rpm applied to the top of the implant (Vaillancourt et al. 1995; Sumer et al. 2014; Pellicer-Chover et al. 2017).

Stresses in the bone while the implant insertion can result in bone resorption. It is especially important to determine where stress and damage occurs when sending the implant. The Von Mises stresses are measured along the VV lines in the bone (Figure 4). Line VV is 10 mm for all bone cavity diameters (Figure 4). Stresses on this line were evaluated. The distances of VV away from the bone cavity surface is fixed at 0.5 mm.

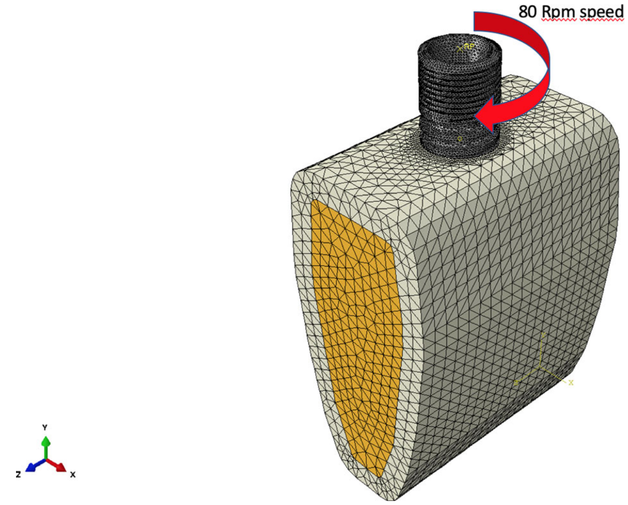

Figure 3. The rotational speed of implant placement protocol.

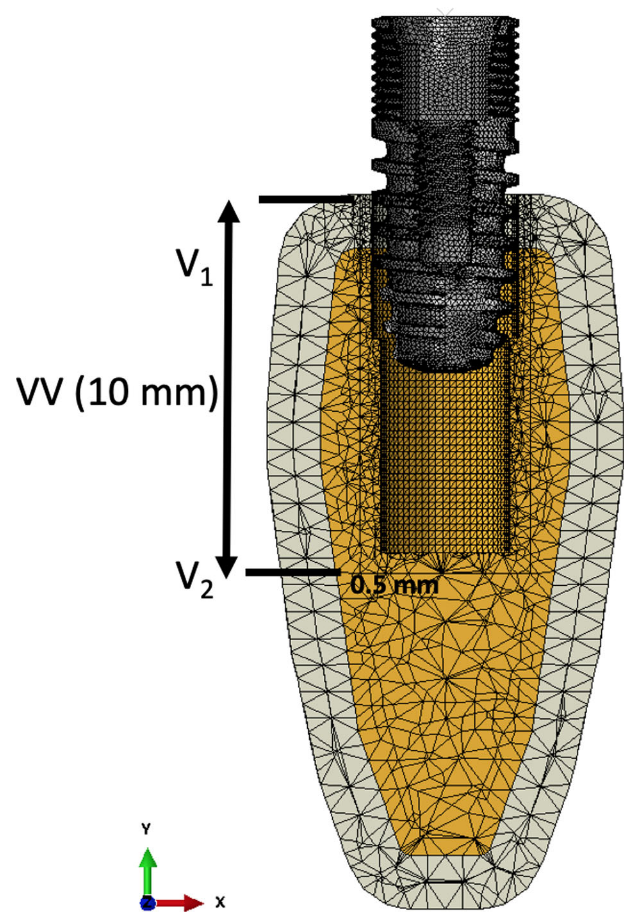

Figure 4. The length of lines VV (in mm).

Results

Von-Mises stresses were evaluated on the bones where both drills were used during implant placement. Stress in bone using cortical drill was lower than using standard drill alone. The stresses that occur in the jawbone at the specified times during the delivery of the implants are shown in Figures 5 and 6. Stresses are measured along the lines VV for the times determined at both implant sockets. Figures 5 and 6 present the stress characteristics within the bone. The mean stress at the time of implant placement to the bone using cortical drill was lower than the standard drill at all times. When Figures 5 and 6 were examined, it was observed that there was intense stress especially when the cortical drill was not used in the neck area of the implant.

When the cortical drill was used, the highest stress was observed in 0.96 s when the implant first contacted the bone, but no stress was observed in the neck region of the implant, because the implant and cortical/cancellous bone are not yet in direct contact

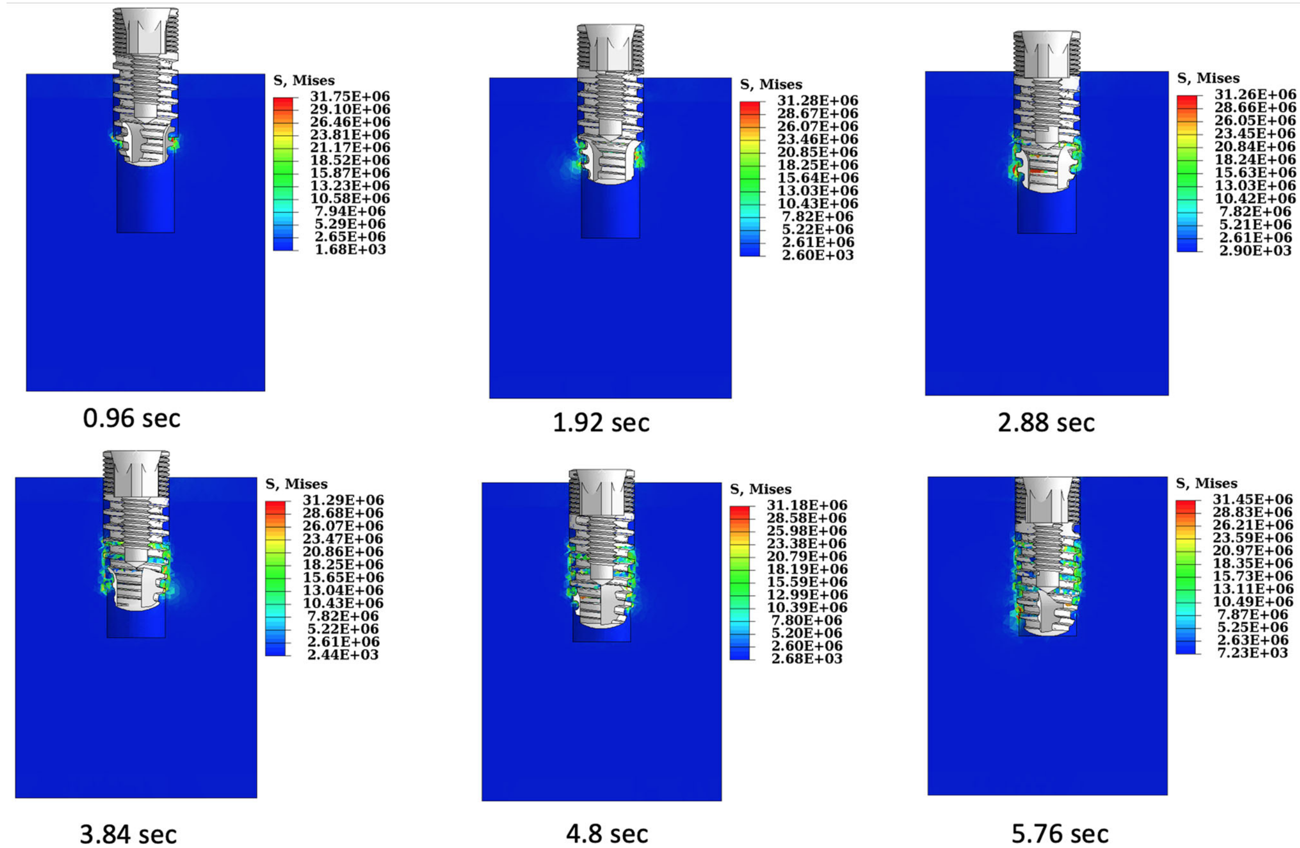

Figure 5. Stress characteristics in the bone during the implant placement at identified times for cortical drill.

(Figures 5 and 6). The highest strain was determined with an average of 15.87 MPa in 0.96 s and the lowest strain with an average of 15.59 MPa in 4.8 s (Figures 5 and 6).

When cortical drill was not used, the highest strain was observed in the neck region at the time of implant insertion (Figures 7 and 8). While the highest strain was determined with an average of 53.69 MPa in 1.73 s, the lowest value was observed with an average of 43.39 MPa in 6.92 s.

When the cortical drill was not used, the strain began in the neck area of the implant, whereas when the cortical drill was used, there was no strain in the neck area of the implant. Stress was also observed in the cancellous bone when cortical drill was used, while cortical bone was stressed when no cortical drill was used.

Figure 9 shows resisting torque-time and resisting torque-penetration graphics, respectively, for the standard drilling and the standard drilling with cortical drill for during implantation. When the implantation process is initiated, it is assumed that the implant is inserted 1.0 mm into the mandible . It can be clearly seen from Figure 9 that the implantation time of the standard drilling is much higher than the standard drilling with the cortical drill. Also, it can be seen that the resisting torque (reached more than 200 Ncm) is much higher at the beginning of standard drilling due to the behavior of cortical bone. Resisting torque level of the standard drilling decreases to almost the same level with cortical drill after the first 3 s. The insertion time of the implant into the bone using standard drill alone was 10.38 s. The insertion time of the implant to the bone using cortical drill was 6.92 s.

The amount of bone damage while the implant insertion was also shown in Figures 10 and 11. It was observed that the implant sent to the bone without cortical drill had more damage. Especially in the marginal cortical bone region, where resorption is quite common, the damage observed when sending the implant is greater (Video). In contrast, when the cortical drill is used, there is no damage to the neck area.

Discussion

The FEA has been frequently used for stress analysis in both science and industry. The FEA allows the estimation of the stress and strain state of extremely geometrically complex systems such as the dental implant–bone system (Geng et al. 2001). This FEA

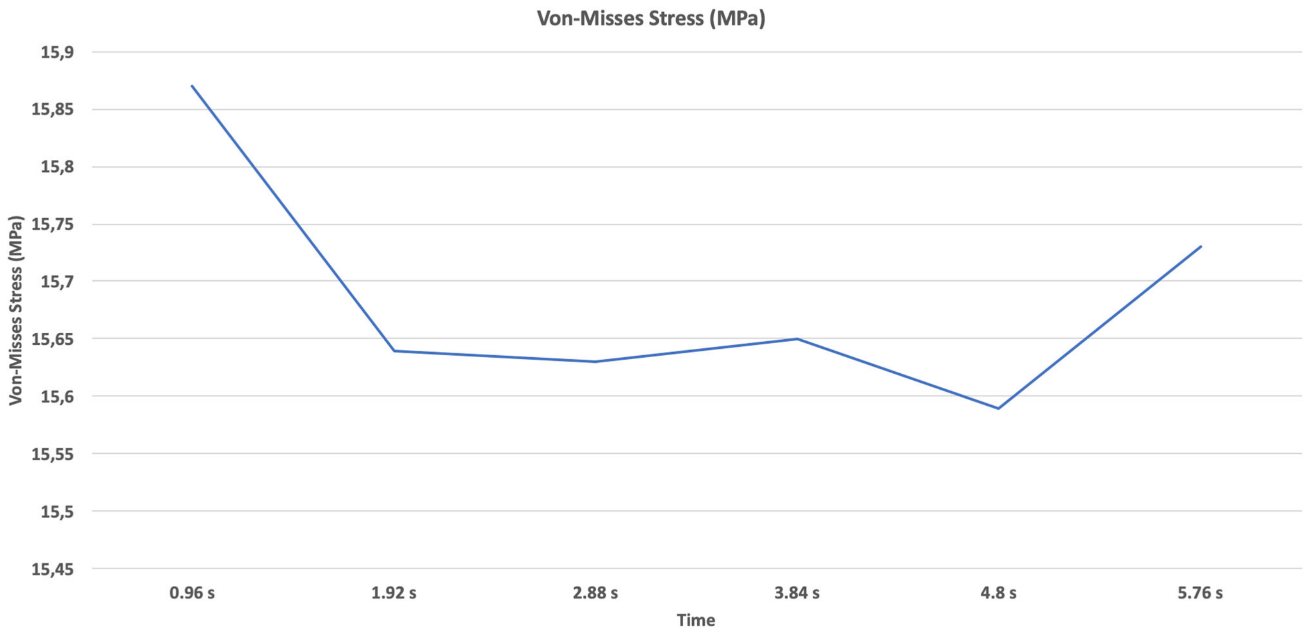

Figure 6. Stress vs time during the implantation for cortical drill.

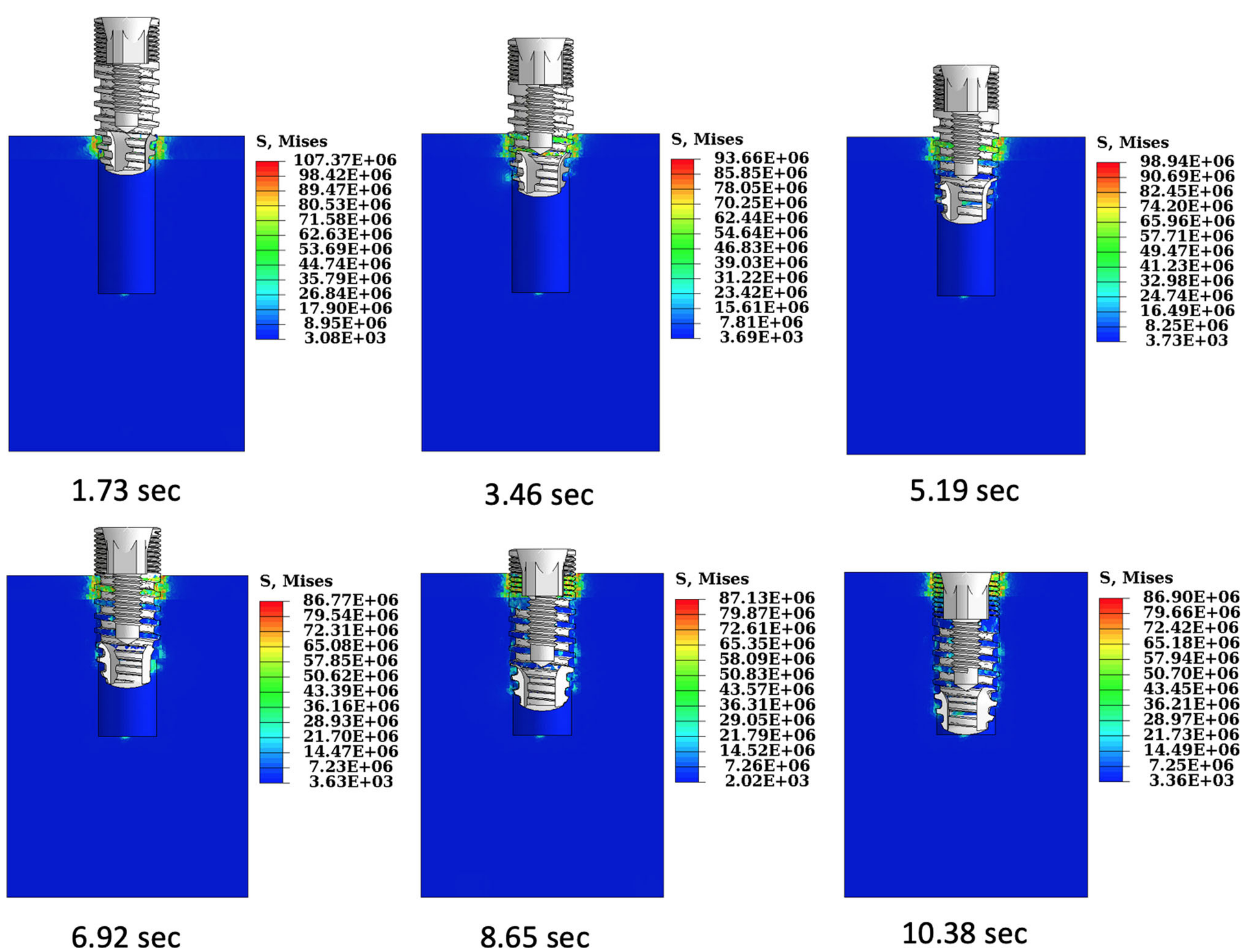

Figure 7. Stress characteristics in the bone during the implant placement at identified times for standard drill.

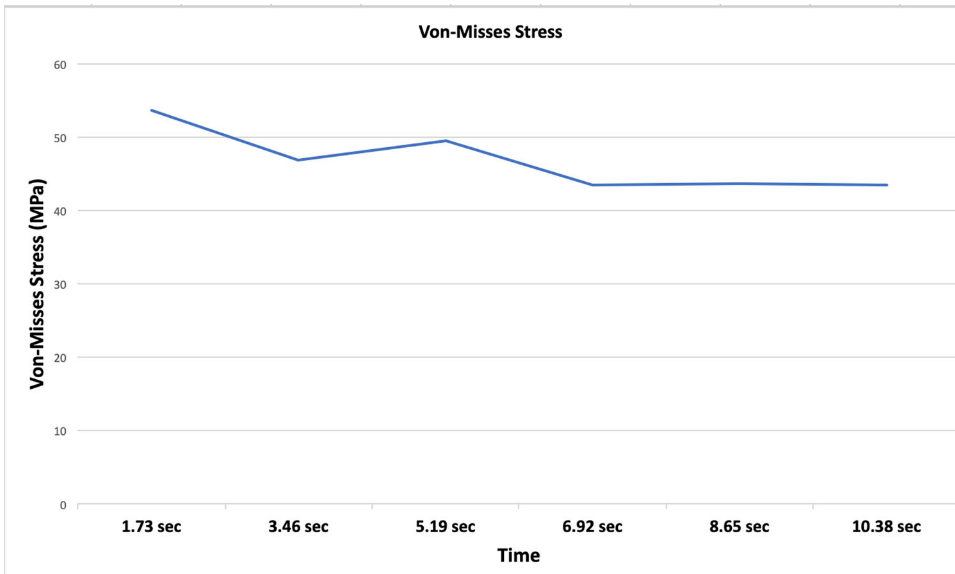

Figure 8. Stress vs time during the implantation for standard drill.

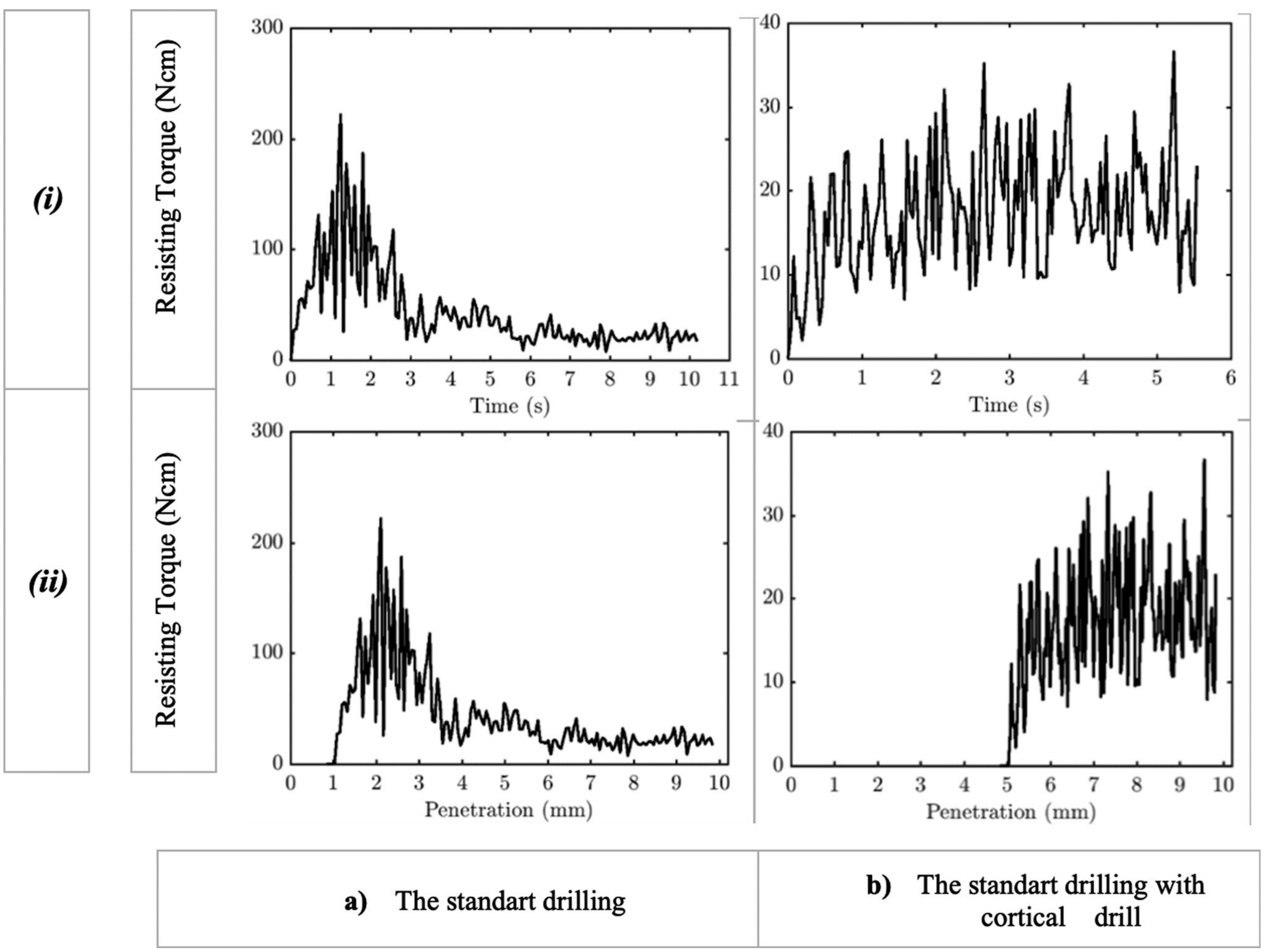

Figure 9. (i) Resisting torque vs. time graphic and (ii) Resisting torque vs. penetration graphic for (a) the standard drilling and (b) the standard drilling with cortical drill for during implantation.

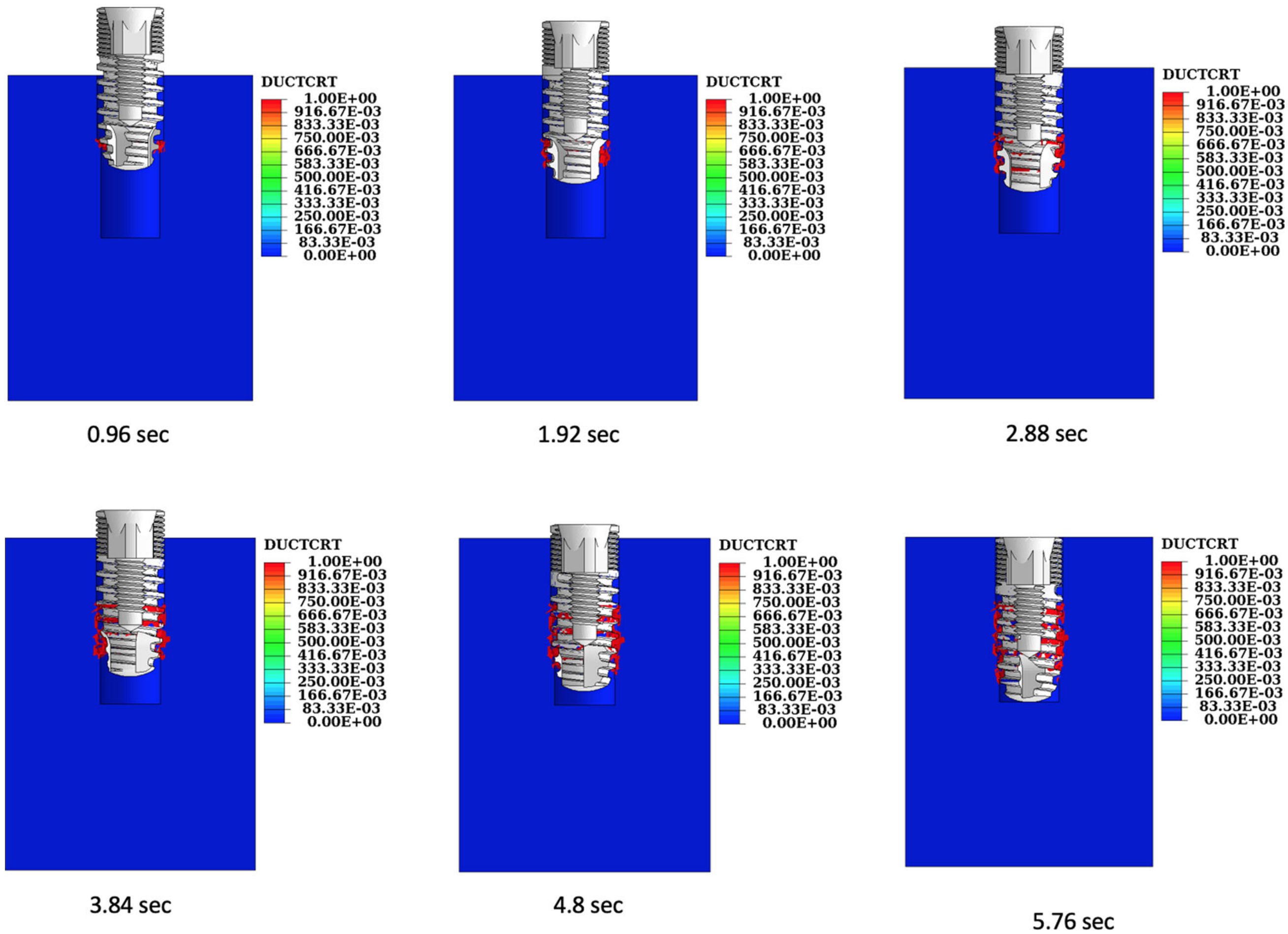

Figure 10. Damage characteristics in the bone during the implant placement at identified times for cortical drill.

enables simulation of complex dynamic physical systems by constructing approximate numerical solutions that describe the response of any system to applied loads (DeTolla et al. 2000). Also the FEA has been used widely to predict the biomechanical performance of various dental implant designs as well as the effect of clinical factors on the success of the implantation (Staden et al. 2006). It is very difficult to evaluate the stress of the implant in the human mouth, therefore FEA is used for this purpose. The length, diameter and shape of the implants and the biomechanical bond between the implant and implants parts have been investigated in many researches. Most of those researches focused on the stresses occurred in bony and material structures after implant placement and have been using static FEM analyses method. Another research interest of the FEA studies related to dental implant and jawbone should focus dynamic FEM analysis as much as static one. However, in the literature, there are very few studies related the dynamic FEA. In this study, the authors aimed to investigate the stresses and possible damages occurring while implant insertion into the jawbone using a novel dynamic FEA study model. For this purpose, 3 D bone models created with and without cortical bone drilling and the stress patterns in both cancellous and cortical bone were examined during implant insertion. Bone stresses has been occurred around the dental implant, especially in the marginal cortical bone, during implant insertion. On the other hand, the main reason for the increased stresses is to obtain primary stability of the implant in the first placement. Clinicians can measure the primary stability of an implant during implant placement owing to the insertion torque. Sometimes this insertion torque can reach high values and causes irreversible damages on the marginal cortical bone around the dental implant and dramatic marginal bone loss and/or failure of the dental implant can be seen in a short period after implantation. As we know that the cortical bone drilling that is recommended by almost all dental implant manufacturers as a final sequence in the implant bed preparation procedure can reduce cortical bone stresses especially in dense bones during the implant insertion in contrast to standard drilling without the cortical drill. But this stress occurred on the jawbone around

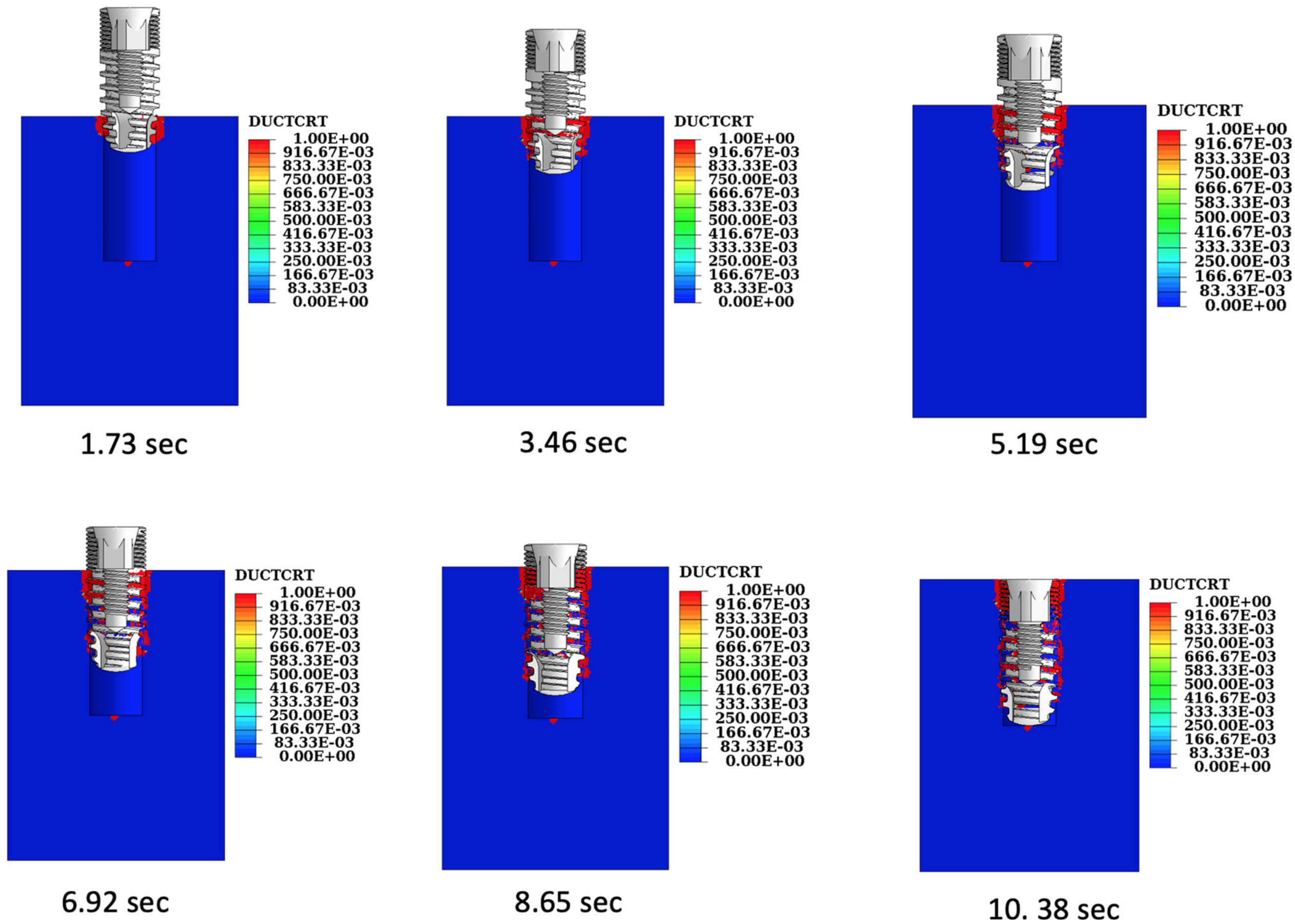

Figure 11. Damage characteristics in the bone during the implant placement at identified times for standard drill.

dental implant cannot be evaluated objectively during the implant insertion clinically. The analysis undertaken in this study is to advance the current understanding of the stress characteristics within the crestal region of the jawbone during the implantation process using the dynamic FEA.

Primary stabilization of implants has a very important place in the success of osseointegration (Branemark 1983; Staden et al. 2008). On the other hand, many factors including the quality and quantity of the bone, the geometry and design of the implant, and the surgical method applied affect the primary stabilization (Shafiullah et al. 2021). It can be considered that the use of cortical drill may have a negative impact on the primary stabilization of the implant. The primary stabilization is directly related to the density and structure of bone tissue (Ödman et al. 1988; Marquezan et al. 2012; Shah et al. 2012). It is difficult to achieve an optimum insertion torque and primary stability, especially in bones with low bone quality and density (Marquezan et al. 2012). Thus, in such bones, it is necessary to maintain the existing bone mass as much as possible (Gayathri 2018). On the other hand, optimum primary stabilization can be achieved in bones with high cortical content (D1-D2) (Gayathri 2018). Accordingly, D2 bone type was selected in this study. In addition, today, while the primary stabilization of drills can be increased by the designs of implants and applied surgical techniques, necrosis and resorption depending on the stress and damage occurring in the bone can cause greater problems. It was observed in this study that the use of cortical drilling reduced the stress and damage in the neck region, but did not have a negative impact on the primary stabilization in the apical region of the implant. The primary goals are to enlarge the neck area, to prevent stress and damage around the implant, and to ensure optimum healing of the region.

Misch stated that there are four bone types according to the bone density in the edentulous maxilla and the ridge of the mandible (Misch 1990). The type and quality of the bone have an important place in the success of the implant. The bone types with the most crestal bone resorption and related failure are D1 and D2 bones. D3 and D4 bones can distribute the incoming stress more easily due to their cancellous structure. (Monstaporn et al., 2020; Uhthoff and Jaworski 1978). Stress distribution has been shown to occur primarily during the primary contact area of bone and dental implant, mostly over cortical crestal bone (Misch 1990; Monstaporn et al., 2020). In this study, it was preferred to study on D2 bone, where crestal bone resorption is relatively higher, and while the implant was being delivered, it was investigated by dynamically modeling where stress and damage were observed in the bone.

During the implant placement, the stress in the bone has an important role in the long-term success of the implant stability. Many factors are important in the successful completion of the implant placement such as; the insertion speed, the torque applied to the implant, the preparation of the region before implant placement and bone stresses around the dental implant (Staden et al. 2008). There are limited studies examining stress during implant placement by FEM analysis (Staden et al. 2008). Staden et al. (2008) used a simplified modeling approach to examine the process of implant insertion. Authors created a new model for each 1 mm implant placement and created a step-wise analysis of the implant insertion over the change in torque value without changing the time (Staden et al. 2008). Also, authors didn't simulate any rotational continuous movement while inserting the implant. In this study, dynamic explicit finite element analysis was performed during implant placement. Dynamic analysis was performed on a single model.

During the osseointegration process, optimal stress levels are required to maintain normal bone repair (Degidi et al. 2009). During the implant insertion process, the creation of the ideal stress level will prevent fracture formation in the bone where the implant is placed, and optimum wound healing and osseointegration will be obtained (Schmid et al. 2002; Staden et al. 2008). According to Wolff's theory, resorption or healing in the bone is directly related to the stress within the bone. Furthermore, Meredith N. et al. said that it was suggested that excessive insertion torque lead to crestal stress and heat at the border between implant and bone, and mechanical injury can cause degeneration of the bone at the implant-tissue interface (Meredith 1998). The results of the analysis in this study showed that when the cortical drill was not used, the stress and damage to the bone in the neck region of the implant when sending the implant was quite high. It can be seen that the stress profile peaks at the top point along the line V–V in both sockets (Figures 5 and 7). This is due to the cortical bone located at the top. As the implant progresses through the bone, the decrease in stress is caused by contact with the cancellous bone. By using cortical drill, the implant moves forward from the wide socket at the first insertion and provides direct contact with the cancellous bone and the resulting stress is reduced compared to the socket where the standard drill is used. Stress and damage in the neck area, where the amount of resorption is frequently observed, affects the success and life of the implant. Although, implant insertion time is not related just only using the cortical drill in clinical practice and the time can change according to the different rpm that the clinician program the implant and oral surgery motor, it will contribute to shortening the operation time.

Exposure of the bone to various factors such as excessive stress and heat causes cellular damage and resorption in the bone (Ravishankar 2021). Most of the studies in the literature have focused on examining the stress and associated damage caused by dental implants in the bone after osseointegration and loading (Dinc et al. 2021; Linkevicius et al. 2021; Saglanmak et al. 2021). On the other hand, studies that dynamically examine stress and damage occurring in the bone while delivering the implant to the prepared socket are limited. Excessive stress and damage that will occur during implant delivery will adversely affect healing and hence the success of the implant. This study aimed to draw attention to the regions in which stress and damage occurs in the bone while delivering the implant. The results of the study showed that the stress tension and damage in the crestal bone region, which plays important role in the success of the implant, was maximum when the cortical drill was not used, and the use of the cortical drill minimized this stress. Clinically, the use of cortical drills is recommended in bones with high cortical density, the present study supports this knowledge. The results obtained in the study indicated that the maximum stress and damage occur especially in the crestal bone region, and stress and damage begin during implant delivery before loading. In the light of the results of the study, it is thought that the use of cortical drills in the implant drilling protocol may be a necessity rather than a producers' recommendation.

Limitation of the current study is the lack of current knowledge of the dynamic mechanical properties of the jawbone. As we know that the Young's modulus, Poisson's ratio and density of a jawbone were introduced and well defined for static FEM analyses studies in the literature (Rieger et al. 1989; Meijer et al. 1994; Baggi et al. 2008). But, according to our knowledge, there isn't yet an exact definition of the mechanical properties of the jawbone under dynamic forces.

In this study, a simplified and effective 3 D FEA modeling procedure was proposed to examine the properties of stress occurring in the mandible during the implantation procedure. The study considers realistic geometry, material properties, loading and support conditions as well as biomechanics for both the implant and the jawbone. As a result of this study, when the cortical drill was not used, it was observed that intense stress and damage occurred especially in the marginal cortical bone area during the implantation procedure. In the use of cortical drills, it was observed there was no damage and stress in the marginal cortical bone. In this pilot study, bones stresses around the dental implant were analyzed during the implant insertion using a novel 3 D dynamic FEA model, but further dynamic FEM studies are needed to investigate the bone behaviors under different scenarios with introducing dynamic mechanic properties of the jawbone.

Disclosure statement

No potential conflict of interest was reported by the authors.

Funding

The author(s) reported there is no funding associated with the work featured in this article.

ORCID

Recep Ekici http://orcid.org/0000-0002-4420-8431

References

Baggi L, Cappelloni I, Maceri F, Vairo G. 2008. Stress-based performance evaluation of osseointegrated dental implants by finite-element simulation. Simul Modell Pract Theory. 16(8):971–987.

Bozkaya D, Muftu S, Muftu A. 2004. Evaluation of load transfer characteristics of five different implants in compact bone at different load levels by finite elements analysis. J Prosthet Dent. 92(6):523–530.

Branemark P-I. 1983. Osseointegration and its experimental background. J Prosthet Dent. 50(3):399–410.

Brunski JB. 1992. Biomechanical factors affecting the bone-dental implant interface. Clin Mater. 10(3):153–201.

Deck C, Nicolle S, Willinger R. 2004. Human head FE modelling: Improvement of skull geometry and brain constitutive laws. In Proceedings of the 2004 International IRCOBI Conference on the Biomechanics of Impact, Graz, Austria, p. 79–92.

Degidi M, Perrotti V, Strocchi R, Piattelli A, Iezzi G. 2009. Is Insertion torque correlated to bone–implant contact percentage in the early healing period? A histological and histomorphometrical evaluation of 17 human-retrieved dental implants. Clin Oral Implants Res. 20(8):778–781.

DeTolla DH, Andreana S, Patra A, Buhite R, Comella B. 2000. The role of the finite element model in dental implants. J Oral Implantol. 26(2):77–81.

Dinc MM, Turkoglu P, Selvi F. 2021. biomechanical evaluation of stress distributions at the implant-abutment complex and peri-implant bone around mandibular dental implants with different neck geometries and inclinations. In Proceedings of the Institution of Mechanical Engineers, Part H: Journal of Engineering in Medicine, p. 09544119211022985.

Gayathri S. 2018. Osseodensification technique–a novel bone preservation method to enhance implant stability. Acta Sci Dent Sci. 2:17–22.

Gedrange T, Bourauel C, Köbel C, Harzer W. 2003. Three-dimensional analysis of endosseous palatal implants and bones after vertical, horizontal, and diagonal force application. Eur J Orthod. 25(2):109–115.

Geng JP, Tan KB, Liu GR. 2001. Application of finite element analysis in implant dentistry: A review of the literature. J Prosthet Dent. 85(6):585–598.

Hasan I, Heinemann F, Bourauel C. 2014. Biomechanical finite element analysis of self-tapping implants with different dimensions inserted in two bone qualities. Biomed Eng/Biomed Technik. 59(3):203–212.

Lewinstein I, Banks-Sills L, Eliasi R. 1995. Finite element analysis of a new system (IL) for supporting an implant-retained cantilever prosthesis. Int J Oral Maxillofac Implants. 10(3):355–366.

Linkevicius T, Linkevicius R, Gineviciute E, Alkimavicius J, Mazeikiene A, Linkeviciene L. 2021. The influence of new immediate tissue level abutment on crestal bone stability of subcrestally placed implants: A 1-year randomized controlled clinical trial. Clin Implant Dent Relat Res. 23(2):259–269.

Marquezan M, Osório A, Sant'Anna E, Souza MM, Maia L. 2012. Does bone mineral density influence the primary stability of dental implants? A systematic review. Clin Oral Implants Res. 23(7):767–774.

Meijer HJA, Starmans FJM, Steen WHA, Bosman F. 1994. Location of implants in the interforaminal region of the mandible and the consequences for the design of the superstructure. J Oral Rehabil. 21(1):47–56.

Meredith N. 1998. Assessment of implant stability as a prognostic determinant. Int J Prosthodont. 11(5):491–505.

Misch CE. 1990. Density of bone: Effect on treatment plans, surgical approach, healing, and progressive boen loading. Int J Oral Implantol. 6(2):23–31.

Mohammed H, Atmaram GH, Schoen FJ. 1979. Dental implant design: A critical review. J Oral Implantol. 8(3):393–410.

Monstaporn M, Rungsiyakull C, Jia-Mahasap W, Pleumsamran N, Mahrous A, Rungsiyakull P. 2020. Effects of bone types on bone remodeling of a dental implant: A review of the literature. CM Dent J. 42(2):13–19.

Ödman J, Lekholm U, Jemt T, Brånemark P-I, Thilander B. 1988. Osseointegrated titanium implants-a new approach in orthodontic treatment. Eur J Orthod. 10(2):98–105.

Pellicer-Chover H, Peñarrocha-Oltra D, Aloy-Prosper A, Sanchis-Gonzalez J-C, Peñarrocha-Diago M, Peñarrocha-Diago M. 2017. Comparison of peri-implant bone loss between conventional drilling with irrigation versus low-speed drilling without irrigation. Med Oral Patol Oral Cir Bucal. 22(6):e730.

Ravishankar K. 2021. Crestal bone and dental implant: An overview. Glob Acad J Dent Oral Health. 3(2):9–14.

Rieger MR, Fareed K, Adams WK, Tanquist RA. 1989. Bone stress distribution for three endosseous implants. J Prosthet Dent. 61(2):223–228.

Saglanmak A, Gultekin A, Cinar C, Szmukler-Moncler S, Karabuda C. 2021. Effect of soft tissue thickness on crestal bone loss of early loaded implants with platform switching: 1-and 5-Year data. Quintess Int. 52(5):426.

Schmid MR, Schiel HJ, Lambrecht JT. 2002. Torque of endosseous dental screw type implants. Schweizer Monatsschrift Fur Zahnmedizin = Revue Mensuelle Suisse d'odonto-Stomatologie = Rivista Mensile Svizzera Di Odontologia e. Stomatologia. 112(8):804–813.

Schrotenboer J, Tsao Y-P, Kinariwala V, Wang H-L. 2008. Effect of microthreads and platform switching on crestal bone stress levels: A finite element analysis. J Periodontol. 79(11):2166–2172.

Shafiullah RS, Hariharan R, Krishnan CS, Azhagarasan NS, JayaKrishnaKumar S, Saravanakumar M. 2021. Influence of cortical layer and surgical techniques on the primary implant stability in low-density bone: An in vitro study. J Contemp Dent Pract. 22(2):146–151.

Shah R, Mahajan A, Shah N, Dadhania AP. 2012. Preemptive analgesia in third molar impaction surgery. Natl J Maxillofac Surg. 3(2):144–147.

Staden RCV, Guan H, Loo Y-C. 2006. Application of the finite element method in dental implant research. Comput Methods Biomech Biomed Eng. 9(4):257–270.

Staden RCV, Guan H, Johnson NW, Loo Y-C, Meredith N. 2008. Step-wise analysis of the dental implant insertion process using the finite element technique . Clin Oral Implants Res. 19(3):303–313.

Sumer M, Keskiner I, Mercan U, Misir F, Cankaya S. 2014. Assessment of heat generation during implant insertion. J Prosthet Dent. 112(3):522–525.

Uhthoff HK, Jaworski ZF. 1978. Bone loss in response to long-term immobilisation. J Bone Joint Surg Brit. 60(3): 420–429.

Vaillancourt H, Pilliar RM, McCammond D. 1995. Finite element analysis of crestal bone loss around porous-coated dental implants. J Appl Biomater. 6(4):267–282.

Weinstein AM, Klawitter JJ, Anand SC, Schuessler R. 1976. Stress analysis of porous rooted dental implants. J Dent Res. 55(5):772–777.

Wolff J. 1892. The law of bone transformation. Berlin: Hirschwald.