The Effect of Crown Geometry on Stress Distribution of a Single Implant Restoration: A Finite Element Analysis

Kron Geometrisinin Tek İmplant Restorasyonunun Kuvvet Dağılımı Üzerine Etkisi: Sonlu Elemanlar Analizi

Mümin KÜÇÜK,[a]
M. Erhan ÇÖMLEKOĞLU,[b]
Mehmet ZOR[a]

[a]Department of Mechanical Engineering, Dokuz Eylul University Faculty of Medicine,
[b]Department of Prosthodontics, Ege University Faculty of Dentistry, İzmir

Geliş Tarihi/*Received:* 02.05.2009
Kabul Tarihi/*Accepted:* 12.10.2009

Yazışma Adresi/*Correspondence:*
M. Erhan ÇÖMLEKOĞLU
Ege University Faculty of Dentistry, Department of Prosthodontics, İzmir, TÜRKİYE/TURKEY
erhancomlek@yahoo.com

ABSTRACT Objective: The influence of occlusal loading location on stress distribution of an implant and surrounding bone is less documented in literature. This study presents the stress analysis of a mandibular second molar implant restored with a metal-ceramic crown under different loading conditions using the three-dimensional finite element analysis. **Material and Methods:** The implant model was adapted from ITI (Straumann, Switzerland) with a 1-piece (solid) 4.1 x 10 mm screw-shaped abutment and the parameters for a cobalt-chromium metal framework and feldspathic ceramic were used for crown modelling. The crown, implant, abutment and the alveolar bone were respectively modelled with SolidWorks software. Then stress distributions under 58 N load were calculated by ANSYS 10.0 software in 3 cusp angles (3°, 22° and 45° with the horizontal plane). The distribution of stresses were plotted for some critical points which were the transition points of compression to tension or tension to compression under load. **Results:** Maximum stresses occurred at the implant-abutment junction in all models with differing values. As the cusp angle increased, maximum Von Mises and shear stresses increased while principal stresses decreased. Tensile stresses were observed at the implant-alveolar bone junction while compressive stresses occurred at the other areas without causing deformation on the alveolar bone. **Conclusion:** The crown geometry of a single crown with shallower occlusal morphology produced more favorable stress distribution.

Key Words: Finite element analysis; dental implants, single-tooth; dental prosthesis, implant-supported

ÖZET Amaç: Oklüzal yük konumunun implant ve çevre sert dokular üzerine etkisi literatürde az belgelenmiştir. Bu çalışmada, metal-seramik kron ile restore edilmiş alt çene ikinci azı bölgesindeki bir implantın farklı yükleme durumları altındaki stres dağılımları, üç boyutlu sonlu elemanlar analizi yöntemiyle araştırılmıştır. G**ereç ve Yöntemler:** İmplant ile tek parça (solid), 4.1 x 10 mm vida şeklindeki dayanak modelleri için ITI (Straumann, İsviçre) esas alınırken, kron modellemesi için kobalt-krom metal altyapı ve feldspatik seramik parametreleri kullanılmıştır. Kron, implant, dayanak ve alveoler kemik sırasıyla SolidWorks yazılımıyla modellenmiştir. Üç farklı tüberkül açısı (yatay düzlemle 3°, 22° ve 45°) için 58 N yük altındaki stres dağılımları ANSYS 10.0 yazılımı ile hesaplanmıştır. Baskıdan gerilmeye ya da gerilmeden baskıya doğru geçiş noktaları gibi kritik bölgeler için yük altındaki stres dağılımları belirlenmiştir. **Bulgular:** Bütün modeller için maksimum stres değerleri implant-dayanak birleşiminde gerçekleşmiştir. Tüberkül açısı arttıkça, maksimum Von Mises ve makaslama streslerinde yükselme, asal streslerde ise azalma gözlenmiştir. İmplant-alveoler kemik birleşiminde gerilme stresleri, diğer bölgelerde ise alveoler kemikte deformasyona neden olmaksızın baskı stresleri oluşmuştur. **Sonuç:** Tek kron restorasyonlarında, sığ oklüzal morfoloji ile daha kabul edilebilir yük dağılımı elde edilebilir.

Anahtar Kelimeler: Sonlu eleman analizi; diş implantları, tek diş; diş protezi, implant destekli

Turkiye Klinikleri J Dental Sci 2010;16(2):136-41

The introduction of osseointegrated dental implants have increased patients' functional and esthetic demands and prompted their use based on well-documented high success rates.[1-4] Occlusal loading of osseointegrated implants has been defined as a determining factor for long-term success in implant therapy.[5,6] The implant body-abutment-crown assembly should resist to stresses generated as a consequence of functional occlusal forces that are dependent on geometry, material properties and loading conditions.

Occlusal forces and moments act on different parts of the restoration and are transferred to the implants. In patients with normal dentition and implant denture wearers, axial forces are observed. However, the distribution of these forces depend on the direction of the load as well as the properties of the restorative material used.[7,8] Occlusal forces exceeding the capacity of the bone-implant interface to absorb stresses may lead to failure of the implants.[9] Cuspal inclination as well as bone type, horizontal offset, and occlusal anatomy are the important biomechanical factors that contribute to implant overload.[10,11]

For problems involving complicated geometries, it is very difficult to achieve an analytical solution. Therefore, the use of numerical methods such as finite element analysis (FEA) is required. FEA is a technique for obtaining a solution to a complex mechanical problem by dividing the problem domain into a collection of much smaller and simpler domains (elements) in which the field variables can be interpolated with the use of shape functions. An overall approximated solution to the original problem is determined based on variational principles. Because the components in a dental implant-alveolar bone system are extremely complex geometrically, FEA has been viewed as the most suitable tool for analyzing them. A mesh is needed in FEA to divide the whole domain into elements. The process of creating the mesh, elements, their respective nodes, and defining boundary conditions is referred to as “discretization” of the problem domain.[12]

Three-dimensional (3D) finite-element analysis has been widely used among the methods for the evaluation of implant biomechanics for the quantitative evaluation of stresses and strains in the bone due to technical limitations of stress assessment in bone *in vivo*.[13,14]

Masticatory forces induce axial loads and bending moment and result in stress gradients in the implant as well as in the bone. For the success or failure of a dental implant, the manner in which stresses are transferred to the surrounding bone is of vital importance.[12] Therefore, the objective of this study was to evaluate the stress patterns on an alveolar bone- implant-abutment-crown assembly under loading with 3 different cusp angles.

MATERIAL AND METHODS

A 3D finite element model of a missing mandibular second molar replacing implant with the abutment, crown and surrounding type II bone block were used in the study (Figure 1a and 1b). A one-piece, screw-shaped dental implant with a solid abutment tightened on the implant (radius: 4.1mm, length: 10 mm) (ITI; Institute Straumann AG, Waldenburg, Switzerland) was selected for the study. The implant and its superstructure were modelled with the use of a software (SolidWorks, Dassault Systems, Suresnes, France).

Three crowns with different cuspal inclinations (3º, 22º and 45º) were modelled (Figure 1c). The parameters of a feldspathic ceramic (Vita, Bad Sackingen, Germany) crown with a base metal (Co-Cr) framework (Wiron 99, Bego, Bremen, Germany) was used to simulate the superstructure. Ceramic and metal thicknesses used in this study were 0.5–1.5 mm. Cement thickness was ignored.[15] The geometry of the crown model has been described by Wheeler.[16]

All materials were assumed to be linearly elastic, homogeneous, and isotropic.[17] The required analytical properties were derived from a literature survey, and are listed in Table 1.[18] The whole model consisted of 8019 nodes and 45221 elements.

An average vertical force of 58 N was applied axially on the three buccal cuspal tips.[19] The applied forces were static. Stress levels were calculated using Von Mises, shear and principal stres values. The final element on *x* axis for each design was assumed to be fixed which defined boundary condition. Stress findings were evaluated for each cuspal

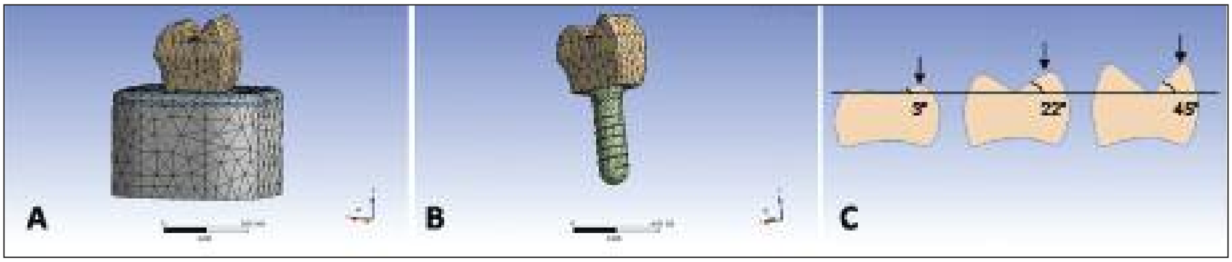

FIGURE 1: (A) Three dimensional model of the implant-alveolar bone-abutment-crown assembly, **(B)** Mesh model of the implant-abutment-crown assembly, **(C)** Different cuspal inclinations and distribution of load applied to finite element model.

design (3°, 22° and 45°). Boundary conditions, loading and mathematical model were prepared with a finite element software (ANSYS 10.0, Ansys Corp., Houston, USA) to display stress values and distributions.

RESULTS

Application of axial force on different cuspal angles in an implant-crown design in the present study influenced the localizations of stresses.

Maximum Von Mises stress distributions were observed around the cervical region of the solid abutment and abutment-implant junction in all 3º, 22º and 45º inclinations (115,03 MPa, 128,33 MPa and 146,32 MPa, respectively) (Figure 2a-c). As the cuspal angle increased, maximum Von Mises and shear stres values increased, and the principal stress values decreased (Table 2). Tensile stresses were observed at the implant-alveolar bone junction and while compressive stresses were concentrated at the other regions (Figure 3 a-c). The alveolar bone was not observed to be under significant strain after analysis of the stress distributions in the alveolar bone model. Maximum stress values within the cortical bone surrounding the implant were 91,29 MPa for 3º cuspal angle, while 97,29 MPa for 22º and 122,85 MPa for 45º (Figure 3 a-c).

DISCUSSION

The principal difficulty in simulating the mechanical behavior of dental implants is the modelling of human bone tissue and its response to applied mechanical force. Certain assumptions need to be made to make the modelling and solving process possible. The complexity of the mechanical characterization of bone and its interaction with implant systems has forced authors to make major simplifications. Some assumptions such as detailed geometry of the bone and implant to be modelled, material properties, boundary conditions and the interface between bone and implant influence the accuracy of the FEA results significantly.[20]

TABLE 1: Material properties used for analysis.[14]

Material	Young's Modulus (GPa)	Poisson's Ratio (μ)
Ceramic	65	0.24
Titanium	110	0.33
Alveolar bone	18	0.3
Cobalt-chromium alloy	206	0.33

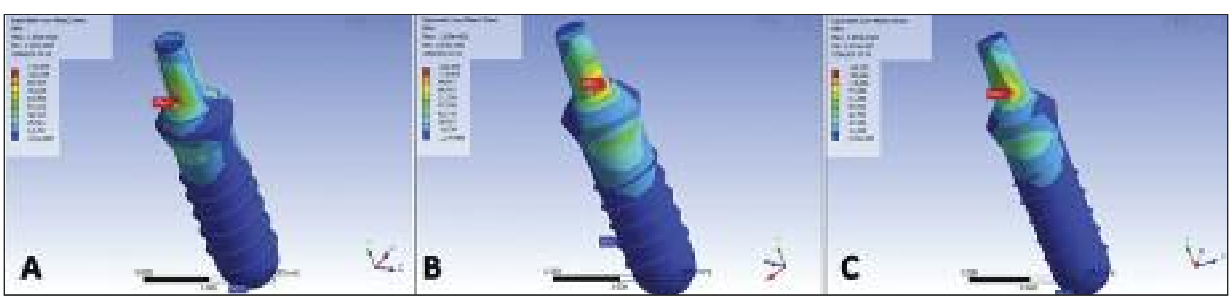

FIGURE 2: Von Mises stress distributions within implant and abutment. **(A)** for 3º cuspal inclination **(B)** 22º cuspal inclination, **(C)** for 45º cuspal inclination.

TABLE 2: Maximum stress values upon force applications on different cuspal angles.

	Maximum stress values		
Cuspal inclination	Von Mises (MPa)	Shear (MPa)	Principal (MPa)
3º	115.03	60.33	91.29
22º	128.33	66.41	97.29
45º	146.32	76.96	122.85

Functional forces transmitted to the supporting bone by the restorative material, abutment and the implant create stresses in implant supported fixed partial dentures.[21] The analysis of stresses in restorative materials and supporting tissues is of great importance since the created stresses should be at physiological levels, and high stress concentrations should be eliminated. In the present study, a finite-element stress analysis method was used to evaluate the stresses generated in the abutment, implant, and supporting bone with various materials used in implant-crown design under functional forces. In a previous finite-element stress analysis study,[15] all materials were accepted to be linearly elastic, homogeneous, and isotropic, and cement thickness was ignored since it was found not to affect the stress distribution.[22] All structures in the model in the present study were assumed to be homogeneous and isotropic and linearly elastic and the cement thickness was ignored. However, the properties of the materials and living tissues modelled in finite element analyses differ such as the actual cortical bone of the mandible is defined to be transversely isotropic and non-homogeneous.[21,23] Besides, in our study, implant–bone interface was assumed to be in full contact, which does not simulate clinical situations.[9] Among the other limitations of the present study, it is also important to point out that the stress distribution patterns may have been different, depending on the materials and properties assigned to each layer of the model. In a previous study on the effect of superstructure material on stress distribution in an implant-supported fixed prosthesis, cobalt–chromium framework with porcelain for the occlusal surface was found to be the optimal combination for suprastructure construction.[24] Therefore the implant supported crown in the present study was also modelled with Co-Cr framework material and ceramic.

In the present study, a 4.1x10 mm solid-screw dental implant was selected since no implant fracture was reported.[25] The design of the occlusal surface of the model may influence the stress distribution pattern, therefore, the locations of the force applications were described as especially the buccal cusp tip since it is well documented that lateral forces are more detrimental in terms of non-homogenous stress distribution in all types of restorations[21,26,27] and implant supported fixed partial dentures are not an exception. Besides, the forces might be transmitted directly to the implant body without the presence of any periodontal ligament supplying stress absorption mechanism as in a natural tooth.

All physiologic biomechanical processes are interrelated, therefore cuspal inclination as well as bone type, horizontal offset, and occlusal anatomy may play a role in implant overload.[28] In this study, metal-ceramic crowns with 3 different cuspal angles were tested. The crown with 45º inclination ex-

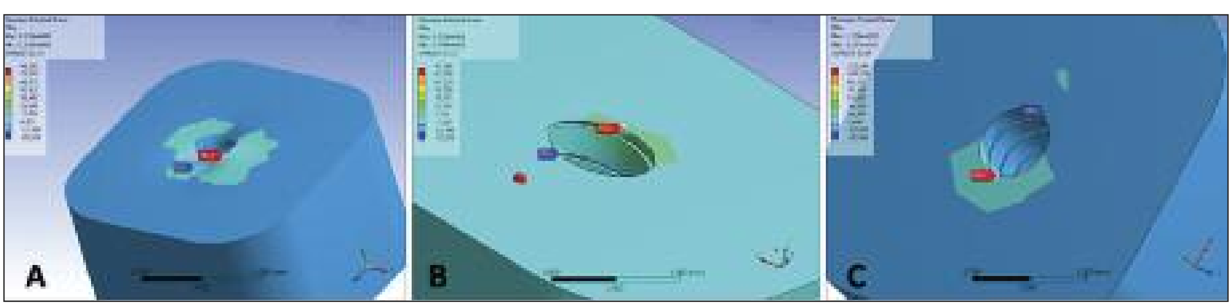

FIGURE 3: Stress distributions within cortical bone. **(A)** for 3º cuspal inclination **(B)** 22º cuspal inclination, **(C)** for 45º cuspal inclination.

hibited the highest stres distribution around the cervical region of the implant, however, crowns with 3º and 22º angulations showed similar load distributions. These results may be attributed to the fact that, as the cuspal angle increased, even under axial loads the bending moment occurs and results in unfavorable stress gradients in the implant as well as in the crestal bone.[12]

Implant overload stem form masticatory forces or parafunctional habits may decrease bone density around the cervical region of implants and lead to crater-like defects.[12,25,28] In the present study, maximum stresses were concentrated around the collar region of implant and the crestal part of the cortical bone due to the rigid connection between the implant and bone. The results of this study showed that, using high cuspal inclination produced high stresses on the alveolar bone. On the other hand, with low cuspal angulations more favorable stress patterns due to vertical resultant force rather than lateral force occured around the surrounding bone.

CONCLUSION

Within the limitations of this study, the following conclusions were drawn:

1. Application of axial force on different cuspal angles in an implant-crown design influenced the localizations and values of stresses at implant and bone tissue surrounding the implant.

2. The crown with 45º inclination exhibited the highest stress distribution around the cervical region of the implant, however, crowns with 3º and 22º angulations showed similar load distributions.

3. The crown geometry of a single crown with shallower occlusal morphology produced more favorable stress distribution.

Acknowledgement

This study was conducted and completed as the MSc Thesis named "İmplant Uygulanmış İnsan Çenesinde Kaplama Geometrisinin Gerilme Üzerine Etkisi" of Kaan BAŞTÜRK, Mechanical Engineer under supervision of Associate Professor Mehmet ZOR.

REFERENCES

1. Hellem S, Karlsson U, Almfeldt I, Brunell G, Hamp SE, Astrand P. Nonsubmerged implants in the treatment of the edentulous lower jaw: a 5-year prospective longitudinal study of ITI hollow screws. Clin Implant Dent Relat Res 2001;3(1):20-9.
2. Bravi F, Bruschi GB, Ferrini F. A 10-year multicenter retrospective clinical study of 1715 implants placed with the edentulous ridge expansion technique. Int J Periodontics Restorative Dent 2007;27(6):557-65.
3. Mengel R, Behle M, Flores-de-Jacoby L. Osseointegrated implants in subjects treated for generalized aggressive periodontitis: 10-year results of a prospective, long-term cohort study. J Periodontol 2007;78(12):2229-37.
4. Blanes RJ, Bernard JP, Blanes ZM, Belser UC. A 10-year prospective study of ITI dental implants placed in the posterior region. I: Clinical and radiographic results. Clin Oral Implants Res 2007;18(6):699-706.
5. Cibrika RM, Razzoog ME, Lang BR, Stohler CS. Determining the force absorption quotient for restorative materials used in implant occlusal surfaces. J Prosthet Dent 1992; 67(3):361-4.
6. Skalak R. Biomechanical considerations in osseointegrated prostheses. J Prosthet Dent 1983;49(6):843-9.
7. Brunski JB. Biomechanical factors affecting the bone–dental implant interface. Clin Mater 1991;10(2):153–201.
8. Assif D, Oren E, Marshak BL, Aviv I. Photoelastic analysis of stress transfer by endodontically treated teeth to the supporting structure using different restorative techniques. J Prosthet Dent 1989;61(4):535-43.
9. İplikçioğlu H, Akça K. Comparative evaluation of diameter, length, and number of implants supporting three unit fixed partial prostheses on stress distribution in the bone. J Dent 2002;30(1):41-6.
10. Weinberg LA, Kruger B. A comparison of implant/prosthesis loading with four clinical variables. Int J Prosthodont 1995;8(5):421-33.
11. Rangert B, Krogh PH, Langer B, Van Roekel N. Bending overload and implant fracture: a retrospective clinical analysis. Int J Oral Maxillofac Implants 1995;10(3):326-34.
12. Geng PJ, Tan KBC, Liu GR. Application of finite element analysis in implant dentistry: A review of the literature. J Prosthet Dent 2001;85(4):585-98.
13. Hassler CR, Rybicki EF, Cummings KD, Clark LC. Quantification of bone stress during remodeling. J Biomechanics 1980;13(2):185-90.
14. Huiskes R, Chao EYS. A survey of finite element analysis in orthopedic biomechanics: The first decade. J Biomechanics 1983; 16(3):385–409.
15. Hojjatie B, Anusavice KS. Three dimensional finite element analyses of glass ceramic dental crowns. J Biomechanics 1990;23(10):1157-66.
16. Wheeler RC, Ash MM. An Atlas of Tooth Form. 4th ed. Philadelphia: WB Saunders Co; 1989. p.64-8.
17. Brunski JB, Puelo DA, Nanci A. Biomaterials and biomechanics of oral and maxillofacial implants: current status and future developments. Int J Oral Maxillofac Implants 2000;15(1):15--47.
18. Soumeire J, Dejou J. Shock absorbability of various restorative materials used on implants. J Oral Rehabil 1999;26(3):394–401.

19. Van Eijden TMGJ. Three dimensional analysis of human bite force magnitude and moment. Arch Oral Biol 1991;36(4):535-9.

20. Van Oosterwyck H, Duyck J, Vander Sloten J, Van der Perre G, De Cooman M, Lievens S, et al. The influence of bone mechanical properties and implant fixation upon bone loading around oral implants. Clin Oral Implants Res 1998;9(3):407-18.

21. Cochran DL. The scientific basis for clinical experiences with Straumann implants including the ITI dental implant system: A consensus report. Clin Oral Implant Res 2000;11(1):33-58.

22. Matsushita Y, Kitah M, Misuta K, Ikeda H, Suetsugu T. Two dimensional FEM analysis of hydroxyapatite implants: Diameter, effects on stress distribution. J Oral Implants 1990;16(1):6-11.

23. Demirbaş KA, Eyüpoğlu TF, Önal B. [Biomechanical evaluation of the restorationperformed with direct composite veneer]. Turkiye Klinikleri J Dental Sci 2004;10(1):5-10 .

24. Sertgöz A. Finite element analysis study of the effect of superstructure material on stress distribution in an implant supported fixed prosthesis. Int J Prosthodont 1997;10(1):19-27.

25. Eskitascioglu G, Usumez A, Sevimay M, Soykan E, Ünsal E. The influence of occlusal loading location on stresses transferred to implant-supported prostheses and supporting bone: A three-dimensional finite element study. J Prosthet Dent 2004;91(2):144-50.

26. Güngör MA, Küçük M, Dündar M, Karaoğlu C, Artunç C. Effect of temperature and stress distribution on all-ceramic restorations by using a three-dimensional finite element analysis. J Oral Rehabil 2004;31(2):172-8.

27. Şaklar F, Topbaş C. [Evaluation of stress distribution caused by traumatic forces applied on a maxillary central incisor]. Turkiye Klinikleri J Dental Sci 2000;6(2):115-9.

28. Weinberg LA. Reduction of implant loading with therapeutic biomechanics. Implant Dent 1998;7(4):277-85.