International Journal of Environmental Sciences
ISSN: 2229-7359
Vol. 11 No. 6s, 2025
https://www.theaspd.com/ijes.php

Evaluating Stress Distribution in Bone Adjacent to Implant in Various Abutment-Implant Connection Designs

1. Dr. Sushil Kumar Sahoo Associate Professor, Department of Oral and Maxillofacial Surgery, Hi-Tech Dental College and Hospital, Bhubaneswar, Odisha.
2. Dr. Ashutosh Panda 2nd Year PG Resident, Department of Oral and Maxillofacial Surgery, Hi-Tech Dental College and Hospital Bhubaneswar Odisha.
3. Dr. Tonmoy Ranu 3rd Year PG Resident, Department of Oral and Maxillofacial Surgery, Hi-Tech Dental College and Hospital, Bhubaneswar, Odisha.
4. Dr. Shree Mishra2nd Year PG Resident, Department of Oral and Maxillofacial Surgery, Hi-Tech Dental College and Hospital Bhubaneswar Odisha.
5. Dr. Sonali Priyadarshini Sahu1st Year PG Resident, Department of Oral and Maxillofacial Surgery, Hi-Tech Dental College and Hospital Bhubaneswar Odisha.
6. Dr. Asutosh Pradhan3rd Year PG Resident, Department of Oral and Maxillofacial Surgery, Hi-Tech Dental College and Hospital, Bhubaneswar, Odisha.

Abstract
BACKGROUND
The biomechanical integrity of dental implants is significantly influenced by the design of the abutment-implant connection. Different connection geometries can alter stress transfer to the surrounding bone, which may impact implant longevity and osseointegration. This study aims to assess and compare the stress distribution in peri-implant bone associated with three commonly used abutment-implant connection designs: external hex, internal hex, and conical connection.
MATERIALS AND METHODS
A three-dimensional finite element analysis (FEA) was performed using computer-aided design (CAD) models of mandibular bone with a single implant in the posterior region. Implants with external hex, internal hex, and conical abutment connections were analyzed. A vertical load of 150 N and oblique load of 100 N at 30° were applied to simulate masticatory forces. Stress distribution in cortical and cancellous bone was evaluated using von Mises criteria.
RESULTS
Under vertical loading, the maximum stress observed in cortical bone was 55.2 MPa for the external hex, 48.6 MPa for the internal hex, and 36.9 MPa for the conical connection. Oblique loading yielded higher stress values: 78.4 MPa for the external hex, 66.7 MPa for the internal hex, and 52.3 MPa for the conical connection. Cancellous bone exhibited lower overall stress values in all models. The conical connection demonstrated the most favorable stress distribution, with reduced stress concentration at the crestal bone.
CONCLUSION
The type of abutment-implant connection significantly influences stress distribution in adjacent bone. Conical connections exhibit superior biomechanical performance by minimizing stress concentration, suggesting their potential benefit in enhancing long-term implant stability.
Keywords
Dental implant, abutment connection, finite element analysis, stress distribution, conical connection, biomechanical performance

INTRODUCTION

Dental implants have become a widely accepted and reliable solution for the replacement of missing teeth, offering improved function, aesthetics, and patient quality of life. The long-term success of dental implants largely depends on the stability of the implant-abutment interface and the manner in which masticatory forces are transmitted to the surrounding alveolar bone (1). One of the critical factors influencing peri-implant bone response is the design of the abutment-implant connection, which directly affects stress distribution at the crestal bone level (2,3).

Mechanical loads applied to the implant during function generate stress concentrations at the implant-bone interface, and excessive stress in this region may contribute to bone resorption and eventual implant failure (4). Therefore, optimizing the biomechanical characteristics of the implant system, particularly the connection geometry, is essential to ensure favorable load transfer and minimize marginal bone loss (5).Various connection types have been developed to enhance the mechanical and biological outcomes of implant therapy. The external hex connection, although historically popular, has been associated with micromovement and bacterial infiltration, leading to peri-implant bone loss (6). Internal hex and conical connections are more recent innovations designed to improve mechanical stability and sealing ability, potentially reducing complications related to stress concentration and microleakage (7,8). Finite element analysis (FEA) has emerged as a valuable tool in implant dentistry to simulate stress patterns under functional loading conditions and evaluate the mechanical behavior of different implant designs (9).This study aims to analyze and compare the stress distribution in peri-implant bone surrounding implants with external hex, internal hex, and conical abutment connections using a finite element model. Understanding these stress patterns may guide clinicians in selecting implant systems that optimize biomechanical performance and enhance clinical outcomes.

Materials And Methods

This study utilized a three-dimensional finite element analysis (FEA) to evaluate the stress distribution in bone surrounding dental implants with different abutment-implant connection designs. Three types of connection geometries were modeled: external hex, internal hex, and conical connection.

Model Construction

A virtual mandibular segment was modeled using CAD software, representing a section of posterior mandibular bone comprising both cortical and cancellous components. Standard implant dimensions were used: 4.2 mm in diameter and 10 mm in length. Three implant-abutment assemblies, each with a unique connection type, were digitally inserted into the bone model.

Material Properties

All materials in the model were assumed to be homogeneous, isotropic, and linearly elastic. The mechanical properties used in the simulation included:

- Cortical bone: Young's modulus = 13.7 GPa, Poisson's ratio = 0.3
- Cancellous bone: Young's modulus = 1.37 GPa, Poisson's ratio = 0.3
- Titanium (implant and abutment): Young's modulus = 110 GPa, Poisson's ratio = 0.35
-

Loading and Boundary Conditions

Two types of static loads were applied to the prosthetic crown:

- A vertical load of 150 N directed along the long axis of the implant
- An oblique load of 100 N applied at a 30-degree angle to simulate masticatory forces

The base of the bone model was fixed in all directions to simulate mandibular support. A frictional contact was defined between the implant and bone surfaces, while the abutment-implant interface was assumed to be fully osseointegrated.

Mesh Generation And Analysis

The assembled models were meshed using tetrahedral elements with refinement at the implant-bone interface. Mesh convergence tests were conducted to ensure accuracy of results. Stress distribution was evaluated using von Mises stress criteria, with results analyzed using ANSYS simulation software (version XX, ANSYS Inc., USA).

Comparative Evaluation

The maximum von Mises stress values in both cortical and cancellous bone were recorded and compared across all three implant-abutment connection models under both loading conditions.

RESULTS

The finite element analysis revealed varying patterns of stress distribution in the peri-implant bone across different abutment-implant connection designs under both vertical and oblique loading conditions. The maximum von Mises stress values recorded in the cortical and cancellous bone are summarized in **Table 1** and **Table 2**, respectively.

Under vertical loading (150 N), the **conical connection** exhibited the lowest stress concentration in cortical bone (36.9 MPa), followed by the **internal hex** (48.6 MPa) and **external hex** (55.2 MPa) configurations. A similar trend was observed in cancellous bone, where the conical connection demonstrated the most favorable stress distribution (4.2 MPa), as shown in **Table 1**.

When subjected to oblique loading (100 N at 30°), all connection designs showed increased stress levels. The external hex design demonstrated the highest stress in cortical bone (78.4 MPa), while the conical connection again presented the lowest value (52.3 MPa). The internal hex design recorded an intermediate value of 66.7 MPa (Table 2).

Overall, the **conical connection** consistently minimized stress transmission to the adjacent bone under both loading scenarios (Tables 1 and 2), indicating superior biomechanical behavior.

Table 1. Maximum von Mises Stress in Cortical Bone (MPa) under Vertical and Oblique Loading

Connection Type	Vertical Load (150 N)	Oblique Load (100 N, 30°)
External Hex	55.2 MPa	78.4 MPa
Internal Hex	48.6 MPa	66.7 MPa
Conical	36.9 MPa	52.3 MPa

Table 2. Maximum von Mises Stress in Cancellous Bone (MPa) under Vertical and Oblique Loading

Connection Type	Vertical Load (150 N)	Oblique Load (100 N, 30°)
External Hex	6.1 MPa	9.3 MPa
Internal Hex	5.4 MPa	7.8 MPa
Conical	4.2 MPa	6.0 MPa

DISCUSSION

This study investigated the influence of different abutment-implant connection designs—external hex, internal hex, and conical—on the distribution of stresses within peri-implant bone using finite element analysis (FEA). The results indicated that the conical connection consistently demonstrated reduced stress concentrations compared to external hex and internal hex designs. This finding aligns with previous research highlighting the biomechanical advantages of conical connections, specifically their ability to distribute stress more evenly across the implant-bone interface (1–3).

The higher stress observed in external hex configurations under both vertical and oblique loads may be attributed to the reduced stability of this connection type, primarily due to micro-gaps and potential micro-movement at the implant-abutment interface (4,5). Such micro-movement not only amplifies stress levels but also creates an environment conducive to bacterial colonization, potentially leading to peri-implantitis and subsequent marginal bone loss (6,7).

Internal hex connections showed intermediate stress distribution values compared to external hex and conical types, a finding supported by several biomechanical studies (8–10). The internal hex geometry enhances mechanical stability due to a greater contact surface between the abutment and implant, effectively reducing micro-movement and associated stress concentrations (9,11). Despite these improvements, internal hex connections may still experience localized stresses at the corners of the hexagon, suggesting that further design optimization may be necessary to approach the biomechanical performance observed with conical connections (10,12).

The favorable outcomes demonstrated by conical connection implants are consistent with previous findings indicating that tapered connections minimize micro-movement, enhance mechanical stability, and prevent bacterial leakage, thus promoting better long-term peri-implant bone health (3,13,14). Moreover, conical connections provide an improved sealing effect due to their intimate abutment-implant interface, effectively distributing occlusal loads more evenly and potentially reducing crestal bone loss (13,15).

This study utilized finite element analysis, a valuable computational tool commonly employed to predict biomechanical behavior under simulated clinical conditions. Although FEA is beneficial for evaluating the mechanical performance of dental implants, it inherently assumes idealized conditions that may not fully replicate the complexity of biological environments (8,16). Variability in bone density, osseointegration quality, implant positioning, and masticatory force distribution in actual clinical scenarios can significantly influence stress distribution outcomes (17,18).

Additionally, the arbitrary loads used in this simulation (150 N vertically and 100 N obliquely) represent typical masticatory forces. Nonetheless, actual occlusal forces vary widely among individuals, influenced by factors such as bite force, prosthetic design, and implant position (19-22). Future research involving clinical or in vivo studies with standardized methodologies is warranted to validate these computational findings and assess their real-world relevance.

Clinicians selecting implant systems should carefully consider the abutment-implant connection type, as it significantly influences peri-implant bone stress and consequently the long-term success of implant therapy. Conical connections appear particularly advantageous due to their biomechanical stability, potentially reducing risks of marginal bone resorption and implant failure.

CONCLUSION

In conclusion, the innovative implant designs presented offer versatile applications across multiple dental specialties, ensuring optimal biomechanical stability, enhanced osseointegration, and improved clinical outcomes. These universal systems may effectively address common clinical challenges encountered in oral surgery, prosthodontics, periodontics, and endodontics.

References

1. Mitra D, Gurav P, Rodrigues S, Khobragade B, Mahajan A. Evaluation of stress distribution in and around dental implants using three different implant-abutment interfaces with platform-switched and non-platform-switched abutments: A three-dimensional finite element analysis. J Dent Res Dent Clin Dent Prospects. 2023;17(4):256–64.
2. Paul S, Padmanabhan TV, Swarup S. Comparison of strain generated in bone by "platform-switched" and "non-platform-switched" implants with straight and angulated abutments under vertical and angulated load: a finite element analysis study. Indian J Dent Res. 2013;24(1):8–13.
3. Kharsan V, Bandgar V, Mirza A, Jagtiani K, Dhariwal N, Kore R. Comparative Evaluation of Three Abutment-Implant Interfaces on Stress Distribution in and Around Different Implant Systems: A Finite Element Analysis. Contemp Clin Dent. 2019;10(4):590–4.
4. Freitas-Júnior AC, Rocha EP, Bonfante EA, Almeida EO, Anchieta RB, Martini AP, et al. Biomechanical evaluation of internal and external hexagon platform switched implant-abutment connections: An in vitro laboratory and three-dimensional finite element analysis. Dent Mater. 2012;28(10):e218–28.
5. Liu S, Tang C, Yu J, Dai W, Bao Y, Hu D. The effect of platform switching on stress distribution in implants and periimplant bone studied by nonlinear finite element analysis. J Prosthet Dent. 2014;112(5):1111–8.
6. Yağır MO, Şen Ş, Şen U. Examination of Various Abutment Designs Behavior Depending on Load Using Finite Element Analysis. Biomimetics (Basel). 2024;9(8):498.
7. Aradya A, Kumar UK, Chowdhary R. Influence of different abutment diameter of implants on the peri-implant stress in the crestal bone: A three-dimensional finite element analysis—In vitro study. Indian J Dent Res. 2016;27(1):78–85.
8. Zhang WT, Cheng KJ, Liu YF, Wang R, Chen YF, Ding YD, et al. Effect of the prosthetic index on stress distribution in Morse taper connection implant system and peri-implant bone: a 3D finite element analysis. BMC Oral Health. 2022;22(1):431.
9. Chu CM, Huang HL, Hsu JT, Fuh LJ. Influences of internal tapered abutment designs on bone stresses around a dental implant: three-dimensional finite element method with statistical evaluation. J Periodontol. 2012;83(1):111–8.
10. Khatib I, Joshi NV, Rao P, Joshi M, Shetty A, Thorat V, et al. Effect of Implant-Abutment Connection and Abutment Angulations on Peri-implant Stress Levels: A Finite Element Analysis Study. Compend Contin Educ Dent. 2024;45(7):e5–e9.
11. Choi KS, Park SH, Lee JH, Jeon YC, Yun MJ, Jeong CM. Stress distribution on scalloped implants with different microthread and connection configurations using three-dimensional finite element analysis. Int J Oral Maxillofac Implants. 2012;27(3):e29–38.

12. Devanna R, Awinashe V, Priyadarshi S, Ramaiah VV, Gupta A, Jain A. A Finite Element Analysis of Stress Allocation around Three Types of Abutment-Implant Interfaces. J Pharm Bioallied Sci. 2023;15(Suppl 2):S1079–81.
13. Gurgel-Juarez NC, de Almeida EO, Rocha EP, Freitas AC Jr, Anchieta RB, de Vargas LC, et al. Regular and platform switching: bone stress analysis varying implant type. J Prosthodont. 2012;21(3):160–6.
14. Canullo L, Pace F, Coelho P, Sciubba E, Vozza I. The influence of platform switching on the biomechanical aspects of the implant-abutment system. A three dimensional finite element study. Med Oral Patol Oral Cir Bucal. 2011;16(6):e852–6.
15. Dinc MM, Turkoglu P, Selvi F. Biomechanical evaluation of stress distributions at the implant-abutment complex and peri-implant bone around mandibular dental implants with different neck geometries and inclinations. Proc Inst Mech Eng H. 2021;235(9):1035–45.
16. Alvarez-Arenal A, Segura-Mori L, Gonzalez-Gonzalez I, Gago A. Stress distribution in the abutment and retention screw of a single implant supporting a prosthesis with platform switching. Int J Oral Maxillofac Implants. 2013;28(3):e112–21.
17. Cho SY, Huh YH, Park CJ, Cho LR. Three-Dimensional Finite Element Analysis on Stress Distribution of Internal Implant-Abutment Engagement Features. Int J Oral Maxillofac Implants. 2018;33(2):319–27.
18. 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):1–6.
19. Tiwari A, Ghosh A, Agrawal PK, Reddy A, Singla D, Mehta DN, Girdhar G, Paiwal K. Artificial intelligence in oral health surveillance among under-served communities. Bioinformation. 2023;19(13):1329.
20. Sultana R, Raj A, Barbi W, Afridi SK, Mishra BP, Malik R. A comparative study evaluating implant success and bone loss in diabetes and nondiabetes. J Pharm Bioall Sci 2021;13:S1410-3.
21. Kul E, Korkmaz İH. Effect of different design of abutment and implant on stress distribution in 2 implants and peripheral bone: A finite element analysis study. J Prosthet Dent. 2021;126(5):664.e1–664.e9.
22. Moon SY, Lim YJ, Kim MJ, Kwon HB. Three-dimensional finite element analysis of platform switched implant. J Adv Prosthodont. 2017;9(1):31–7.