COMPARATIVE EVALUATION OF THREE DIFFERENT IMPLANT ABUTMENT CONNECTION ON STRESS DISTRIBUTION AROUND THREE DIFFERENT IMPLANT SYSTEM UNDER FUNCTIONAL LOAD: A 3-D FINITE ELEMENT ANALYSIS

INTRODUCTION

Dental implant restoration has been widely accepted as one of the treatment modalities to replace missing teeth and to restore human masticatory function. Most dental implant systems consist of two main parts: the abutment and the implant body.

In recent years, geometries of implant connections have been developed with different mechanical, biological, and esthetical characteristics. Two basic geometries are available: internal and external connections. External connections usually have an external hexagon on the implant platform, whereas internal connections can be divided into internal hexagons, internal octagons, and Morse taper connections. External connection presents a number of disadvantages, such as little contact length between the restoration and the hexagonal part of the implant head, some degree of rotation between the platform and the internal hexagon of the restoration, and high tension created in the screw connection.

Internal connection reduces mechanical and biological complications, such as screw loosening, fracture, and marginal bone loss. The greater depth of the connection in the fixture body allows more homogeneous dissipation of the mechanical stress; the stress is spread on the implant wall and, consequently, to the bone surrounding the entire implant and not only at the crestal level.

Internal hexagon connection systems are advantageous for producing anti-rotational, stable, and more resistant restoration with better force distribution. However, in internal hex restorations, adjustment of divergent implant angles might be difficult. Among the internal connection types, the taper joint system with a conical seal, or Morse taper, presents some of the advantages of the internal hex connection coupled with a better sealing of the joints.[1]

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. The design configuration of the abutment connection also plays a vital role in uniformly transferring occlusal stresses to the bone. There are various biomechanical techniques to evaluate the stress distribution of occlusal forces in bone around dental implants.[2]

A precise and well-designed connection leads to high rotational stability. Finally, a stable interlocking fit between implant and abutment reduces the occurrence of micro movements and guarantees that the retaining screw will remain in place without being exposed to the risk of screw loosening or screw breakage. There are different kinds of internal connections on the market, although the most reliable one has not been recognized. Therefore, this study was design to examine the role of internal connection design on stress/strain distributions within an implant structure.[3]

The finite element method may be applied to all kinds of materials in many kinds of situations: solids, fluids, gases, and combinations thereof; static or dynamic, and, elastic, inelastic, or plastic behaviour. R. Courant was a first researcher who developed this technique.[4] His main goal was to minimize the various calculative procedures to gain absolute solution to bio-mechanical system. He used ritz method to solve such numeric equations. Later in Turner et al. attempted to describe this method by developing broader definition of these numeric analyses.[5] Weinstein in 1976 used this technique in implant dentistry to evaluate various loads of occlusion on implant and adjacent bone. Since then, evolution of this technique has been observed in a very rapid and sophisticated scale in micro-computer as well as analysis of large-scale structural system.[6]

The finite element method (FEM) is a mathematical model analysis that gives detailed qualitative solution of the interaction between prosthesis, implant, and surrounding bone. The FEM can present possible changes in shape, based on stress values when the specific material properties, the forces applied, and the boundary conditions are predetermined.

In other words, FEA is a method whereby, instead of seeking a solution function for the entire domain, it formulates solution functions for each finite element and combines them properly to obtain a solution to the whole body. A mesh is needed in FEA to divide the whole domain into small elements. The process of creating the mesh, elements, their respective nodes, and defining boundary conditions is termed "discretization" of the problem domain. Since the components in a dental implant-bone system is an extremely complex geometry, FEA has been viewed as the most suitable tool to mathematically, model it by numerous scholars.

When applying FEA to dental implants, it is important to consider not only axial forces and horizontal forces (moment-causing loads), but also a combined load (oblique bite force), since these are more realistic bite directions and for a given force will cause the highest localized stress in cortical bone. Finite element analysis (FEA) has been applied to assess the mechanical characteristics of the implant–abutment connection in loading tooth and implant-supported prostheses.

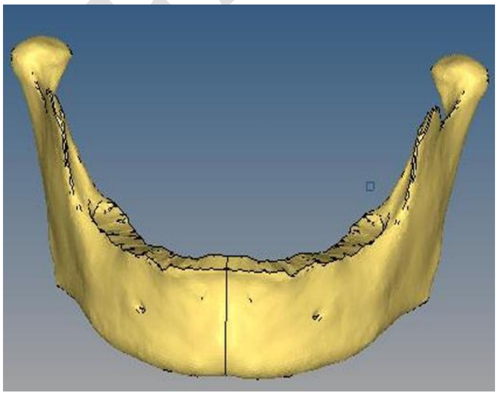
MATERIALS AND METHODS

Three 3-D finite element models of a mandibular first molar was prepared.
Three different types of implant-abutment connections used in commercially well-known implant systems were selected. The features of these connections were as follows:

1. Sample A: Tri-channel internal connection (Nobel BioCare, Goteborg, Sweden)
2. Sample B: Internal Conical-Hex (ADIN, Northern Israel)
3. Sample C: Internal OctaMorse taper method (Osstem, Korea).

1) Bone

Three 3-D finite element models of a mandibular first molar was prepared. Cortical bone thickness at the top and bottom as 2 mm and 3 mm.[7]Cancellous bone thickness 9 mm. The 3-D tetrahedral structural solid finite elements using ANSYS R 18.1 software was used to model the bone, implant, abutment.



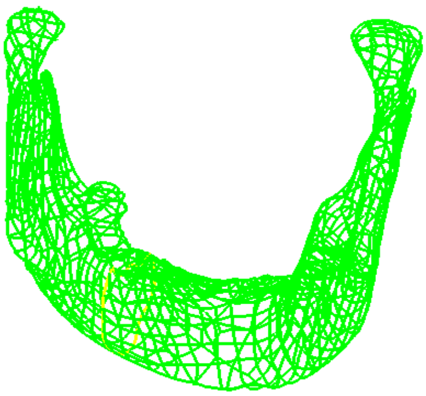
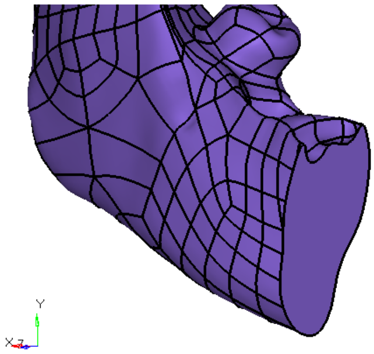


Fig 1: Geometrical Model of the bone Fig 2: Mandible (Step file format)



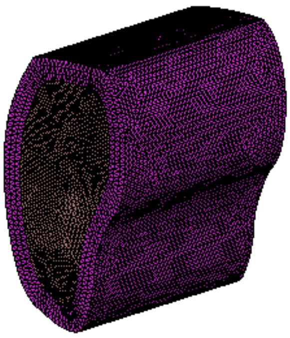
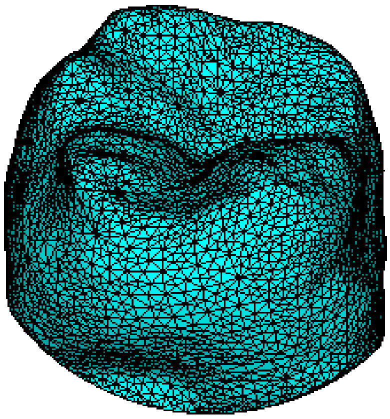


Fig 3: section at molar regionFig 4: 3D view of molar region



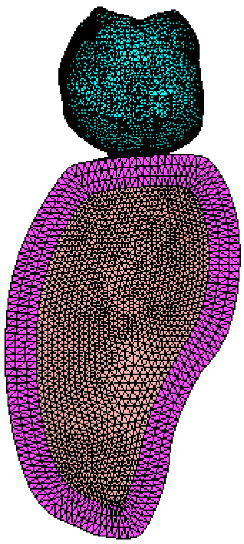


Fig 5: Crown Mesh

Fig 6: Assembly Mesh

Interface conditions

The bone-implant interface was assumed to be perfect, simulating complete osseointegration. Therefore, the connections between the 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 was free meshing because the model was not geometrically symmetric. Tetrahedral and octahedral elements were meshed, which were then assigned three degrees 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.[8]

Specifying material properties

All the materials used in the models consisted of the PFM crown, implants, abutments, and abutment screws. Compact and cancellous bone were presumed to be homogeneous, isotropic, and linearly elastic, similar to one another.

IMPLANT- ABUTMENT CONNECTION

1. SAMPLE A,B& C:

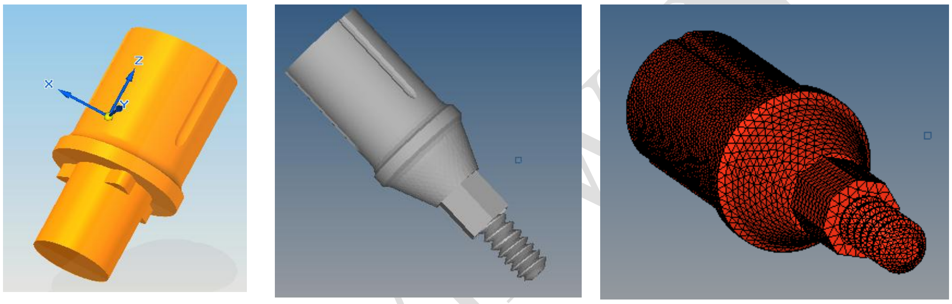


Fig 7: Tri-channel Internal Conical-Hex Internal OctaMorse

Applying boundary conditions

TABLE 1: NUMBER OF ELEMENT AND NODES

Details	Elements	Nodes
Tri connection	**634574**	**893268**
Conical Hex	**579679**	**799941**
Octa- connection	**567236**	**785424**

Table 2: Material properties[9]

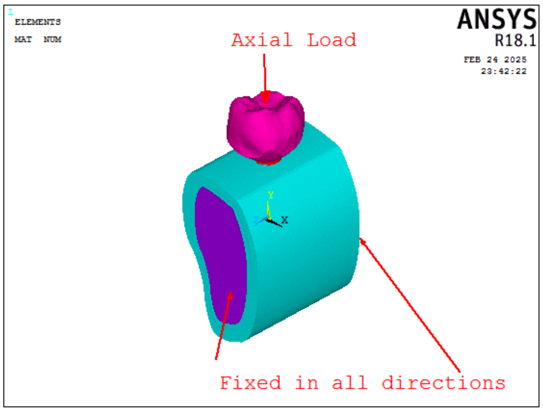
Material	Modulus of elasticity (MPa)	Poisson's ratio
Crown	**140**	**0.28**

Cortical bone	**14,000**	**0.30**
Cancellous bone	**620**	**0.30**
Titanium	**102,000**	**0.35**
Abutment-abutment screw	**11,400**	**0.38**

Application of load

The magnitude of applied loads was within physiologic limits. In studies conducted by Haraldson et al., patients treated with osseointegrated fixed prostheses had a maximum biting force between 42 and 412 N.[10] In this study, two types of loading conditions were simulated.

1) 200-N force applied directed axillary to the abutment surface.
2) 100-N force directed at 30 degrees to the long axis of the implant.[11]



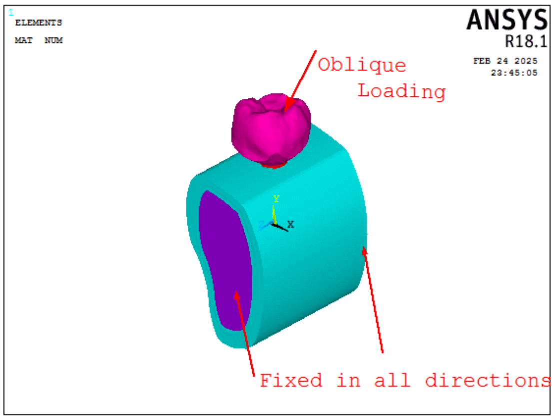


Fig 8: Axial Boundary conditionsFig 9: Oblique Loading boundary conditions

Finite element analysis

These different models were analysed by the Processor, and the results were displayed by the Post Processor of the FE Software (ANSYS R18.1, Canonsburg, Pennsylvania, United States) in the form of color-coded maps using von Mises stress analysis. The von Mises stress values were defined as the beginning of deformation for ductile materials. Metallic implant failure occurred when von Mises stress values exceeded the yield strength of the implant material. The von Mises stresses were most commonly reported in FEA studies to summarize the overall stress state at a point. The "equivalent stress of von Mises" was an expression that yielded an effective absolute magnitude of stresses, considering principal stresses in three dimensions. The types of stresses in FE studies were generally described by means of direction (shear, tension, and compression) or by an effective absolute value.[12]

Von Mises stress, which depended on the overall stress field, was a widely used indicator for assessing potential damage. Therefore, it was selected for presenting the results, as it played a crucial role in analysing the stresses within the implant material.[8]

In finite element (FE) studies, stresses were typically categorized based on direction (shear, tension, or compression) or represented by an effective absolute value, known as the von Mises equivalent stress. The highest von Mises stress was shown in red, while the lowest was depicted in blue. Intermediate stress values were represented by shades transitioning from bluish-green, green, and greenish-yellow to yellowish-red, in increasing order of stress magnitude.

RESULT

The samples were evaluated for stress in the implant, abutment, at the implant–abutment interface, and overall stress values [Figures 10-11] using 3D FEA analysis. Stress at 200 N, 100 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 89.3116MPa, when 100-N forces with 15° tilt [Table 1-2].

Stress Distribution

- **Study of Trilobe, Conical Hex & Octa implant abutment connection with 200 N axial load. (TABLE 1)**

  Maximum Von Mises stress = 35.549 MPa appeared during axial load in Octa implant system & least stress = 32.9642 MPa in Conical Hex System.

  Max displacement in octa system = 0.006499mm & least displacement = 0.006109mm in conical hex system.

- **Study of Trilobe, Conical Hex & Octa implant abutment connection with 100 N With 15° tiltoblique load. (TABLE 2)**

  Maximum Von Mises stress = 89.3116MPa appeared during axial load in Trilobe implant system & least stress = 79.8479 MPa in Conical Hex System.

  Max displacement in octa system = 0.021679mm & least displacement = 0.019005mm in Trilobe system.

TABLE 1 Stress values using three-dimensional finite element analysis

Axial Loading (200 N)	Sample A (Tri – Nobel)	Sample B (Conical Hex-Addin)	Sample C (Octa-OSSTEM)
Overall Stress (MPa)	35.4691	32.9642	35.549
Overall deformation(mm)	0.006208	0.006109	0.006499
Abutment Stress (MPa)	34.0505	32.9642	35.549

TABLE 2 Stress values using three-dimensional finite element analysis

Oblique – Loading (100 N)	Sample A (Tri – Nobel)	Sample B (Conical Hex-Addin)	Sample C (Octa-OSSTEM)
Overall Stress (MPa)	89.3116	79.8479	85.0499
Overalldeformation(mm)	0.019005	0.019803	0.021679
Abutment Stress (MPa)	86.9053	79.8479	85.0499

SAMPLE: A , B & C Abutment Stress (MPa) Axial Loading (200 N)

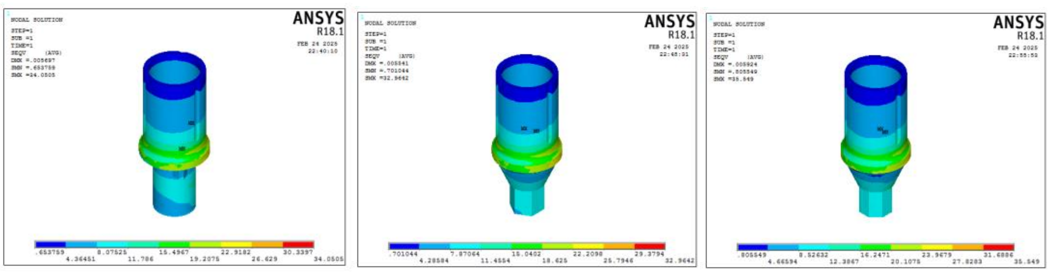


Fig10

SAMPLE: A , B & C Abutment Stress (MPa) Oblique Loading (100 N)

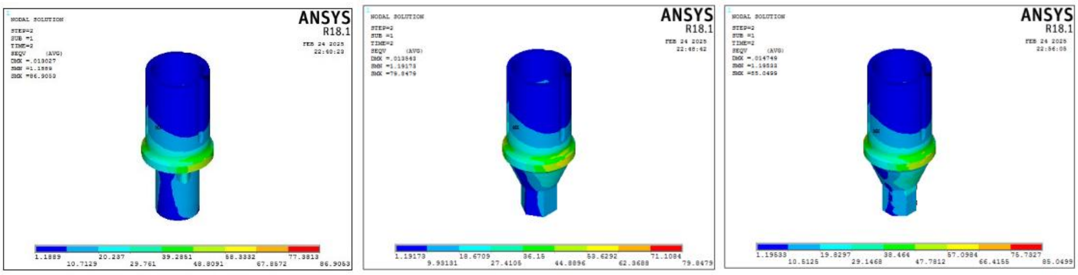


Fig 11

DISCUSSION

Finite Element Analysis (FEA) is a method used to assess the mechanical behaviour of complex structures. Its key advantages include its repetitive nature, cost-effectiveness, versatility in various analyses, adaptability to different clinical conditions, and high precision. Additionally, FEA does not require sophisticated equipment and allows for the design and evaluation of custom-made implants. This approach enables operators to numerically determine stress and strain levels at any specific point within the analysed object. FEA can be conducted in either two-dimensional (2D) or three-dimensional (3D) models.

According to Meijer et al, 1992FEA can be performed at 2D or 3D. 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.[13]

According to Quaresma et al, 2008, Lin et al. 2000 and Hansson et al. 2007 Conical implant–abutment connection systems performed better as a force transmission system than internal hexagonal and external flat top systems.[14]

According to Tsuge et al,2008 Five different implant systems with the cross-sectioning method using a Scanning Electron Microscope and found that the trilobe shape (Replace Select) gave rise to the largest microgap when compared with the other systems.[15]

According to Semper et al, 2009, Saidin et al.2012 Increasing the number of edges increases the degree of rotational freedom. In this regard, internal hexagonal abutments should provide high stability, followed by internal octagonal.[16]

According to Yamanshiet al,2012Compared external hex, internal cone, internal straight connections, and effect of implant–abutment interface, and peri-implant stress showed largest amount of abutment movement, higher labial bone stresses in external hex, whereas internal conical connection had lowest abutment movement and low labial pericoronal bone stresses.[17]

According to Pellizer et al.2014 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.Internal octagon + cone presented the lowest stress concentrations; external hex exhibited the greatest stresses, which were in accordance with our study.[18]

The conical internal hexagonal implant–abutment connection designs provide more biomechanically suitable prosthetic options than other systems, particularly, in cases with increased vertical dimension in the posterior regions.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.

It's important to understand that the resistance of an abutment to fracture is not determined solely by the shape or geometry of the implant–abutment connection. Other elements and design features can also play a significant role. These may include the number of parts involved (such as whether the abutment is a one-piece or two-piece connection), the length and thickness of the screw, the design of the threads, the type of material used, and the surface area of contact.

CONCLUSION AND DRAWBACKS

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. 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.In addition, 100% implant–bone interface was established, which does not necessarily simulate clinical situations.

The abutment-implant tri-channel internal connection has maximum stress distribution in and around implant at different forces applied followed by to internal octa Morse Taper and the least stress distribution in the internal – hex morse taper connection.

REFERENCES

1. Maeda Y, Satoh T, Sogo M. In vitro differences of stress concentrations for internal and external hex implant-abutment connections: a short communication. J Oral Rehabil. 2006;33:75–78.
2. Geng JP, Tan KB, Liu GR. Application of finite element analysis in implant dentistry: a review of the literature. J Prosthet Dent. 2001;85:585 598.
3. Barbier L, Vander Sloten J, Krzesinski G, Schepers E, Van der Perre G. Finite element analysis of non-axial versus axial loading of oral implants in the mandible of the dog. J Oral Rehabil. 1998;25:847–858.
4. Yettram AL, Wright KW, Pickard HM (1972) Finite element stress analysis of crowns of normal and restored teeth. Acta Orthop Scand 3:304-4.
5. Turner MJ, Clough RW, MactinandL HC (1956) Stiffness and defection analysis of complex structure. J aeronaut science 23:805-823.
6. Yijunliu (2003) Introduction to FEM –CAE research laboratory university of Cincinnati U.S.A. Accessed.
7. Liu J, Pan S, Dong J, Mo Z, Fan Y, Feng H. Influence of implant number on the biomechanical behaviour of mandibular implant-retained/supported overdentures: a

three-dimensional finite element analysis. Journal of dentistry. 2013 Mar 1;41(3):241-9.

8. 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.
9. Rungsiyakull C, Chen J, Rungsiyakull P, Li W, Swain M, Li Q. Bone’s responses to different designs of implant-supported fixed partial dentures. Biomech Model Mechanobiol 2015;14:403-11.
10. Cehreli MC, Iplikcioglu H. In vitro strain gauge analysis of axial and off-axial loading on implants supported fixed partial dentures. Implant Dentistry 2002; 11:286-90.
11. Morneburg TR, Pröschel PA. Measurement of masticatory forces and implant loads: a methodologic clinical study. International Journal of Prosthodontics. 2002 Jan 1;15(1).
12. Jianping Geng, Weiqui Yan, Wei Xu – Application of Finite Element Method in Implant Dentistry.2008.
13. Meijer HJA, Kuiper JH, Starmans FJM, Bosman F. Stress distribution around dental implants: Influence of superstructure, length of implants and height of mandible. J Prosthet Dent 1992;68:96-102.
14. 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.
15. Tsuge T, Hagiwara Y, Matsumura H. Marginal fit and microgaps of implant–abutment interface with internal anti rotation configuration. Dental Materials Journal 2008;27:29–34.
16. Saidin, S., Kadir, M.R.A., Sulaiman, E. and Kasim, N.H.A., 2012. Effects of different implant–abutment connections on micromotion and stress distribution: Prediction of microgap formation. *Journal of dentistry*, *40*(6), pp.467-474.
17. Saidin, S., Kadir, M.R.A., Sulaiman, E. and Kasim, N.H.A., 2012. Effects of different implant–abutment connections on micromotion and stress distribution: Prediction of microgap formation. *Journal of dentistry*, *40*(6), pp.467-474.
18. Pellizzer EP, Carli RI, Falcón-Antenucci RM, Verri FR, Goiato MC, Villa LM, et al. Photoelastic analysis of stress distribution with different implant systems. J Oral Implantol 2014;40:117-22.