ISSN: 2581-5989
PubMed - National Library of Medicine - ID: 101738774

International Journal of Dental Science and Innovative Research (IJDSIR)

IJDSIR : Dental Publication Service

Available Online at:www.ijdsir.com

Volume – 9, Issue – 3, June – 2026, Page No. : 136 – 147

Comparative evaluation of two different types of implant abutment connections on stress distribution pattern using a three dimensional finite element analysis

[1]Dr. Inayat Kaur Duggal, Senior Resident, Department of Prosthodontics, Bhojia Dental College and Hospital, Baddi, Himachal Pradesh, India

[2]Dr. Tarun Kalra, Principal, Professor and Head, Department of Prosthodontics, Bhojia Dental College and Hospital, Baddi, Himachal Pradesh

[3]Dr. Manjit Kumar, Professor, Department of Prosthodontics, Bhojia Dental College and Hospital, Baddi, Himachal Pradesh, India

[4]Dr Usha Devi, Reader, Department of Microbiology, Bhojia Dental College and Hospital, Baddi, Himachal Pradesh, India

Corresponding Author: Dr Tarun Kalra, Principal, Professor and Head, Department of Prosthodontics, Bhojia Dental College and Hospital, Baddi, Himachal Pradesh, India.173205, PhD Scholar MMCODSR MMU, Mullana, Haryana.

Citation of this Article: Dr. Inayat Kaur Duggal, Dr. Tarun Kalra, Dr. Manjit Kumar, Dr Usha Devi, "Comparative evaluation of two different types of implant abutment connections on stress distribution pattern using a three dimensional finite element analysis", IJDSIR – June – 2026, Volume – 9, Issue – 3, P. No. 136 – 147.

Type of Publication: Original Research Article

Conflicts of Interest: Nil

Abstract

The implant-abutment interface plays a crucial role in dictating the lateral and rotational stability of the implant-abutment joint, thereby influencing the overall prosthetic stability of implant-supported restorations. Some connections demonstrate superior prosthesis retention, heightened stability and diminished stress on the implants. It is noteworthy that various prosthetic platforms, when employed with strict adherence to specific indications and limitations, can yield high success rates and help contribute to the overall efficacy of the procedure. This study aimed to evaluate and compare the stress distribution pattern of two different types of implant abutment connections using a three dimensional finite element analysis.

Keywords: Implant abutment connection, Internal hex, Conical hex, Stress distribution & FEA

Introduction

The history of replacing missing teeth with artificial substitutes is a rich and intriguing travelogue spanning centuries. Over the years, the materials used to replace missing teeth have evolved from early materials such as gold ligature wire, seashells, and ivory, to modern metals like chromium, cobalt, iridium, and platinum. The remarkable progress in human creativity and persistence in the field of research and innovation reflects humanity's continuous effort to replace missing teeth. From ancient times to the modern era, humans have sought ways to restore lost teeth, and over the years, these attempts have evolved into the highly advanced dental implant procedures we have today[1].

The narrative is one of continual innovation, where advancements in technology serve as the guiding notes, building a future where dental implants stand as a beacon of hope for those seeking a prosthetic alternative to missing teeth.

An important component of dental implants is the abutment. According to GPT10, a dental implant abutment is formally defined as “that portion of a dental implant that serves to support and/or retain a prosthesis”[3]. The connection between the implant and abutment stands as a crucial synapse that significantly impacts the strength and stability of an implant-supported restoration. This connection, often overlooked in its complexity, plays a central role in the overall success of the implant. The biomechanics of this interface are multifaceted, involving intricate factors that influence the load distribution, stress concentration, and overall mechanical performance of the restoration. This connection, often underestimated in its significance, serves as acornerstone for the overall success of the implant. In clinical practice, adetailed understanding of implant abutment connections is indispensable for effective treatment planning and execution[4].

Aim

The aim of the present *in-vitro* study was the Comparative evaluation of two different types of implant abutment connections on stress distribution pattern using a three dimensional finite element analysis.

Objectives

1. To evaluate stress distribution pattern of internal hex implant abutment connection under vertical loading.
2. To evaluate stress distribution pattern of internal hex implant abutment connection under oblique loading.
3. To evaluate stress distribution pattern of conical hex implant abutment connection under vertical loading.
4. To evaluate stress distribution pattern of conical hex implant abutment connection under oblique loading.
5. To Compare the stress distribution pattern of the two different types of implant abutment connections.

Materials and Method

1. **Computer with minimum specifications:** Windows 10, Intel core i3 processor, 64 GB RAM, 1 TB Hard Disk.
2. **FEA Software:** The models were developed and analysed using Ansys 2023 3D finite element modeling software, Ansys Inc., TX, USA.
3. Cone beam computed tomography
4. Cadaveric mandible.
5. Implant-Abutment-Prosthesis assembly with Internal Hex Implant Abutment Connection.
6. Implant-Abutment-Prosthesis assembly with Conical Hex Implant Abutment Connection.

The study was done in following steps:

1. Generation of STL format of mandible.
2. Transfer of STL format to ANSYS software to generate FEA model via Space Claim software.
3. Modeling of mandible for the study.
4. Modeling of Implant-Abutment-Prosthesis Assembly with Internal Hex Implant Abutment Connection.
5. Modeling of Implant-Abutment-Prosthesis Assembly with Conical Hex Implant Abutment Connection.
6. Stress analysis.

3-D Finite Element (FE) Model Generation

Although, this method can be achieved with a two dimensional (2-D) model, the 2-D model does not simulate the *in vivo* condition in many ways and also may be over simplified for representing a given solution. Therefore, a three dimensional (3-D) model was generated for this study. The first step of modeling the implant, abutment and prosthesis was to prepare an editable geometry in computer. A 3D geometry of mandible was generated. Then the data was imported into Solid Works software to create three dimensional (3-D) models of mandible. Then mandibular segment with implants and superstructures were modeled. All the materials were considered to be isotropic, homogenous and linearly elastic. The model was meshed and analysed by ANSYS Workbench. A 3-D finite bone model was generated on a computer using Space Claim software. The model received two implants, where one implant was placed on each side of the mandible in the first mandibular molar region. Modeling of the Implant, Abutment and Prosthesis with Internal Hex and Conical Hex implant abutment connection geometry was then done (Fig.1,2,3). All the components were individually modeled with having different physical properties[5](Table 1,2,3,4 &5) and assembled (Fig. 3). The entire assembly was then exported for analysis with ANSYS Workbench. Depending on the hardness of foods, the average bite force reportedly ranges from 20 to 120N[6]. In the present study, we applied a static load of 100N vertical load and 100N oblique load at 45 degrees to simulate loading by occlusion as indicated in some previous studies[7,8,9].

This was done to simulate realistic chewing pattern. For a direct and systematic comparison, the same loading conditions, boundary conditions and constraints were applied in both implant abutment connection models. After application of force the magnitude was analysed by ANSYS Workbench, a 3-D finite element analysis program. An assessment of the stress on the assembly, implant-abutment interface and bone-implant interface was performed by using Von Mises stresses. A color scale with stress values was used to evaluate quantitatively the stress distribution on the assembly, implant-abutment interface, and bone-implant interface. The scale for stress runs from 0MPa (blue) to the highest stress values (red). Red indicates areas with highest stress, and blue indicates areas with the lowest stress.

Material Properties

Homogeneous- This term implies that the material properties were same in the entire
structure.

Isotropy- This term implies that the material properties were same in all directions.

Linear Elasticity- This term implies that the deformation or strain of the structure was proportional to the applied force and independent of strain rate.

Assumptions

The program used several assumptions with regard to the mechanical properties of the simulated structures.

- All the materials used in the model were assumed to be homogenous, isotropic and linear elastic.
- Ideal (100%) osseointegration in the interface between the bone and implant was assumed in this study.
- Since in many cases, the literature provides different values for the elastic properties of the same material, average values were chosen for this study.
- Elements and Nodes

The models were meshed with Tetrahedral, Ten-Noded Elements.

Observation and Results

The greatest Von Mises Stress (211.3) was observed on Assembly of Internal Hex Implant abutment connection under oblique loading of 100N. (Table 6, Graph 1& Fig. 4,5).

The greatest Von Mises Stress (159.39) was observed on Abutment of Internal Hex Implant abutment connection under oblique loading of 100N. (Table 7, Graph 2& Fig. 6,7).

The greatest Von Mises Stress (177.86) was observed on Implant of Internal Hex Implant abutment connection under oblique loading of 100N. (Table 8, Graph 3 &Fig. 8,9).

The greatest Von Mises Stress (44.836) was observed on Bone Implant Interface of Internal Hex Implant abutment connection under oblique loading of 100N. (Table 9, Graph 4 & Fig. 10,11)

Table 1: Mechanical properties – Bone

Material	Young's Modulus (GPa)	Poisson's coefficient
Cortical bone	15	0.30
Trabecular bone	1	0.25

Table 2: Mechanical properties – Implant model

Material	Component	Young's Modulus (GPa)	Poisson's coefficient
Ti-6Al-4V alloy	Abutment and screw	107.2	0.30
	Dental implant	110	0.35

Table 3: Mechanical properties – Prosthesis.

Material	Component	Young's Modulus (GPa)	Poisson's coefficient
Cr-Co alloy	Coping	218	0.33
Feldspathic Ceramic	Crown veneering	65	0.25

Table 4: Dimensional specifications – Implant.

Material	Ti-6Al-4V alloy
Macroscopic Design	Threaded
Length	10mm
Diameter	4.2mm

Table 5: Dimensional specifications – Prosthesis

Length	7.5mm
Mesiodistal width	11mm
Buccolingual width	10.5mm
Thickness	2 mm (0.5mm – Alloy) (1.5mm – Ceramic coating)

A: Internal Hex implant abutment connection

B: Conical Hex implant abutment connection

Figure 1: Modeling of different components of the model.

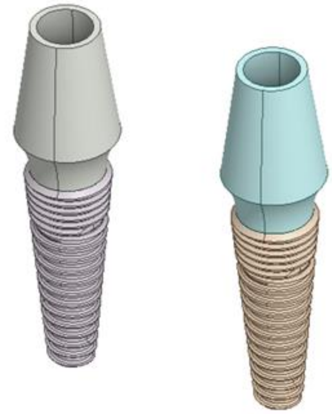


Figure 2: Modeling of Implant and Abutment for Internal Hex implant abutment connection and Conical Hex implant abutment connection.

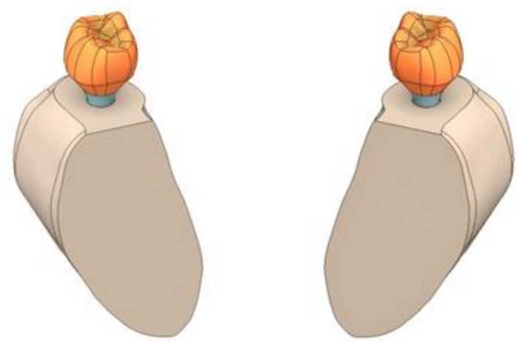
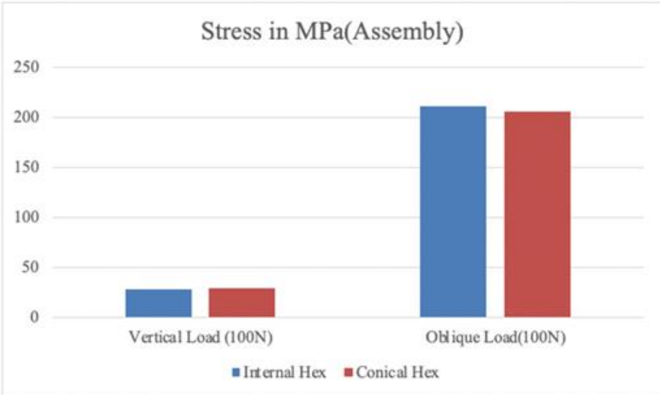


Figure 3: Assembly of different components of the model for Internal Hex implant abutment connection and Conical Hex implant abutment connection

Table 6: Maximum Von Mises stress (MPa) values with vertical and oblique load (100N) on Assembly of Internal Hex and Conical Hex Implant Abutment Connections.

Assembly	Stress in MPa under	
	Vertical Load (100N)	Oblique Load(100N)
Internal Hex	28.008	211.3
Conical Hex	29.546	205.08



Graph 1: Comparison of Von Mises stress (MPa) values with vertical and oblique load (100N) on Assembly of Internal Hex and Conical Hex Implant Abutment Connection

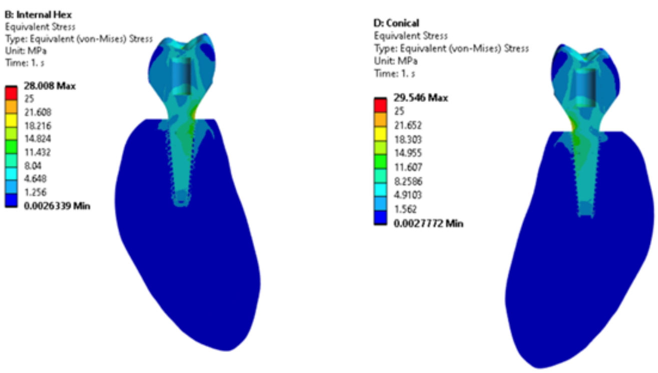


Figure 4: Stress distribution pattern on Assembly of Internal Hex and Conical Hex Implant Abutment Connections under vertical load of 100N

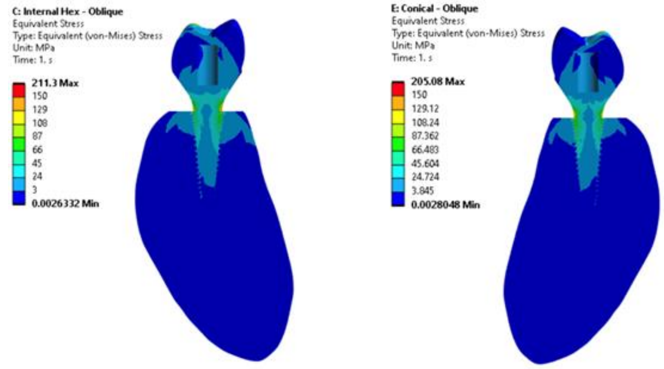
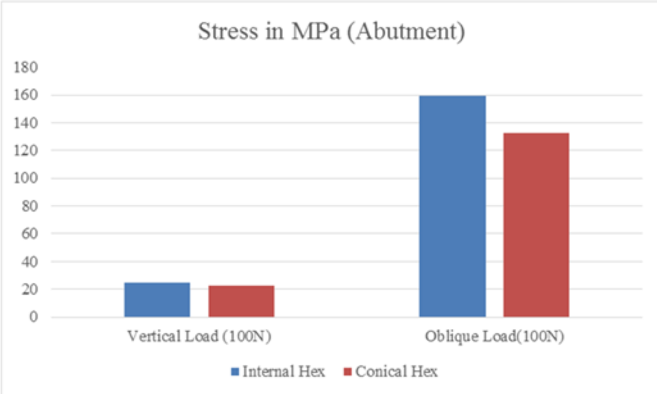


Figure 5: Stress distribution pattern on Assembly of Internal Hex and Conical Hex Implant Abutment Connections under oblique load of 100N.

Table 7: Maximum Von Mises stress (MPa) values with vertical and oblique load (100N) on Abutment of Internal Hex and Conical Hex Implant Abutment Connections.

Abutment	Stress in MPa under	
	Vertical Load (100N)	Oblique Load(100N)
Internal Hex	24.796	159.39
Conical Hex	22.823	132.54



Graph 2: Comparison of Von Mises stress (MPa) values with vertical and oblique load (100N) on Abutment of Internal Hex and Conical Hex Implant Abutment Connections.

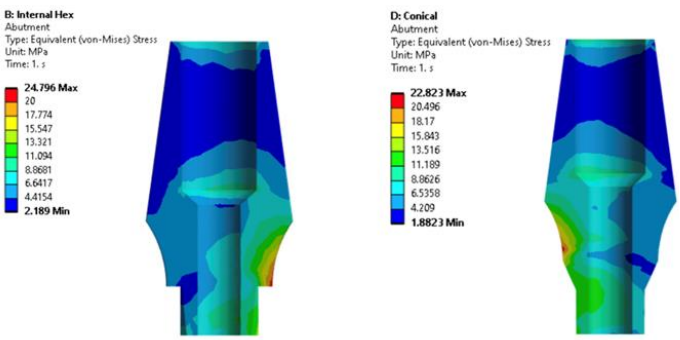


Figure 6: Stress distribution pattern on Abutment of Internal Hex and Conical Hex Implant Abutment Connections under vertical load of 100N.

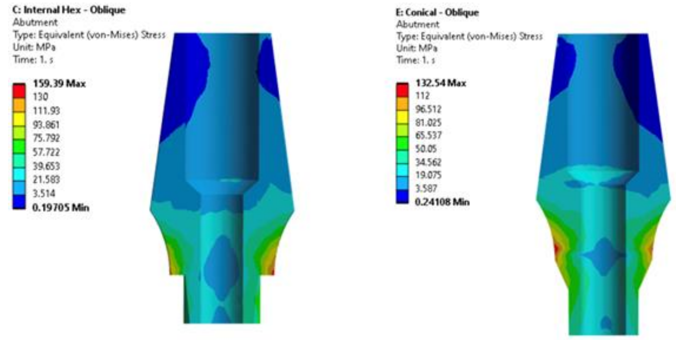
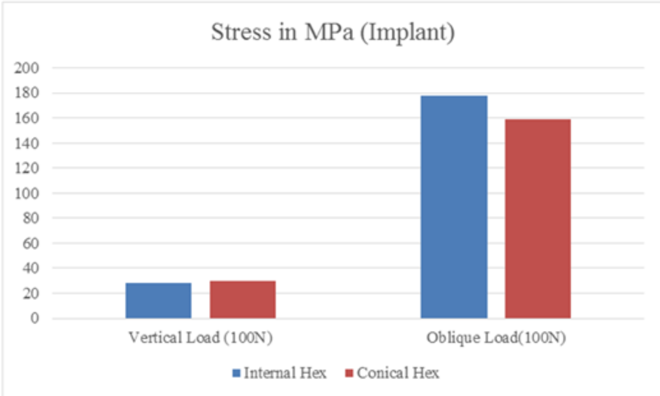


Figure 7: Stress distribution pattern on Abutment of Internal Hex and Conical Hex Implant Abutment Connections under oblique load of 100N.

Table 8: Maximum Von Mises stress (MPa) values with vertical and oblique load (100N) on Implant of Internal Hex and Conical Hex Implant Abutment Connections.

Implant	Stress in MPa under	
	Vertical Load (100N)	Oblique Load(100N)
Internal Hex	28.008	177.86
Conical Hex	29.546	158.86



Graph 3: Comparison of Von Mises stress (MPa) values with vertical and oblique load (100N) on Implant of Internal Hex and Conical Hex Implant Abutment Connections

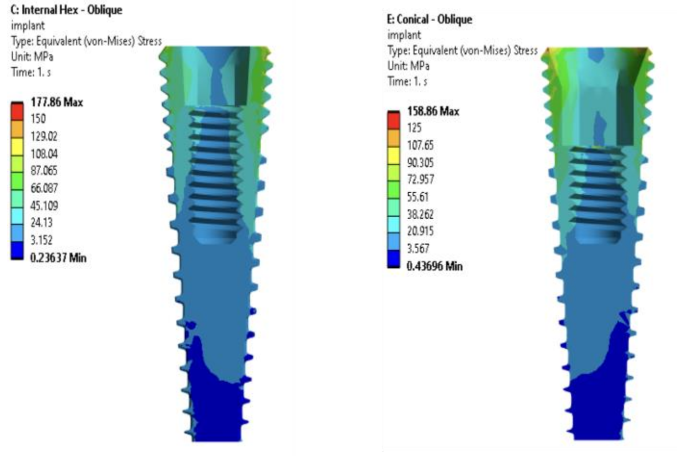


Figure 9: Stress distribution pattern on Implant of Internal Hex and Conical Hex Implant Abutment Connections under oblique load of 100N.

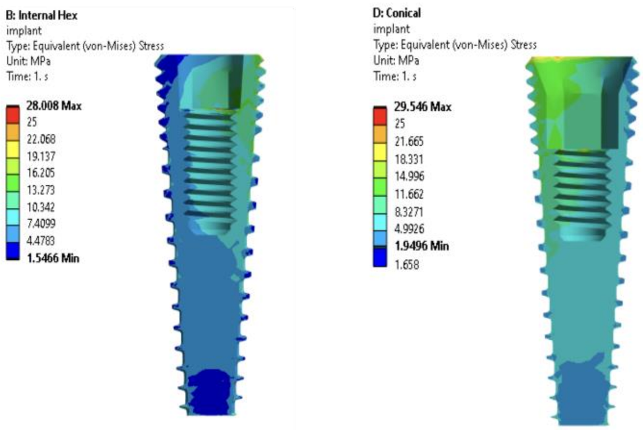
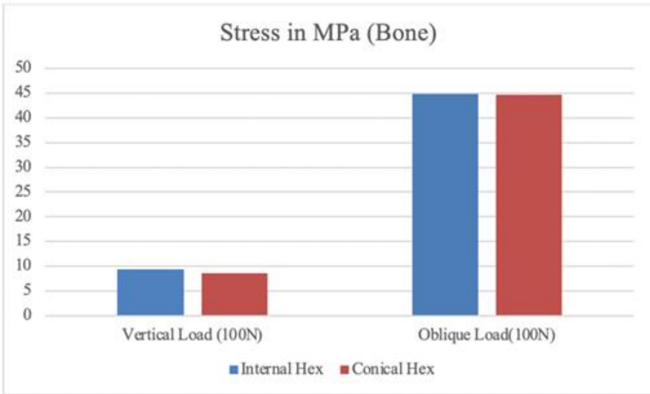


Figure 8: Stress distribution pattern on Implant of Internal Hex and Conical Hex Implant Abutment Connections under vertical load of 100N.

Table 9: Maximum Von Mises stress (MPa) values with vertical and oblique load (100N) on Bone Implant Interface of Internal Hex and Conical Hex Implant Abutment Connections.

Bone Implant Interface	Stress in MPa under	
	Vertical Load (100N)	Oblique Load(100N)
Internal Hex	9.3698	44.836
Conical Hex	8.6902	44.717



Graph 4: Comparison of Von Mises stress (MPa) values with vertical and oblique load (100N) on Bone Implant Interface of Internal Hex and Conical Hex Implant Abutment Connections.

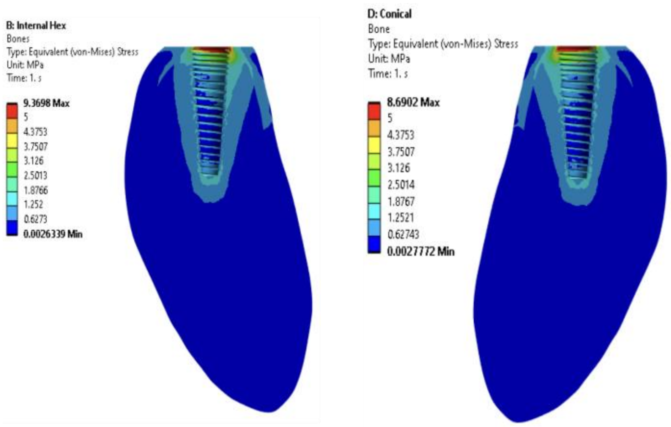


Figure 10: Stress distribution pattern on Bone-Implant Interface of Internal Hex and Conical Hex Implant Abutment Connections under vertical load of 100N.

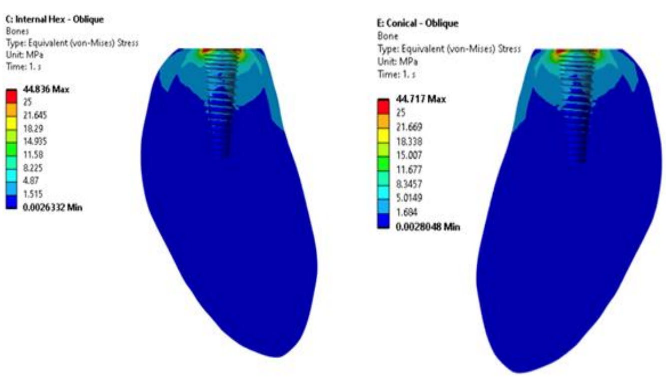


Figure 11: Stress distribution pattern on Bone-Implant Interface of Internal Hex and Conical Hex Implant Abutment Connections under oblique load of 100N.

Discussion

In FEA, the mechanical performance of the implant abutment interface can be evaluated by Von Mises stresses. Von Mises stress criterion is important to interpret the stresses within a ductile material, such as the implant material, as deformation occurs when the Von Mises stress value exceeds the yield strength[10]. In this study, the maximum Von Mises stress values of two different types of implant abutment interfaces under vertical and oblique load of 100N have been evaluated to better understand their stress distribution pattern and prevent biomechanical complications. To realistically simulate the clinical conditions, the implant models supported a PFM mandibular first molar prosthesis of ideal dimensions.

The study evaluated stress distribution pattern of Internal Hex Implant Abutment Connection and Conical Hex Implant Abutment Connection using a three dimensional finite element analysis.

On vertical Loading, the internal hex and conical hex connection assemblies displayed similar values of Von Mises stresses but the pattern of stress distribution in the internal hex implant abutment connection was dispersed more towards the bone as seen in fringes around the entire length of implant threads as opposed to the conical hex connection where the forces were more centrally located and not dissipating towards the bone implant interface beyond the cervical third of the implant. On oblique loading, the stresses increased manifold in both the connections. This observation was similar to observations made by Raoofi et al.[11] and Sarfaraz et al.[12] and they related this finding to the restricted areas of intimate connection between the outer surface of the abutment and inner surface of the fixture in conical connections and concluded that the stress was concentrated on a relatively small area.

Bone Implant Interface

Numerous studies have suggested that internal hex connections generate more stress on the bone while exerting less stress on the connection assembly. In contrast, conical connections exhibit higher stress at the joint assembly but dissipate less stress to the cortical bone, performing better as a force- transmission mechanism than other configurations[13,14.]

In several finite element studies, it has been found that the peak stress in the bone supporting a dental implant occurs in the marginal crest region, near the level where the implant attaches to the bone [5,16,17].

The findings from this study also align with the results of the previous studies in that the peak Von Mises stresses were observed at the crestal level of the surrounding bone. Under oblique loading, the Internal Hex implant abutment connection displayed greater stresses on the surrounding bone as compared to the Conical Hex implant abutment connection and the pattern of stress distribution in the Internal Hex implant abutment connection showed dissipation towards the bone whereas the stress distribution pattern was more internal in the Conical Hex implant abutment connection.

Summary and Conclusion

A three-dimensional finite element analysis was conducted to compare the stress distribution pattern of two different types of implant abutment connections**.** To conduct this study, a three-dimensional geometry of the edentulous mandible, comprising both cortical and cancellous bone, was generated from computerized tomography scans using SolidWorks software. Models with implant, abutment, prosthesis for Internal Hex and Conical Hex Implant abutment connections were created. A vertical and an oblique load of 100N was applied bilaterally to the mandibular molar region, and stress patterns were observed.

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

Implant Abutment Connection design emerged as a crucial factor among metal implant components. Stress distribution in the internal hex connection was more spread out, with stress reaching toward the bone along the full length of the implant threads. In contrast, the conical hex connection exhibited a more centralized stress pattern, with forces primarily concentrated in the cervical third of the implant and not dissipating noticeably toward the bone-implant interface.

Oblique loads generate greater stress as compared to vertical loads. Implant and abutment connections with long opposing lateral surfaces offer better load-bearing capacity than those with a shorter overlap between the implant and abutment. Under oblique loading, the internal hex implant-abutment connection experienced higher stress accumulation compared to the conical hex connection. This is likely due to reduced micromotion and improved resistance to oblique forces in the conical hex design.

An optimal implant-abutment design should minimize microgaps and reduce stress on the marginal bone. Under both vertical and oblique loading, the internal hex connection dissipates more stress into the surrounding bone than the conical hex connection, where peak Von Mises stresses concentrate near the marginal crest.

Furthermore, regarding the scope of the study, limitations in this research highlight the need for further research to correlate 3D finite element analysis (FEA) findings with biological clinical environments.

References

1. Abraham Celeste M. A Brief Historical Perspective on Dental Implants, Their Surface Coatings and Treatments. Open Dent J. 2014 May 16; 8:50-55.

2. Nikita Agarwal, Umesh Pai, Shobha J Rodrigues. Implant Abutment Connections: A Review. Karnataka Prosthodontic Journal. 2017 June; 1(3):13-25.
3. The Glossary of Prosthodontic Terms: Tenth Edition. The Journal of Prosthetic Dentistry. October (2023).130(4S1). Page e8.
4. Manoj Shetty, Krishna Prasad D., Naresh H. G. Shetty, Raghavendra Jaiman. Implant Abutment Connection: Biomechanical Perspectives. NUJHS. June 2014, Vol. 4, No.2, 47-53.
5. Pérez-Pevida E, Chávarri-Prado D, Diéguez-Pereira M, Estrada-Martínez A,Montalbán- Vadillo O, Jiménez-Garrudo A. Consequences of Peri-Implant Bone Loss in the Occlusal Load Transfer to the Supporting Bone in terms of Magnitude of Stress, Strain, and Stress Distribution: A Finite Element Analysis. Biomed Res Int. 2021;3087071:1-10.
6. Dittmer S, Dittmer MP, Kohorst P, Jendras M, Borchers L, Stiesch M. Effect of implant abutment connection design on load bearing capacity and failure mode of implants. J Prosthodont. 2011;20(7):510-516.
7. Tabata LF, Rocha EP, Barao VA, Assuncao WG. Platform switching: biomechanical evaluation using three dimensional finite element analysis. Int J Oral and Maxillofac Implants. 2011;26(3):482-491.
8. Natali AN, Pavan PG, Ruggero AL. Analysis of bone-implant interaction phenomena by using a numerical approach. Clin. Oral Impl. Res. 2006;17(1):67-74.
9. Sarfaraz H, Shenoy K, Hussain A, Sucheta P. A Comparison of Four Implant Abutment Connection Designs on the Stress Distribution Pattern using 3D Finite Element Analysis. Int J Oral Implantol Clin Res. 2015;6(1):1-8.
10. Tang, Chun-Bo & Liu, Si-Yu & Zhou, Guo-Xing & Yu, Jin-Hua & Zhang, Guangdong & Bao, Yidong & Wang, Qiu-Ju. Nonlinear finite element analysis of three implant– abutment interface designs. International Journal of Oral Science. Int J Oral Sci. 2012 Jun;4(2):101-108.
11. 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 Prospects. 2013;7(3):132-139.
12. Sarfaraz H, Shenoy K, Hussain A, Sucheta P. A Comparison of Four Implant Abutment Connection Designs on the Stress Distribution Pattern using 3D Finite Element Analysis. Int J Oral Implantol Clin Res. 2015;6(1):1-8.
13. 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.
14. 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. European Journal of Oral Sciences. 2007;115(5):408-416.
15. Streckbein P, Streckbein RG, Wilbrand JF, Malik CY, Schaaf H, Howaldt HP, Flach M. Non-linear 3D evaluation of different oral implant-abutment connections. J Dent Res. 2012;91(12):1184-1189.
16. Meijer, H. J. A., Starmans, F. J. M., Steen, W. H. A., & Bosman, F. (1993). A three- dimensional, finite-element analysis of bone around dental implants in

an edentulous human mandible. Archs Oral Biol. 1993;38(6):491-496.

17. Oh TJ, Yoon J, Misch Carl E, Wang HL. The Causes of Early Implant Bone Loss: Myth or Science. Journal Periodontol. 2002;73(3):322-333.