sciendo
10.2478/bjdm-2019-0023

BALKAN JOURNAL OF DENTAL MEDICINE BaSS ISSN 2335-0245

How Do Differences of Dental Implants' Internal Connection Systems Affect Stress Distribution? A 3-Dimensional Finite Element Analysis

SUMMARY

Background/Aim: *A factor affecting the success rate of dental implants, which has been used successfully for many years, is the implant-abutment connection system. The purpose of this study was to evaluate the stress distribution of different implant-abutment connection systems under different forces.* ***Material and Methods***: *This in vitro study included a finite element analysis. In the study, the cylindrical and screwed dental implants available in 3 different diameters from 4 different companies were categorized into 12 different models. Two different scenarios of force application were conducted on each model in this study. In the first scenario, 100 N force and 100 N moment were applied in a vertical direction onto a point considered as the center of each tooth. In the second scenario, a 100 N force and moment were applied at a 45° angle in an oblique direction.* ***Results***: *As a result of the forces applied to dental implants of different diameters from different companies, octagon implant-abutment connection systems had less stress accumulation than hexagon implant-abutment connection systems. In addition, when stress accumulation ratios were evaluated according to the diameter of the implants used, it was observed that 3 mm diameter implants accumulated more stress in bone than 4 mm diameter implants; there was no significant difference between 4 mm diameter implants and 5 mm diameter implants.* ***Conclusions***: *Implant-abutment connection system is important for the longevity of implants under the forces. Therefore, this factor should be considered during implant selection.*

Key words: Dental Implant, Stress Distribution, Finite Element Analysis, Implant-Abutment Connection

Özge Özdal Zincir[1], Gökay Karapinar[2], Meral Ünür[3], Ahmet Bülent Katiboğlu[1]

[1] Department of Oral and Maxillofacial Surgery, Faculty of Dentistry, Beykent University, Istanbul, Turkey
[2] Private Practice, Istanbul, Turkey
[3] Department of Oral and Maxillofacial Surgery, Faculty of Dentistry, Istanbul University, Istanbul, Turkey

ORIGINAL PAPER (OP)
Balk J Dent Med, 2019;126-131

Introduction

Dental implants were introduced in the late 1960s for rehabilitation of completely and partially edentulous patients, and since then the awareness and subsequent demand for this form of therapy have increased[1]. Over the past several decades, due to the reliable functional and aesthetic results, dental rehabilitation with implants has been widely accepted by dentists and patients. Conventionally, the implant/abutment interface is described as external or internal connection[2]. A review of existing literature has shown that amongst the cylindrical type of dental implants, the internal implant-abutment connections are greatly preferred. Therefore, the design of internal connection has gained importance in terms of the long-term success of load-bearing implants[3]. The finite element analysis method (FEM) used for implant biomechanical analysis offers many advantages over other methods for simulating the complexity of clinical conditions. It can be used to estimate the stress distribution in jaw bones and displacement of implant connections[4].

The aim of this study was to evaluate the resistance of internal connection systems of different dental implants with a cylindrical form against multiple forces, and the distribution of the applied forces on implant surfaces. Therefore, the negative and positive aspects of different internal connections could be evaluated in response to possible masticatory forces applied to the implant surface and bone.

Material and Methods

In this study, the cylindrical and screwed dental implants available in 3 different diameters from 4 different companies were categorized into 12 different models. For the purposes of adjusting the 3-dimension (3D) network structure and making it more homogeneous, generating the 3D solid model and the FEM analysis; Intel Xeon ® R CPU 3,30 GHz processor, 500 GB Hard disk, a computer equipped with 14 GB RAM and Windows 7 Ultimate Version Service Pack 1 operating system, Activity 880 (smart optics Sensortechnik GmbH, Bochum, Germany), optical scanner and 3D scanner, Rhinoceros 4.0 (Seattle, WA 98103 USA), 3D modelling software, VRMesh Studio (VirtualGrid Inc, Bellevue City, WA, USA), and Algor Fempro (ALGOR, Inc., PA 15238-2932 USA) analysis program were used.

After the models were created geometrically using the Rhinoceros software, they were transferred to the Algor Fempro (Algor Inc., USA) software and prepared for analysis.

All the materials were presumed to be linear elastic, homogenous, and isotropic. The corresponding elastic properties, such as Young's modulus and Poisson's ratio, were determined using values obtained in the literature[5,6], and are summarized in Table 1.

Table 1. Elastic properties of materials used in the analysis

Material	Young's modulus (MPa)	Poisson's ratio
Cortical Bone	13700	0,30
Cancellous Bone	1850	0,30
Titanium	117000	0,35

In order to analyze the model in space, it was necessary to connect it with peripheral points and define its limits.

In this study, all the models were given a zero-degree of freedom from the side and bottom surfaces of the bone tissues (Figure 1).

Two different scenarios of force application were applied on each model in this study. In the first scenario, 100 N force and 100 N moment were applied in a vertical direction onto a point considered as the center of each implant (Figure 2).

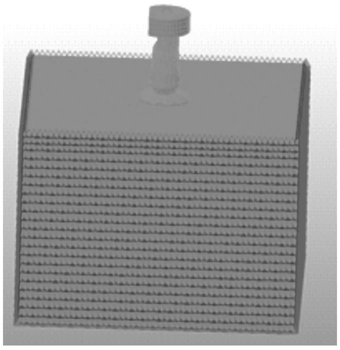


Figure 1. Defining the limit conditions

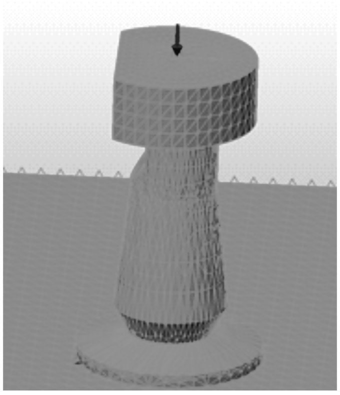


Figure 2. Loading in a vertical direction

In the second scenario, a 100 N force and moment were applied at a 45^{o} angle in an oblique direction (Figure 3).

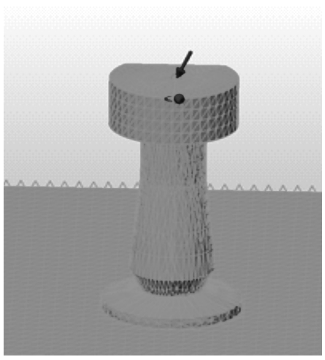


Figure 3. Loading in an oblique direction

Consequently, the stresses developed in the implants varied due to differences in their material and design, and these differences amongst the models were evaluated using the 3D FEM.

Results

The implants from different companies used in this study were categorized into 4 systems and analyzed.

The systems were designated as follows: System-1: First company (hexagon internal connection); System-2: Second company (octagon internal connection); System-3: Third company (octagon internal connection), and System-4: Fourth company (hexagon internal connection). Moreover, as 3 different models were created in each system, they were named and evaluated as follows: In System-1, the 3 mm diameter implant was model-1, the 4 mm diameter implant was model-2, the 5 mm diameter implant was model-3; in System-2, the 3 mm diameter implant was model-4, the 4 mm diameter implant was model-5, the 5 mm diameter implant was model-6; in System-3, the 3 mm diameter implant was model-7, the 4 mm diameter implant was model-8, the 5 mm diameter implant is model-9; in System-4, the 3 mm diameter implant was model-10; the 4 mm diameter implant was model-11, the 5 mm diameter implant was model-12.

In this study, a force of 100 N was applied in a vertical direction or at a 45º angle in an oblique direction from the same distance on all 12 models, and the degrees of stress on the connection systems of the implants and abutments were compared.

The stress accumulation on the implant-abutment connections of the 3 mm diameter dental implants are shown in figures 4-6. In both vertical and oblique forces, the maximum stress accumulation was observed in the model-10 and the least stress accumulation was observed in the model-7.

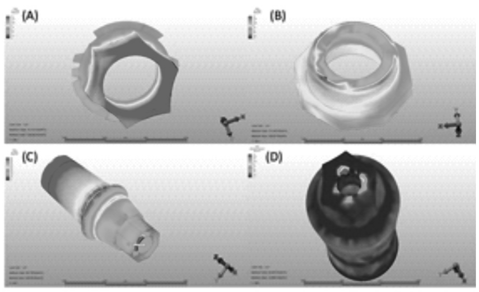


Figure 4. The degree of stress accumulation on the implant-abutment connection as a result of vertical force application on the 3mm diameter implant models. (A): Model-1, (B): Model-4, (C): Model-7, (D): Model-10

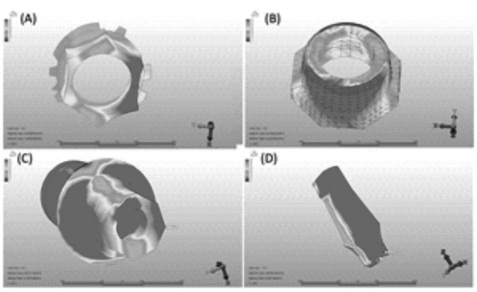


Figure 5. The degree of stress accumulation on the implant-abutment connection as a result of oblique force application on the 3mm diameter implant models. (A): Model-1, (B): Model-4, (C): Model-7, (D): Model-10

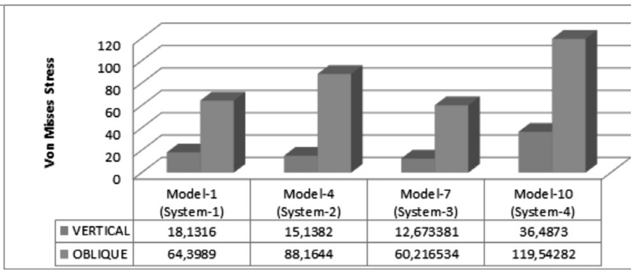


Figure 6. Maximum Von Misses stress degrees (MPa) measured on the implant-abutment connection points as a result of loading the 3 mm diameter implant models

The stress accumulation on the implant-abutment connections of the 4 mm diameter dental implants are shown in Figures 7-9. While the maximum stress accumulation was observed in the model-11 in both vertical and horizontal forces, the least stress accumulation was observed in the model-5 under vertical force and in the model-8 under the oblique force.

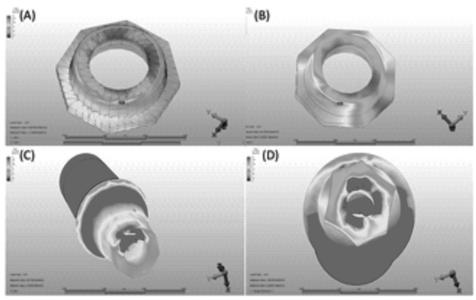


Figure 7. The degree of stress accumulation on the implant-abutment connection as a result of vertical force application on the 4mm diameter implant models. (A): Model-2, (B): Model-5, (C): Model: 8, (D): Model-11

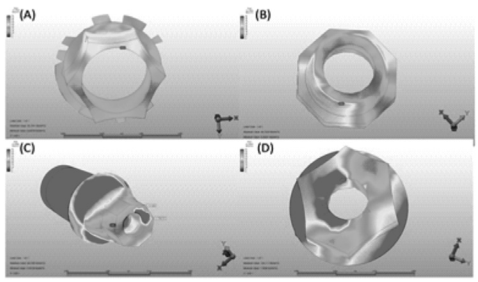


Figure 8. The degree of stress accumulation on the implant-abutment connection as a result of oblique force application on the 4mm diameter implant models. (A): Model-2, (B): Model-5, (C): Model: 8, (D): Model-11

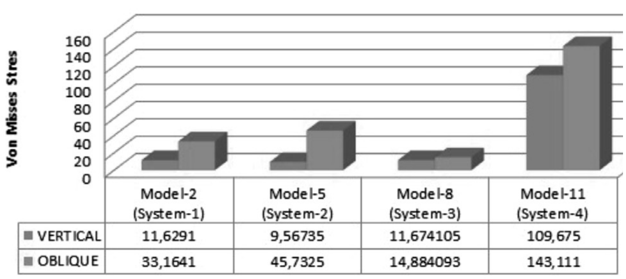


Figure 9. Maximum Von Misses stress degrees (MPa) measured on the implant-abutment connection points as a result of loading the 4 mm diameter implant models

The stress accumulation on the implant-abutment connections of the 5 mm diameter dental implants are shown in figures 10-12. While the maximum stress accumulation was observed in the model-12 in both vertical and horizontal forces, the least stress accumulation was observed in the model-6 under vertical force and in the model-9 under the oblique force.

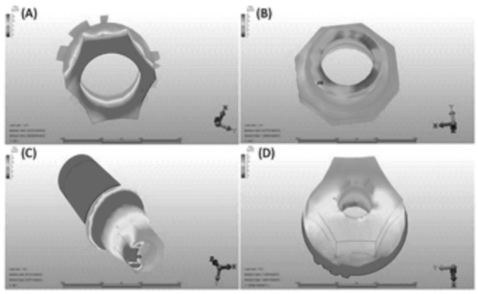


Figure 10. The degree of stress accumulation on the implant-abutment connection as a result of vertical force application on the 5mm diameter implant models. (A): Model-3, (B): Model-6, (C): Model: 9, (D): Model-12

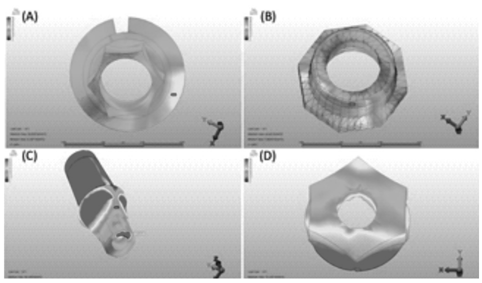
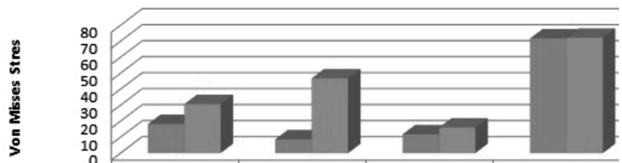


Figure 11. The degree of stress accumulation on the implant-abutment connection as a result of oblique force application on the 5mm diameter implant models. (A): Model-3, (B): Model-6, (C): Model: 9, (D): Model-12



	Model-3 (System-1)	Model-6 (System-2)	Model-9 (System-3)	Model-12 (System-4)
VERTICAL	18,1316	8,57197	11,688498	71,698
OBLIQUE	30,5935	46,682	15,934757	72,1291

Figure 12. Maximum Von Misses stress degrees (MPa) measured on the implant-abutment connection points as a result of loading the 5 mm diameter implant models

Discussion

The fundamental studies conducted by Branemark and Schroeder with pure titanium implants have been thought to herald scientific progress in the field of oral implantology[7,8]. Branemark observed a firm contact between the bone and titanium during a study in 1995, examining revascularization in rabbit tibias with vital microscopy, and investigated this finding in more detail. Branemark *et al.*[9] called this process "osseointegration" and defined it as "the direct contact between living bone tissue and titanium implant, which is observed under magnification using an optical microscope". The same researchers have further defined this process as "the direct structural and functional connection between living bone tissue and implant surface under loading"[10]. Swedish researchers contributed to dental literature by publishing two articles in 1969 and 1977, reporting their studies conducted with an aim to rehabilitate total edentulism cases with fixed dentures, and therefore brought the issue to a scientific level.

FEM is frequently used in order to provide information about the success of metal and ceramic materials. Therefore, the materials and methods used and applied ought to be life-like. The most important advantages of FEM are that it can be conducted in a very short period of time, can be repeated, and their parameters can be standardized[11].

The most important disadvantage of studies conducted with FEM is the difficulty of considering some factors as stable in order to imitate the natural structures, which can show significant changes in the mouth[12]. In this study, the elastic modulus of cortical and cancellous bone, and their Poisson ratios are assumed to be homogeneous in all regions during the modeling. However, when O'Mahony *et al.*[13] examined the mandibular cancellous bone, they emphasized that the bone is transversely isotropic, and that Young's modulus changes from one region to another. In most of the studies where the mechanical characteristics are evaluated, there are no accepted Poisson and Young's modulus values for bone. Therefore, in this study, the values suggested by Valle et al. which are accepted in most of the other studies, were used[11,14]. In addition to this, the implants are considered to be 100% osseointegrated, whereas the histological data shows that the bone-implant union is never 100%[15]. Therefore, similar to other previous studies, it should be accepted that FEM has some limitations in this study as well.

As single tooth implants have been increasingly used, the occurrence of complications such as screw loosening in the external implants and screw breakage have increased. Therefore, the manufacturers have developed connection types in the abutments, such as an internal hexagon, internal octagon, and morse taper, which are combined with implants under the marginal bone. In this type of connection, the abutment is placed into the space inside the implant with a very close (nearly 10 micron) contact. This close contact is very important in terms of preventing micro movements and vibrations.

In the systems with an internal connection, the lateral loads applied to the denture are concentrated on the connection interface and dispersed to all interfaces, thus preventing the forces from directly reaching the screw. The possibility of failure due to fatigue of a screw that is protected in this manner is decreased, since only 10% of the applied force is met by the screw[16].

In this study, it is found that when the stretching and compressing stress values that the dental implant systems with hexagon connection of 2 different companies and octagon connection systems of 2 other different companies from internal connection systems have created on the cortical and spongy bone are compared, there is no meaningful difference between the connection morphologies of both vertical and oblique loadings.

Since the types of internal connection of the dental implants are different from each other, it could lead to differing results such as those observed between the external and internal connections.

Balik *et al*.[17] conducted a study assessing and comparing 5 different implant-abutment connection systems used in the posterior region by means of 3D FEM. External hexagon, conical + internal hexagon, internal hexagon, tube-in-tube, and internal morse taper connection types were compared, and the lowest stress values were observed in the internal hexagon connection system.

Raoofi *et al.*[18] compared 3 different internal connection types by means of 3D FEM. The stress values in the internal hexagon, internal tri-channel, and internal morse taper connections were assessed under axial and non-axial forces. Amongst all models, the lowest stress values were observed in the internal tri-channel connection, and the highest stress values were observed in the internal morse taper connection under non-axial forces.

When the stress values were compared on the implant-abutment connection points of the systems used in this study, it has been found that the same internal implant-abutment connection types of different companies may differ in terms of stress accumulation under oblique and vertical forces. However, when the results of this study are evaluated as a whole, it is seen that stress accumulations under vertical forces were less in octagon internal implant-abutment connection systems. In addition, it is seen that stress accumulations under vertical forces were less in octagon internal implant-abutment connection systems.

In conclusion, in this study, stress distribution of the internal octagon connection and internal hexagon connection systems of different companies varied. Less stress accumulation is observed in internal octagon connection systems under oblique forces. In addition, as the diameter of the implant increased, this stress accumulation decreased. When stress accumulation ratios were compared between the systems in terms of changes in the diameters, the 3 mm diameter implants caused more stress accumulation in bone than the 4 mm diameter implants. However, the 4 mm diameter implants showed no meaningful difference compared to the 5 mm diameter implants in terms of the stress that they cause in bone.

All the materials and tissues used in this study were assumed to be homogeneous, isotropic, and linear elastic. This is a common assumption in most of the FEM studies. However, the biological characteristics in living tissues do not match these conditions absolutely. In future studies, non-linear FEM studies may be conducted by making use of anisotropic materials, so as to further elucidate the results of this study. It is strongly believed that all the results should be supported by future animal or clinical studies of similar quality as this study.

List of Abbreviations

- FEM: Finite Element Analysis Method
- mm: millimeter
- MPa: Mega Pascal = N/mm^2
- N: Newton
- 3D: 3-dimension
- GHz: GigaHertz
- GB: GigaByte

References

1. Adell R, Eriksson B, Lekholm U, Branemark PI, Jemt T. Long-term follow up study of osseointegrated implants in the treatment of totally edentulous jaws. Int J Oral Maxillofac Implants, 1990;5:347-359.
2. Kwon JH, Han CH, Kim SJ, Chang JS. The change of rotational freedom following different insertion torques in three implant systems with implant driver. J Adv Prosthodont, 2009;1:37-40.
3. Finger IM, Castellon P, Block M, Elian N. The evolution of external and internal implant/abutment connections. Pract Proced Aesthet Dent, 2003;15:625-632.
4. Marcián P, Wolff J, Horáčková L, Kaiser J, Zikmund T, Borák L. Micro finite element analysis of dental implants under different loading conditions. Comput Biol Med, 2018;96:157-165.
5. Matsushita Y, Kitoh M, Mizuta K, Ikeda H, Suetsugu T. Two-dimensional FEM analysis of hydroxyapatite implants: diameter effects on stress distribution. J Oral Implantol, 1990;16:6-11.
6. Peyton FA, Craig RG. Current evaluation of plastics in crown and bridge prosthesis. J Prosthet Dent, 1963;13:743-753.
7. Branemark PI, Breine U, Adell R, Hansson BO, Lindström J, Olsson A. Intraosseous anchorage of dental prostheses. I. Experimental studies. Scand J Plast Recons Surg, 1969;3:81-100.
8. Branemark PI, Hansson BO, Adell R, Breine U, Lindström J, Hallen O, Ohman A. Osseointegrated implants in the treatment of the edentulous jaw. Experience from a 10-year period. Scand J Plast Reconstr Surg, 1977;16:1-132.
9. Branemark PI, Zarb GA, Albrektson T. Tissue-integrated prostheses: Osseointegration in clinical dentistry. Chicago: Quintessence, 1985;54:611-612.

10. Zarb GA, Albrektson T. Osseointegration: A requiem for the periodontal ligament. Int J Periodontal Rest Dent, 1991;11:88-91.
11. Akça K, Çehreli MC, İplikçioğlu H. A comparison of three-dimensional finite element stress analysis with in vitro strain gauge measurements on dental implants. Int J Prosthodont, 2002;15:115-121.
12. Akça K, İplikçioğlu H. Finite element stress analysis of the influence of staggered versus straight placement of dental implants. Int J Oral Maxillofac Implants, 2001;16:722-730.
13. O'Mahony AM, Williams JL, Katz JQ, Spencer P. Anisotropic elastic properties of cancellous bone from a human edentulous mandible. Clin Oral Implant Res, 2000;11:415-421.
14. Bidez WM, Misch CE. Issues in bone mechanics related to oral implants. Implant Dent, 1992;1:289-294.
15. Van Zyl PP, Grundling NL, Jooste CH, Terblanche E. Three dimensional finite element model of a human mandible incorporating six osseointegrated implants for stress analysis of mandibular cantilever prosthesis. Int J Oral Maxillofac Implants, 1995;10:51-57.
16. Perriard J, Wiskott WA, Mellal A, Scherrer SS, Botsis J, Belser UC. Fatigue resistance of ITI implant-abutment connectors. A comparison of the standard cone with a novel internally keyed design. Clin Oral Implants Res, 2002;13:542-549.
17. Balik A, Karatas MO, Keskin H. Effects of different abutment connection designs on the stress distribution around five different implants: a 3-dimensional finite element analysis. J Oral Implantol, 2012;38:491-496.
18. Raoofi S, Khademi M, Amid R, Kadkhodazadeh M, Movahhedi MR. Comparison of the effect of three abutment-implant connections on stress distribution at the internal surface of dental implants: a finite element analysis. J Dent Res Dent Clin Dent Prospect, 2013;7:132-139.

Conflict of Interests: Nothing to declare.
Financial Disclosure Statement: Nothing to declare.
Human Rights Statement: None required.
Animal Rights Statement : None required.

Received on Jun 17, 2019.
Revised on July 2, 2019.
Accepted on July 3, 2019.

Correspondence:

Özge Özdal Zincir
Department of Oral and Maxillofacial Surgery
Faculty of Dentistry
Beykent University, Istanbul, Turkey
e-mail: ozgezincir@beykent.edu.tr