How to Cite:
Meenakshi, M., Sahoo, S., Sharma, Y., Sharma, A., Pooja, P., & Shreysha, S. (2022). To evaluate and compare the effect of stresses in peri implant bone after offset placement of implant in edentulous posterior mandible using: 3D finite element analysis (FEA). *International Journal of Health Sciences*, *6*(S2), 10097–10105. https://doi.org/10.53730/ijhs.v6nS2.7634

To evaluate and compare the effect of stresses in peri implant bone after offset placement of implant in edentulous posterior mandible using: 3D finite element analysis (FEA)

Dr. Meenakshi
Post Graduate Student final year) Shree Bankey Bihari dental college, Ghaziabad (U.P.)

Dr. Sukant Sahoo
Head of the Department) Shree Bankey Bihari dental college, Ghaziabad (U.P.)

Dr. Yukti Sharma
Senior lecturer, Shree Bankey Bihari dental college, Ghaziabad (U.P.)

Dr. Aparana Sharma
Senior lecturer, Shree Bankey Bihari dental college, Ghaziabad (U.P.)

Dr. Pooja
BDS, MBA in healthcare, Public Health Intern and researcher in Khushi baby, Jaipur

Dr Shreysha
Post Graduate student first year, Shree Bankey Bihari dental college, Ghaziabad (U.P)

***Abstract*---**Background: In the present era, when it comes to oral rehabilitation, a wide array of options are available to restore missing teeth using fixed and removable prostheses, however, advances in dental implant surgery offer viable option to provide improved functional efficiency, appearance, comfort and speech2 . The desired position of the artificial teeth is determined by aesthetic and functional requirements. Lack of bone volume always results in exposure of implant surface, reduced bone implant contact and finally implant failure. Aim: The aim of this study is to evaluate and compare the effect of stresses in peri implant bone after offset placement of implant in edentulous posterior mandible using: - "3D. finite element analysis (FEA)." Objectives: 1. To evaluate the effect of stresses in peri

Manuscript submitted: 27 March 2022, Manuscript revised: 18 April 2022, Accepted for publication: 9 May 2022

implant bone after offset placement of dental implant using 3D finite element analysis. 2. To analyze the stress distribution patterns in the implant and the peri implant bone interface. 3. To evaluate and compare the stress in peri implant bone after placing the implant in different implant configurations – straight and offset. Materials And Methodology: Graphic Card: Radeon Vega Graphics 3.50 GHz. Finite Element Analysis: Pre-processing: 1. Mesh Generation 2. Specifying material properties' 3. Applying boundary conditions 4. Application of different loads. Post-Processing: Von Mises stress analysis Conclusion On comparative evaluation of the stress distribution pattern around the implant, it was found that there is more stress in peri implant bone after placing implants with offset configurations as compared to straight line configurations.

***Keywords*---**dental implants, offset placement, peri implant, edentulous.

Introduction

Dental implants have proven to be an effective way of restoring masticatory ability of completely or partially edentulous patients[1] . In the present era, when it comes to oral rehabilitation, a wide array of options are available to restore missing teeth using fixed and removable prostheses, however, advances in dental implant surgery offer viable option to provide improved functional efficiency, appearance, comfort and speech[2] . The desired position of the artificial teeth is determined by aesthetic and functional requirements. A well planned and executed prosthesis is desirable to avoid excessive and unnecessary stresses on implant components and peri-implant bone[2] . Sufficient quantity and quality of bone for implant placement is an essential prerequisite for long-term success in oral implant therapy. The quantity of alveolar bone decreases after periodontal disease or after extraction, causing bone loss in both horizontal and vertical directions[3] .

Conventional placement of dental implants is not indicated in some anatomical restrictions such as proximity of the maxillary sinus or inferior alveolar nerve canal in mandible[4] . Lack of bone volume always results in exposure of implant surface, reduced boneimplant contact and finally implant failure. Lack of bone volume is more common in anterior maxilla due to which long-term prognosis for implants in maxilla is less secure than that of edentulous mandible. Following tooth extraction in the anterior part of maxilla, horizontal bone resorption is almost twice as pronounced as vertical resorption. This can be managed either by surgical correction or by positioning the implant in the area with greatest available bone[5] . Under load, bone tissue undergoes remodeling, which ultimately influences success of 1 the dental implant. Bone remodeling is a complex process that involves a sequence of chemical and mechanical mediated biologic events known as mechanotransduction[6] .

Offset configuration has been followed over the years where it is difficult to place the implant in desired position, in order to overcome the difficulty, clinicians have placed the implant either buccally, palatally or lingually due to quantitative or anatomical constraints by only taking emergence profile into consideration[4]. Studies have indicated that vertical and axial loading of implant is best tolerated by underlying bone. And that any deviation from central axis might lead to higher stresses that might cause loss of peri -implant bone[7] . Due to presence of such stresses in the implant-prosthesis complex it becomes necessary to study their nature and effect on the surrounding bone, since, its is not possible to study such stresses efficiently in the human model, therefore, we must turn to other mechanical/computational methods, one such method is Finite Element Analysis (FEA).

Initially finite element analysis (FEA) was developed in the early 1960s for the aerospace industry, FEA was introduced in 1976 to implant dentistry. Since then, it has shown overwhelming capability and versatility in its application in dentistry. [8] The finite element method (FEM) is a numerical analysis technique for obtaining approximate solutions to a wide variety of engineering problems. A finite element model of a problem gives a precise approximation of the governing problem[9] . Three-dimensional finite element analysis (3D-FEA) has been frequently used in implant dentistry to predict the stress distribution pattern in implant components and the peri-implant bone. [10]

Though, conventional method of implant placement is the most preferred methodology, offset placement of an implant has shown its own merits in terms of surgical technique, aesthetics and function, yet, in existing literature, there is a conflict between conventional method and offset method of implant placement, pertaining to the amount of stresses imposed upon the peri-implant bone.

Aim & Objectives

Aim

The aim of this study is to evaluate and compare the effect of stresses in peri implant bone after offset placement of implant in edentulous posterior mandible using: - “3D finite element analysis (FEA).”

Objectives

The objectives of the study

1. To evaluate the effect of stresses in peri implant bone after offset placement of dental implant using 3D finite element analysis.
2. To analyze the stress distribution patterns in the implant and the peri implant bone interface.
3. To evaluate and compare the stress in peri implant bone after placing the implant in different implant configurations – straight and offset.

Material and Methods

Finite element modelling is described as the representation of the geometric model in terms of a finite number of elements and nodes which are building blocks of the numerical representation of the model. An element which may consist of triangular or quadrilateral shapes in mathematical matrix of the collective interaction among degree of freedom whose (displacements) and actions (forces) of structure under load are considered to exist. In addition to the information about elements and nodes, this model also consists information about material and other properties, loading and boundary conditions. Finite element analysis is a numerical method, which is based on principal of dividing the structure into finite numbers of small elements that are inter connected with each other at the corner points or nodes having three degree of freedom which translates in x, y, & z directions. Each element is assigned unique elastic properties (Poisson's ratio and modulus of elasticity) to represent the materials modeled and for reach elements, its mechanical behavior can be written as a function of displacement of the nodes. These nodes are submitted to certain loading conditions, resulting in behavior of the model similar to the structure represents. When a computer analysis is performed, the system of simultaneous equations can be solved to relate all forces and displacements at the nodes. From this stresses and stress contours can be established in each element and thus for the whole body. The method has gained increased usage in biomechanical disciplines includes orthopedic, cardiac, and dental mechanics.

This study was conducted by AnaMac design, Gurugram. A modeling software 'DASSAULT SOLIDWORKS-2021'was used which is a solid modeling computer aided design (CAD) and computer aided engineering (CAE) computer program. Once a structure is numerically created and material properties are assigned it can be analyzed for stress distributions during force application using finite elements software. The finite element used in this study was 'ANSYS R16.2 Workbench. The stresses were expressed as compressive (which are negative) or tensile (which are positive). The global (x, y, z directional axes) combination of the absolute values squared of all stresses has evolved.

System Configuration

A computer with the following system configuration was used

· Windows edition-Windows 10 Pro
· Processor-AMD Ryzen 3 2200G
· RAM: 4.00GB
· 64-Bit operating system, x64- based processor
· Graphic Card: Radeon Vega Graphics 3.50 GHz

Steps involved in study: Finite Elements (3D) Modeling

Construction of geometric model:

a) Modeling of the bone.
b) Modeling of the implant with Abutment.
c) Modeling of the interface.

Finite Element Analysis

Pre-processing

1. Mesh Generation
2. Specifying material properties'
3. Applying boundary conditions
4. Application of different loads.

Post-Processing

Von Mises stress analysis

Finite Element Model (FEM)

Computed tomographic (CT) images of the edentulous mandible at the first premolar and molar region of an adult male was used to develop models by means of a modeling software SOLIDWORKS 2021 and rendered by using the key shot software.

Finite Element Analysis

Pre processing

a) Mesh generation
   The three-dimensional finite element model corresponding to the geometric model was meshed using ANSYS 16.2 software. The type of meshing is free meshing because the model is not geometrically symmetric. The element size was selected according to default settings. The type of element suitable for this particular study was four noded tetrahedral (1ST order tetra) element which was assigned three degree of freedom per nodes, namely translation in the x, y and z directions. The element was constructed so that their size aspect ratio would yield reasonable solution accuracy. The first model with straight design incorporated 514558 elements and 104544 nodes and with offset designs incorporated 911200 elements and 176906 nodes respectively. All nodes at the base of 3D models were restrained to determine the boundary conditions.
b) Specifying material properties
   For the execution and accurate analysis of the program and interpretation of the result, two material properties were utilized i.e., young's modulus and Poisson's ratio. The implants, abutments, crowns placed on cortical and cancellous bone were considered. The mechanical properties of the interface material were mathematically calculated, assuming it to be acomposite material. This condition assumes a uniform strain on both the implant and bone portions of the composite. Thus, bonding between the implant and the bone remains intact during theapplication of stress. According to the mathematical equation, interface martial is a composite of young's modulus of different materials such as the titanium alloy and bone. The young's modulus of the whole system as composite is calculated considering as iso-strain condition. The young's modulus of the composite is derived in terms

of the elastic modulus and the volume fraction of the implant and bone. The load on the composite is equal to the sum of the load on the implant and the load on the bone.

Methodology

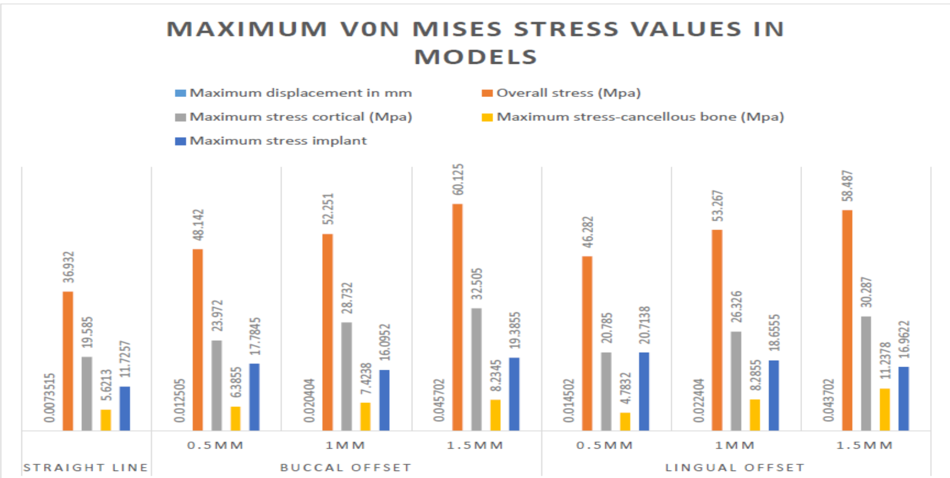
Mechanical properties of different material used Young's modulus Poisson's ratio (Gpa) (U) Cortical bone 13.70 0.30 Cancellous bone 1.37 0.30 Implant (titanium alloy) 110,000 0.35 Porcelain 70,000 0.190
Table No-1 Mechanical properties of different materials used in the model

a) Simulation of the models:
To simulate the model during masticatory movements, two types of loads were applied, which are in oblique and vertical directions. The 100.0 N oblique and 100.0 N vertical loads were applied. The oblique load was applied at 30 degrees relatively to the long axis of the implants to the long axis. Osseointegration was considered to be complete.

FEA Data Collection

ANSYS software was used for qualitative and quantitative stress analysis, considering material properties, meshing and loading. The maximum von Mises stresses (Maximum Equivalent Stress) at the implant surface and abutment and also strain values were reported as qualitative analysis. The stress distribution pattern was also evaluated by color coded diagrams as the quantitative analysis, in which areas with the highest and the lowest stresses were depicted as red and blue, respectively.

Results



Graph 1- Distribution of stress on different implant configurations (straight and offset)

	Straight line	Buccal offset			Lingual offset		
		0.5mm	1mm	1.5mm	0.5mm	1mm	1.5mm
Displacement	0.0073515	0.012505	0.020404	0.045702	0.014502	0.022404	0.043702
Stress on cortical bone	19.585	23.972	28.732	32.505	20.785	26.326	30.287
Stress on cancellous bone	5.6213	6.3855	7.4238	8.2345	4.7832	8.2855	11.2378
Stress on implant	11.7257	17.7845	16.0952	19.3855	20.7138	18.6555	16.9622
Overall stress	36.932	48.142	52.251	60.125	46.282	53.267	58.487

Table 2- Distribution of stress on different implant configurations (straight and offset)

From the above table and graph, it was deduced that there is more displacement in offset design as compared to straight line design. For overall stress, it was also observed that in offset design there is more stress as compared to straight line design. For both cortical and cancellous bone the higher stress is found to be in offset design. So, this finding suggests that offset, transfer more stress to bone as compared to straight line design. For implants also more stress interference is seen in offset design as compared to straight line. So, all the above findings suggests that there is more stress in peri implant bone after placing the implant with offset design.

	Straight line	Buccal offset			Lingual offset			Row Totals
		0.5mm	1mm	1.5mm	0.5mm	1mm	1.5mm	
Displacement	0.0073515 (0.01730294)	0.012505 (0.0225556)	0.020404 (0.2448236)	0.045702 (0.028176949)	0.014502 (0.02168473)	0.022404 (0.02495878)	0.043702 (0.0274014)	0.1665705
Stress on cortical bone	19.585 (18.92345776)	23.972 (24.64786549)	28.732 (26.77896486)	32.505 (30.81569365)	20.785 (23.76095367)	26.326 (27.89634874)	30.287 (29.78536454)	182.192
Stress on cancellous bone	5.6213 (5.89646758)	6.3855 (7.68497464)	7.4238 (7.68745987)	8.2345 (8.78945436)	4.7832 (6.76589765)	8.2855 (7.78654578)	11.2378 (8.55786467)	51.9716
Stress on implant	11.7257 (12.67549375)	17.7845 (16.67896567)	16.0952 (17.67453965)	19.3855 (20.54496579)	20.7138 (15.89647659)	18.6555 (18.74635677)	16.9622 (19.79697858)	121.3224
Overall stress	36.932 (36.92702428)	48.142 (48.76345802)	52.251 (52.65897643)	60.125 (60.67853965)	46.282 (46.76549832)	53.267 (53.45092674)	58.487 (58.32981754)	355.486
Column totals	73.8713515	96.296505	104.522404	120.295702	92.578502	106.55604	117.017702	711.138571

Table no-3 Chi-square test (comparison of different implants configurations)

The chi-square statistic is – 46.802275 . The result is significant at $p < 0.5$ at which chi-square is 36.42.

These finding suggests that there is significant difference between all parameters and all implant configurations.

Discussion

A main factor influencing the final success or failure of implant supported restorations is the manner in which stresses are transferred to the implant and surrounding bone. The present study aimed to investigate the stress distribution around the peri implant bone after offset placement of implant in the mandibular posterior region. The key finding of the present study was that the stress concentration in the peri-implant bone increased by the increasing the amount of offset placement. The present results also showed that stress concentration in peri-implant bone in different configurations of offset placement was more than that of in line configuration.

The study results also showed that the stress concentration in peri implant bone increased as the amount of offset placement increased. For bone stresses, the offset placements present no obvious advantage on stress reduction over the in-line placement, and they could provoke a higher stress at the bone around some implants and might increase the risk of implant failure. Based on all these results, it is suggested that the amount of offset should be as minimum as possible. The present findings are in partially disagreement with those of Anitua and Orive who compared the effect of the distal offset configurations of a single implant supported prosthesis on bone stress distribution with that of straight alignment using finite element analysis. They indicated that the offset implant placement produced less stress compared to the straight-line configurations. This disagreement can be explained by the fact that conditions they evaluated were quite different from those of the present study. In Anitua and Orive study, static load of 200N and 230 N were applied on the mesial and distal borders of the prosthetic unit, respectively. On the other hand, the present study, a vertical and oblique force of 100N was applied to the long axis of implant. Since the load transfer from implants to surrounding bone largely depends on the type and direction of the loading, the difference in the direction of the loads between the two studies could be partially responsible for the disagreement.46 Moreover, it should be noted that the material properties of mandible are other key factors that could greatly affect the predictive accuracy of the FEA studies.

Finite Element Analysis is a sophisticated engineering tool, which has been used extensively in design optimization and structural analysis. It represents one of the most developments in the history of the computational method. This method originated in the aerospace industry as a tool to study stress in complex airframe structures. It grew out of what was called the matrix analysis method used in aircraft design. Turner et al first used the modern version of the finite element method in the engineering practice. Argyris and Clough coined the term finite element in 1960.

Conclusion

On comparative evaluation of the stress distribution pattern around the implant, it was found that there is more stress in peri implant bone after placing implants with offset configurations as compared to straight line configurations. The stress concentration in peri-implant bone in different configurations of offset placement was more than that of in-line configurations. In the offset configurations, the amount of stress concentration in the peri implant bone increased by increasing the amount of offset placement. Therefore, it is suggested that the amount of offset should be minimum as possible. For bone stresses, the offset placements present no obvious advantage on stress reduction over the in-line placement, and they could provoke a higher stress at the bone around some implants and might increase the risk of implant failure.

References

1. Atwood DA. Some clinical factors related to rate of resorption of residual ridges- J. Pros. Dent. 1962 May 1;12(3):441-50.
2. Shimura Y, Sato Y, Kitagawa N, Omori M. Biomechanical effects of offset placement of dental implants in the edentulous posterior mandible. Int. J of implant dent. 2016; Dec2(1):1-3
3. Pinar turkoglu and murat audin. Influence of offset placement of mandibular dental implants over the stress levels of bone. EC dental science 2019;18(4):583-93.
4. Hakimeh siadat; effect of offset implant placement on the stress distribution around a dental implant; A three -dimensional finite element analysis, J of oral implantology 2015; Apri14(6):646- 51.
5. Omori M, Sato Y, Kitagawa N, Shimura Y, Ito M. A biomechanical investigation of mandibular molar implants: reproducibility and validity of a finite element analysis model. International J of implant dentistry. 2015 Dec;1(1):1-3.
6. Rubo JH, Capello Souza EA. Finite-element analysis of stress on dental implant prosthesis. Clinical implant dentistry and related research. 2010 Jun;12(2):105-13.
7. Murat Sutpideler M, Eckert SE, Zobitz M. An KN Finite element analysis of effect of prosthesis height, angle of force application, and implant offset on supporting bone. Int J Oral Maxillofac Implants. 2004 Nov1;19(6)819-25
8. Trivedi S. Finite element analysis: A boon to dentistry. J of ora biol and cran resea. 2014 Sep 1;4:200-3.
9. Piccioni MA, Campos EA, Saad JR, de Andrade MF, Galvão MR, Abi Rached A. Application of the finite element method in Dentistry. RSBO Revista Sul-Brasileira de Odontologia. 2013;10(4):369-77.
10. Geng JP, Tan KB, Liu GR. Application of finite element analysis in implant dentistry: a review of the literature. The J of pros dent. 2001 Jun 1;85(6):585-98.
11. Anitua E, Orive G. Finite element analysis of the influence of the offset placement of an implantsupported prosthesis on bone stress distribution. J Biomed Mater Res B Appl Biomater. 2009;89:275-81