Comparative Evaluation of Three Abutment–Implant Interfaces on Stress Distribution in and Around Different Implant Systems: A Finite Element Analysis

Abstract

Aim: This study was conducted to compare and evaluate the effect of three abutment–implant connections on stress distribution around three different implants under similar material properties and loading condition using finite element analysis (FEA). Materials and Methods: Three different types of implant–abutment connections were selected. The features of these connections are Sample A: Tri-channel internal connection (Nobel Biocare); Sample B: Internal conical-hex Morse Taper (ADIN); and Sample C: Internal octa-Morse taper method (Osstem). The following softwares – ANSYS Version: 14.5 for FEA; Meshing software: Hypermesh 11; and CATIA: to produce computerized models of implants and for mandibular modeling were used. The implants were scanned with a high-quality scanner. All the above data were used to produce computerized models by CATIA software. Within the implant system, finite element method modeling was performed by implementing bonded conditions on the abutment–implant interfaces implementing four different load conditions. The computerized model was transferred to ANSYS software. A statistical analysis was done to compare the groups. Results: The samples were evaluated using three-dimensional FEA analysis. It was found that stress at 100 N, 100 N with 15° tilt, 300 N, and 300 N with 15° tilt was found to be highest in Sample A followed by Sample C and Sample B, and the difference was statistically insignificant. Conclusion: Within the limitations of the present study, the tri-channel internal connection showed maximum stresses and least by the internal conical-hex Morse Taper and internal octa-morse taper connection.

Keywords: Abutment–implant connections, ADIN, finite element analysis, Nobel Biocare, Osstem, stress distribution

Introduction

With continuous changes in the design of implant–abutment interface, the structural complexity has made it difficult to calculate occlusal forces and stresses in the bone around dental implant. Finite element analysis (FEA) has been applied to assess the mechanical characteristics of the implant–abutment connection in loading tooth and implant-supported prostheses. However, most of the previous reports lack rigorousness in model construction.¹

There is inadequate information regarding biomechanical comparison of different types of internal connections.² Hence, in this study, the effect of three abutment–implant connections on stress distribution around three different commercially available implants was compared using FEA.

Materials and Methods

Three different types of implant–abutment connections used in commercially well-known implant systems were selected. The features of these connections are as follows:
1. Sample A: Tri-channel internal connection (Nobel Biocare, Goteborg, Sweden) [Figure 1]
2. Sample B: Internal conical-hex Morse Taper (ADIN, Northern Israel) [Figure 2]
3. Sample C: Internal octa-Morse taper method (Osstem, Korea) [Figure 3].

Length of 10 mm and diameter of 4.3 mm were selected.

The above-mentioned implants were scanned using a high-quality scanner. Series of scans were taken from the inner surfaces of each type of implant. All the above data were used to produce...
Kharsan, et al.: Comparative evaluation of three abutment–implant interfaces on stress distribution in and around different implant systems: A finite element analysis

A computerized model by CATIA software (computer-aided three-dimensional [3D] interactive application, Dassault Systèmes, Vélizy-Villacoublay, France). Number of elements and nodes were tabularized [Table 1].

**Interface conditions**

The bone-implant interface was assumed to be perfect, simulating complete osseointegration. Therefore, the connections between implant cortical and implant cancellous bones were designed to be bonded as well as the interface between cancellous and cortical bones.

**Mesh generation**

The 3D finite element (FE) model corresponding to the geometric model was meshed using HyperMesh 11 software (Altair Engineering Inc., Troy, Michigan). The type of meshing is free meshing because the model is not geometrically symmetric. Tetrahedral and octahedral elements were meshed which were assigned 3° of freedom per node, namely, translation in the x, y, and z directions. The elements were constructed so that their size aspect ratio would yield reasonable solution accuracy.

**Specifying material properties**

All the materials to be used in the models consisted of implants, abutments, and abutment screws; compact and cancellous bone were presumed to be as homogeneous, isotropic, and linearly elastic as one another [Table 2].

**Applying boundary conditions**

Boundary conditions were set as fixed. Omitting support at the bottom permitted bending of the model. These aspects gave the model a more realistic representation of the clinical situation.[5,4]

**Application of loads**

The magnitude of applied loads was within physiologic limits. In studies conducted by Haraldson et al., patients treated with osseointegrated fixed prosthesis had a maximum biting force between 42 and 412 N.[5] In this study, four types of loading conditions were simulated:

- 100-N force applied directed axially to the abutment surfaces
- 100-N force directed at 15° to the long axis of the implant
- 300-N force directed axially to the abutment surfaces
- 300-N force applied at 15° to the long axis of the implant.

All the forces were applied to the entire outer surface of the abutment.

**Finite element analysis**

These different models were analyzed by Processor and the results were displayed by Post-Processor of the FE Software (ANSYS R14.5, Canonsburg, Pennsylvania, United States) in the form of color-coded maps using von Mises stress analysis. The von Mises stress values are defined as the beginning of deformation for ductile materials. Metallic implants failure occurs when von Mises stress values exceed the yield strength of an implant material. The von Mises stresses are most commonly reported in FEA studies to summarize the overall stress state at a point. The “equivalent stress of von Mises” is an expression that yields an effective absolute magnitude of stresses, taking into account principal stresses in three dimensions.

The von Mises stresses depend on the entire stress field and are widely used indicator for damage occurrence. Thus, von Mises stresses were chosen for the presentation of results. They are important for interpreting the stresses occurring within the implant material.[6] The type of stresses in FE studies are generally described by means of direction (shear, tension, and compression) or by an effective absolute

---

**Table 1: Number of elements and nodes**

| Details         | Elements | Nodes |
|-----------------|----------|-------|
| Nobel Biocare   | 446,841  | 83,097|
| Adin            | 454,892  | 85,682|
| Osstem          | 448,645  | 84,802|

**Table 2: Material properties**

| Material                     | Modulus of elasticity (MPa) | Poisson’s ratio |
|------------------------------|-----------------------------|-----------------|
| Cortical bone                | 14,000                      | 0.30            |
| Cancellous bone              | 620                         | 0.30            |
| Titanium                     | 102,000                     | 0.35            |
| Abutment-abutment screw[1]   | 11,400                      | 0.38            |

---

[1] Contemporary Clinical Dentistry | Volume 10 | Issue 4 | October-December 2019
magnitude of principal stresses (equivalent stress of von Mises). Stress distribution in the FE model comes in numerical values and in color coding. Maximum value of von Mises stress is denoted by red color. Minimum value of von Mises stress is denoted by blue color. The in-between values are represented by bluish green, green, greenish yellow, and yellowish red in the ascending order of stress distribution.

Results

The samples were evaluated for stress in the implant, abutment, at the implant–abutment interface, and overall stress values [Figures 4-7] using 3D FEA analysis. Stress at 100 N, 100 N with 15° tilt, 300 N, and 300 N with 15° tilt was found to be highest in Sample A followed by Sample C and Sample B, and the difference was statistically insignificant. Furthermore, the maximum values were seen in Sample A of 256.1 N/mm when 300-N forces with 15° tilt [Table 3].

Discussion

FEA is an approach to evaluate mechanical behavior in complex structures. The most prominent features of applying FEA are its repetitiveness, low costs, diversity in different analyses design, adaptability to various clinical conditions, high precision, lack of any need to sophisticated equipment, and also the possibility of design and analysis in individually custom-made implants. The operator can determine the stress and strain level numerically in any spot on the object of analysis. FEA can be performed at 2D or 3D.

Meijer et al. recommended that we should not use 2D models for stress analysis in implants. Because in the 2D analysis, the designed object is symmetrical geometrically. In asymmetrical objects or in cases when the force is angulated, the 3D analysis must be used. The present study was performed as a 3D FEA. The original implants were scanned and models were prepared. The structures in the model were all assumed to be homogeneous and isotropic and to possess linear elasticity.

Understanding the biomechanical aspects of different types of connections, (in particular the abutment–implant connection) and their effects on stress/strain fields in implant/bone systems may assist in reducing the risk of implant failure.

The majority of previous FEA studies have not modeled real implants. In our modeling, commercially available

| von Mises stress (Mpa) | Sample A | Sample B | Sample C |
|------------------------|----------|----------|----------|
| 100 N                  | 17.98    | 56.9     | 8.296    |
| 100 N at 15° tilt      | 30.63    | 85.44    | 12.85    |
| 300 N                  | 53.96    | 170.71   | 24.88    |
| 300 N at 15° tilt      | 91.82    | 256.1    | 38.5     |
| 300 N                  | 100 N    | 28.38    | 12.16    |
| 300 N at 15° tilt      | 36.23    | 61.43    | 34.92    |
| 300 N                  | 100 N    | 8.296    | 36.5     |
| 300 N at 15° tilt      | 108.69   | 85.15    | 104.67   |
| 300 N                  | 28.38    | 61.43    | 6.8      |
| 300 N at 15° tilt      | 108.69   | 85.15    | 24.71    |
| 300 N                  | 36.23    | 61.43    | 62.38    |

Table 3: Stress values using three-dimensional finite element analysis

Figure 4: Stress in Sample A, B, and C abutment and abutment–implant interface at 100 N

Figure 5: Stress in Sample A, B, and C abutment and abutment–implant interface at 100 N with 15° tilt
actual implants were scanned providing resemblances to real models as much as possible. Hence, through this study, a more logical selection of appropriate internal connection systems could be made possible.

The implant–abutment connection can be regarded as a key point to success.[9] Long-term clinical success requires decreased magnitude of stress distribution in the bone surrounding the dental implants. The conical internal hexagonal and screw-in implant–abutment connection designs provide more biomechanically suitable prosthetic options than other systems, particularly, in cases with increased vertical dimension in the posterior regions. The weakest point of dental endosseous implant fixtures is the implant–abutment connection; hence, it must resist maximal and permanent masticatory forces as well as penetration by bacteria.[9] Quaresma et al., in their study, stated lower stresses in bone in the conical group compared to the internal hexagonal group. In addition, Lin et al. (2000) and Hansson et al. (2007) reported that conical implant–abutment connection systems performed better as a force transmission system than internal hexagonal and external flat top systems. There is an increasing trend in the stress values with increase in forces applied in all the three samples, with a significant increase in the stress in Sample B, whereas similar stresses are observed in the Sample A and C with no significant difference. Yamashi et al., in their study, compared external hex, internal cone, internal straight connections, and effect of implant–abutment design on abutment micromovement, implant–abutment interface, and peri-implant stress showed largest amount of abutment movement, higher labial bone stresses, whereas internal conical connection had lowest abutment movement and low labial pericoronal bone stresses. The comparison between the internal cone, trilobe, internal hex, internal octagon implant–abutment connections on micromotion, and abutment stress distribution showed stress concentration at vertices of nonconical abutments; conical abutments showed more uniformly distributed stresses; internal hex connection showed the greatest stresses, followed by internal conical, octagonal, and the trilobed connection. In our study also, tri-channel internal connection (Nobel Biocare, Goteborg, Sweden) showed maximum stresses and least by the internal conical-hex Morse Taper (ADIN, Northern Israel) and internal octa-morse taper method (Osstem, Korea). Pellizer et al. in their study compared the internal hex, external hex, internal octagon with cone, internal cone, internal locking taper Conexao Implant system, ITI (Strautmann) and Bicon connections for their strain/stress distribution around implants under vertical and oblique loads. Greatest stress concentration was seen in the cervical and apical thirds. Internal octagon + cone presented the lowest stress concentrations; external hex exhibited the greatest stresses, which were in accordance with our study. Nishioka compared internal cone, internal hex, and external hex Conexao Implant System (ConexaoSystemas de Proteese) for strain/stress distribution around implants and effect of implant–abutment connection and implant fixture alignment and found statistically significant difference comparing the implant–abutment connections. Morse taper and internal hexagon did not reduce strain around implants, no statistical significance in the placement configuration was observed. However, these findings were contrasting to the present study results where the three different internal connection designs showed no statistical significant difference in the stress/strain distribution. Although the stress distribution was highest at larger loads applied and in the oblique direction for all the three implants, it was maximum in the tri-channel internal connection with statistically significant difference from the other two connections. The lowest stress was seen with the internal octa-Morse taper (Ostem).
implant–abutment connection. The only variation is seen at the 300N with 15° tilt where the least stress is seen with conical-hex Morse Taper with similar values with the internal octa-Morse taper implant–abutment connection.

It should be recognized that it is not only the geometry of the implant–abutment interface that might influence abutment fracture resistance but also other components and design factors as well. These could include the number of components (one-piece or two-piece abutment connections), screw length and diameter, thread design, material, as well as contact area.

**Conclusion and Drawbacks**

In the present study, only the geometry of the interface was assessed, further studies considering other aspects and factors also need to be done and further evaluated. Furthermore, the natural structures simulated in the FEA model were considered to be homogenous which may not be the case in clinical conditions and the results may vary. For instance, it is well described that cortical bone of the mandible is transversely isotropic and inhomogeneous. In addition, 100% implant–bone interface was established, which does not necessarily simulate clinical situations. Thus, the inherent limitations in this study should be considered. Within the limitations of the present study, the abutment-implant tri-channel internal connection has maximum stress distribution in and around implant at different forces applied as compared to internal conical-hex Morse Taper and the least stress distribution in the internal octa - morse taper connection.

**Financial support and sponsorship**

Nil.

**Conflicts of interest**

There are no conflicts of interest.

**References**

1. Tang CB, Liu SY, Zhou GX, Yu JH, Zhang GD, Bao YD, Wang QJ. Nonlinear finite element analysis of three implant-abutment interface designs. Int J Oral Sci. 2012;4:101-8.
2. Raoofi S, Khademi M, Amid R, Kadkhodazadeh M, Movahedi MV. 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;7:132-9.
3. Ritz W, Uber E. In eneue method ezur Losungge wissen variations- Problem eder mathematic chemphysik. J Reine Angew Math 1909;135:1-61.
4. Meijer HJA, Kuiper JH, Starmans FJM, Bosman F. Stress distribution around dental implants: Influence of superstructure, length of implants and height of mandible. J Prosthodont 1992;6:86-92.
5. Cehreli MC, Inlikioglu H. In vitro strain gauge analysis of axial and off-axial loading on implants supported fixed partial dentures. Implant Dentistry 2002;11:286-90.
6. Lu Wiskott HW, Belser UC. Lack of integration of smooth titanium surfaces: A working hypothesis based on strains generated in the surrounding bone. Clin Oral Implants Res 1999;10:429-44.
7. Kenney R, Richards MW. Photoelastic stress patterns produced by implant-retained overdentures. J Prosthodont 1998;80:559-64.
8. Bozkaya D, Muftu S, Muftu A. Evaluation of load transfer characteristics of five different implants in compact bone at different load levels by finite elements analysis. J Prosthodont. 2004;9:523-30.
9. Schmitt C, Nogueira-Filho G, Tenenbaum HC, Lai JY, Brito C, Hendrik D, et al. Performance of conical abutment (Morse Taper) connection implants: A systematic review. J Biomed Materials Res Part A 2014;102:552-74.
10. Quaresma SE, Cury PR, Sendyk WR, Sendyk C. A finite element analysis of two different dental implants: Stress distribution in the prosthesis, abutment, implant, and supporting bone. J Oral Implantol 2008;34:1-6.
11. Lin CL, Chang SH, Chang WJ, Kuo YC. Factorial analysis of variables influencing mechanical characteristics of a single tooth implant placed in the maxilla using finite element analysis and the statistics-based Taguchi method. Eur J Oral Sci 2007;115:408-16.
12. Hansson S. A conical implant-abutment interface at the level of the marginal bone improves the distribution of stresses in the supporting bone. An axisymmetric finite element analysis. Clin Oral Implants Res 2003;14:286-93.
13. Yamanishi Y, Yamaguchi S, Imazato S, Nakano T, Yatani H. Influences of implant neck design and implant-abutment joint type on peri-implant bone stress and abutment micromovement: Three-dimensional finite element analysis. Dent Mater 2012;28:1126-33.
14. Pellizzer EP, Carli RI, Falcón-Antenucci RM, Verri FR, Goiatto MC, Villa LM, et al. Photoelastic analysis of stress distribution with different implant systems. J Oral Implantol 2014;40:117-22.
15. Nishioka RS, de Vasconcellos LG, de Melo Nishioka GN. Comparative strain gauge analysis of external and internal hexagon, Morse taper, and influence of straight and offset implant configuration. Implant Dent 2011;20:e24-32.