International Journal of Health Sciences and Research
Volume 14; Issue: 5; May 2024
Website: www.ijhsr.org
ISSN: 2249-9571

Original Research Article

A 3-D Finite Analysis of Stress and Strain Around a Dental Implant with Different Abutment Angulations and Two Crest Module Design

Dr. Dushyant Soni[1], Dr. Nitin Modi[2], Dr. Dron Lakhani[3], Dr. Rajshree Bhandari[4], Dr. Amrit Assi[5], Dr. Anuva Aseri[6], Dr. Sanjana Goyal[7], Dr. Sonika Patil[8]

[1]Professor and HOD, Department of Prosthodontics, Crown and Bridge & Implantology, Vyas Dental College and Hospital, Jodhpur, India
[2]Reader, Department of Prosthodontics, Crown and Bridge & Implantology, Vyas Dental College and Hospital, Jodhpur, India
[3]Professor, Department of Prosthodontics, Crown and Bridge & Implantology, Vyas Dental College and Hospital, Jodhpur, India
[4]Reader, Department of Prosthodontics, Crown and Bridge & Implantology, Vyas Dental College and Hospital, Jodhpur, India
[5]Senior Lecturer, Department of Prosthodontics, Crown and Bridge & Implantology, Vyas Dental College and Hospital, Jodhpur, India
[6]Post Graduate Student, Department of Prosthodontics, Crown and Bridge & Implantology, Vyas Dental College and Hospital, Jodhpur, India
[7]Post Graduate Student, Department of Prosthodontics, Crown and Bridge & Implantology, Vyas Dental College and Hospital, Jodhpur, India
[8]Post Graduate Student, Department of Prosthodontics, Crown and Bridge & Implantology, Vyas Dental College and Hospital, Jodhpur, India

Corresponding Author: Dr. Dushyant Soni

DOI: https://doi.org/10.52403/ijhsr.20240501

ABSTRACT

Objectives: The purpose of this study was to record and evaluate the distribution and values of stress and strain generated within the cortical and cancellous bone around an implant with 00 and 150 abutment angulation and two different crest module designs (smooth collar and collar with micro threads) on applying a load of 178 N.
Material and methods: A finite element model of a cross section of maxilla including the residual alveolar ridge and an implant along with an abutment was designed using computer software. The modelled implant-abutment was placed within the alveolar bone of maxilla. A Finite Element stress and strain analysis was done for implants with 00 and 150 angulations with two different crest module designs in maxilla. A similar load was applied to all the models. After this, the software processed the data with the help of recorded values. This data was then subjected to comparative assessment.
Results: Stress distribution of the threaded and smooth collar implants with 0 degree angulation was appeared to be maximum displacement of 0.033mm in the cortical bone surrounding the implant neck on the labial side and strain was maximum in the cancellous bone around the neck of the implant. For the 150 angulation maximum displacement of 0.070mm was observed at the tip in both threaded and smooth collar implants and gradually decreased apically.

Conclusion: Maximum of the stress concentration was observed in the cortical bone around the implant neck irrespective of the abutment angulation and implant design. The stress and strain values of the angulated abutment were 1.5-2 times more as compared to the straight abutment.

Keywords: FEA, abutment angulation, anterior maxilla, implant design, stress, strain

INTRODUCTION

Restoration of partially and completely edentulous jaws is treated and rehabilitates through osseointegrated implant procedures. Over the years, the conducted studies have stated that there are 95% survival rates for mandibular implants and 65-85% for maxillary arch. [1]

Loss of osseointegration and failure in the maxillary arch can be correlated with exceed of mechanical stress limit in the maxillary arch, leads to loss of prosthesis. Factors such as implant design (collar design), implant position and angulation, bone type and direction and magnitude of the occlusal load influences the stress and strain distribution on the implants. [1,2] Bone around the implant is supposed to be greater than 1mm to attain the desired osseointegration. Angulation and the direction of the implant placement is directly related to the bone density and morphology. [4,5]

Bone remodelling such as bone formation and resorption is influenced by the occlusal forces and that determines the stress and strain acting on the implant. Stress and strain is directly related to the forces acting on the implant. [6]

To maintain the bone implant interface, biomechanical factors plays to be the most implant factors such as direction of the force being applied on the implant. Transverse component as always accompanied with the vertical component on the implant. [6, 7].

Implant companies have introduced pre-angled abutments and crest module design (threaded and smooth collar) for the biomechanical factors playing on the implants. Photo elastic and strain gauge measurement were being used to collect the comparative data that were the stress and stain generated through different crest module and angled abutments. [7]

Finite element analysis is a numerical method of structural analysis based on the principle of dividing a structure into a finite number of small elements that are connected to each other at the corner points or nodes. For each different element, its mechanical behaviour can be written as a function of the displacement of the nodes. In other words, FEA is a method where by, instead of seeking a solution function for the entire structure, one formulates the solution functions for each finite element and combines them properly to obtain the solution to the whole structure. FEA was initially developed in the early 1960 to solve structural problems in the aerospace industry. In 1976 Weinstein et al were the first to use FEA in implant dentistry. It is an effective computational tool that has been adapted from the engineering arena to dental implant biomechanics. With FEA, many design feature optimizations have been predicted and will be applied to potential new implant systems in the future. [10, 11]

MATERIALS & METHODS

A 3-D model of anterior maxillary segment in the incisal region with an implant and its abutment was constructed on a personal computer with specification Intel Core i32 - 2330M Processor,2 GB RAM, using software's Hypermesh 13.0 & ANSYS 12.1 for the following four situations those are implants with straight abutment, i.e. 0 degree abutment with smooth/machined collar and micro-threads collar, and implants with 15 degree angled abutment with smooth/machined collar and micro-threads collar.

Steps involved in the study were first finite element modeling comprising construction of FE models with model geometry and modeling of implants with abutments. Second, mesh generation followed by specifying materials properties and application. Finite element analysis with von mises stress and strain analysis.

Then, modelling of the implant with abutment was done with a simulated 13 mm × 3.5 mm Nobel active implant with two different collar designs (smooth collar and micro threads collar) was used for this study. The implants were apposed by cortical bone in the crestal region and by cancellous bone for the remainder of the implant bone interface. The abutments of two different angulations (0, 15 degrees) were connected to implant representing four restorative situations. The abutment was made of same alloy. The implant abutment complex was placed in the middle of the anterior maxilla. The platform of the implant was modelled as being flush with the alveolar ridge surface to mimic effectively a real clinical situation.

Then, specifying material properties were analysed through Young's modulus (elastic modulus) and Poisson's ratio. The cortical bone, cancellous bone and implant with abutment were presumed to be linearly elastic, homogenous and isotropic. The corresponding elastic properties such as Young's Modulus (Є) and Poisson's ratio (δ) of cortical bone, cancellous bone and implant were determined according to literature survey. (Table 1)

Material	Elastic modulus	Poisson's ratio
Cortical bone	13.4 GPa	0.30
Cancellous bone	1.37 GPa	0.31
Titanium alloy	110 GPa	0.33

Table 1: Mechanical properties of different materials used in the Model

The bone implant interface was assumed to be perfect, simulating complete osseointegration. The implant and abutment were assumed to be connected as a single unit. After that load application was done, this is a part of the procedure where an attempt was made to simulate actual clinical situation. It was assumed that force applied to the palatal surface of the maxillary prosthesis would be parallel to the long axis of the mandibular incisor.

Therefore, the load applied near the cingulum area of the prosthesis had an angle of 130 degrees with the long axis of the abutment. The magnitude of the force used was 178 N which is also within the range of mean values reported in the literature.

After applying load on each model, a record of the patterns and values of stress and strain developing around the implant in the bone was displayed using different colours showing different range of stress and strain in cortical and cancellous bone.

RESULT

The study was conducted to analyze the distribution and values of stress and strain generated in the bone around an implant placed in an anterior maxilla using two different abutment angulations (0 degree and 15 degree) and two crest module (Smooth and threaded) designs. Stress and strain were calculated using Von Mises criteria.

Stress distribution

Study of threaded collar implant and smooth collar implant with 0 degree abutment angulation:

1. Maximum displacement of 0.033 mm observed at abutment tip in both threaded and smooth collar implants and gradually decreased apically. (Figure 1)
2. Maximum displacement in the cortical bone =0.019 mm in both threaded and smooth collar implants. (Figure 1) Maximum displacement in cancellous bone =0.017mm in both threaded and smooth collar implants.
3. For cortical bone the maximum Von Mises stress was located around the implant neck and decreased towards the apical end of the implant. The Stresses in crestal bone were less in smooth collar implant as compared to threaded collar implant. For cancellous bone the maximum Von Mises stress was located around the neck portion of the implant

on the labial side. The Stresses in cancellous bone were less in smooth collar implant as compared to threaded collar implant.

4. The maximum stress in the cortical bone with 0 degree abutment and threaded collar was 83.6 MPa and with smooth collar implant was 62.99 MPa respectively.
5. The maximum stress in the cancellous bone with 0 degree abutment and threaded collar was 10.0 MPa and with smooth collar implant was 8.9 MPa respectively. Von Mises stress in the neck of the implant with threaded collar was (110 Mpa) more than the smooth collar implant (93 MPa).

Study of threaded collar implant and smooth collar implant with 15 degree abutment angulation:

1. Maximum displacement of 0.070 mm observed at abutment tip in both threaded and smooth collar implants and gradually decreased apically. (Figure 2)
2. Maximum displacement in the cortical bone = 0.043 mm in both threaded and smooth collar implants. (Figure 2) Maximum displacement in cancellous bone = 0.037 mm in both threaded and smooth collar implants.
3. For cortical bone the maximum Von Mises stress was located around the implant neck and decreased towards the apical end of the implant. The Stresses in crestal bone were less in smooth collar implant as compared to threaded collar implant.
4. For cancellous bone the maximum Von Mises stress was located around the neck portion of the implant on the labial side. The Stresses in cancellous bone were less in smooth collar implant as compared to threaded collar implant.
5. The maximum stress in the cortical bone with 15 degree abutment and threaded collar was 122.61 MPa and with smooth collar implant was 117.40 MPa respectively. The maximum stress in the cancellous bone with 15 degree abutment and threaded collar was 17.25 MPa and with smooth collar implant was 15.25 MPa respectively.

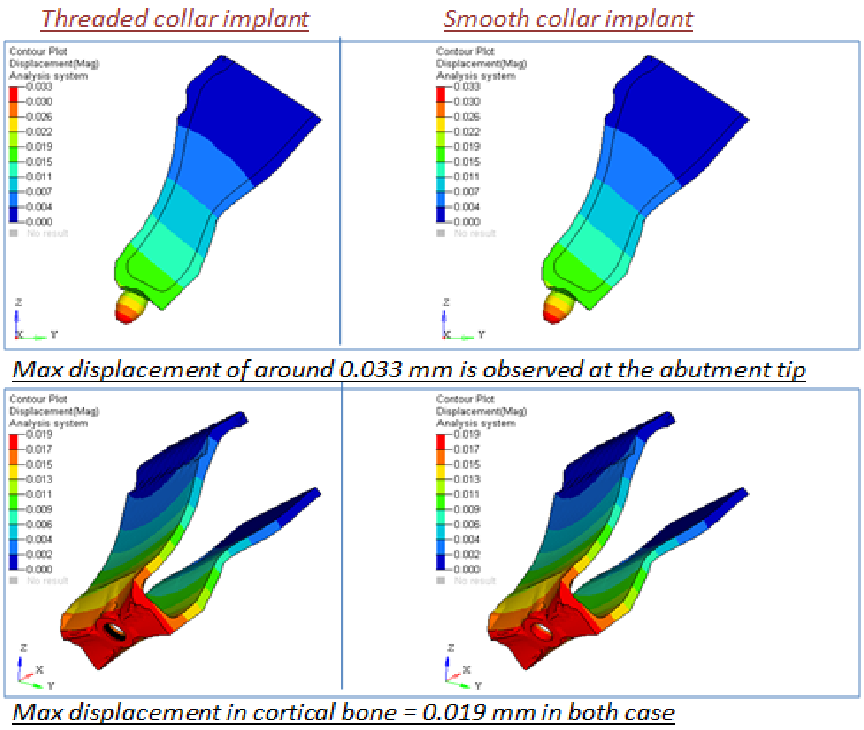

Figure 1: Straight abutment case

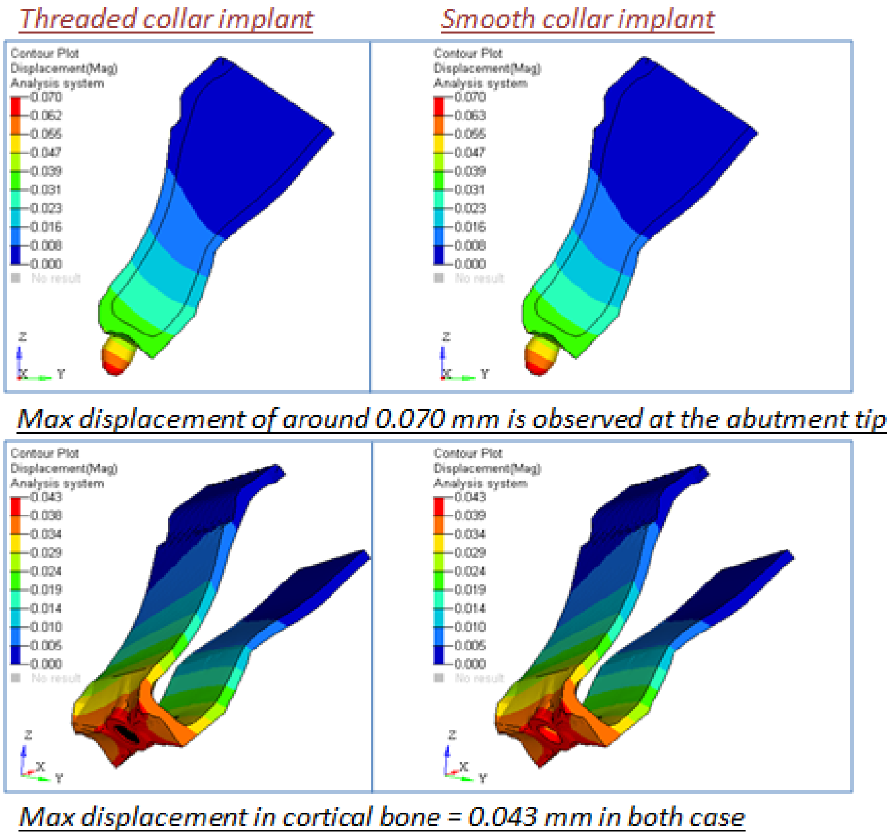

Figure 2: Angulated abutment case

Strain distribution

1. **Study of threaded collar implant and smooth collar implant with 0 degree abutment angulation:**
2. The maximum Von Mises strain was located in the cancellous bone around the neck of the implant on the labial side. For cortical bone the maximum Von Mises strain was located around the neck of implant and decreased towards the apical end of the implant The strain in cortical bone were less in smooth collar implant (0.003 mm/mm) as compared to threaded collar implant (0.004mm/mm.) (Figure 3)
3. For cancellous bone the maximum Von Mises strain was located around the neck portion of the implant on the labial side. The Strain in cancellous bone was less in smooth collar implant (0.0006 mm/mm) as compared to threaded collar implant (0.0007 mm/mm). Von Mises strain in neck portion of implant was (0.001 mm/mm) in both smooth collar and threaded collar.

Study of threaded collar implant and smooth collar implant with 15 degree abutment angulation:

1. For cortical bone the maximum Von Mises strain was located around the neck of implant and decreased towards the apical end of the implant. The strain in cortical bone were less in smooth collar implant(0.005mm/mm) as compared to threaded collar implant (0.006 mm/mm) (Figure 4)
2. For cancellous bone the maximum Von Mises strain was located around the neck portion of the implant on the labial side. The Strain in cancellous bone was less in smooth collar implant (0.001 mm/mm) as compared to threaded collar implant (0.002mm/mm). Von Mises strain in neck portion of implant was (0.002 mm/mm) in both smooth collar and threaded collar.

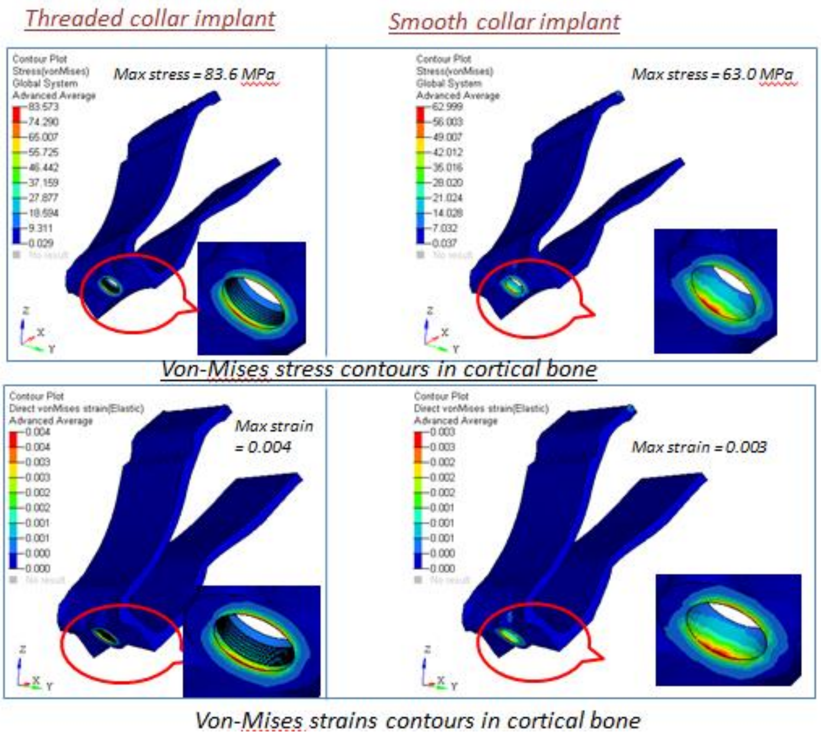

Figure 3: Stress and strain in cortical bone

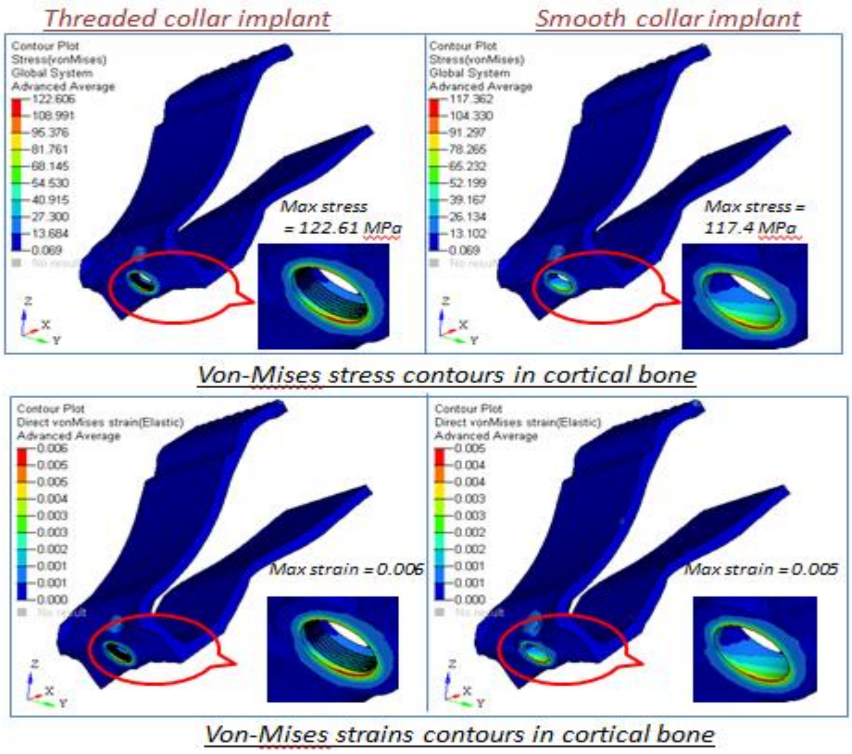

Figure 4: Stress and strain in cortical bone

DISCUSSION

Implant loading is directly related to the success rate of the dental implants and their capability to retain for longer period of time. Restoration of partially and completely edentulous jaws is treated and rehabilitates through osseointegrated implant procedures. Over the years, the conducted studies have stated that there is 95% survival rates for mandibular implants and 65-85% for maxillary arch. [1]

In the anterior maxillary region, when teeth are lost it is difficult to predict the pattern of bone loss. Conventional (zero degree) abutment exhibit difficulty in restoring due to change in bone density and morphology. [3]

Hence, it is essential to stimulate the models design that can replicate the clinical situations. For the present study, a 3-D Finite Element model of the anterior maxilla was selected with cortical bone of the maxilla to be as 1mm layer that depicts the clinical situation. [3, 8]

Clelland et al8 used 15 degree and 20 degree angled abutment in their finite element analysis and found that there was an increase in the magnitude of stress and strain as the abutment angulation increased. All reported stresses and strains were found to be within the physiologic zone as reported by Frost. In another study Brosh et al[9] used 15 degree and 25 degree angulated abutment and calculated the stress and strain along the implant bone interface using strain gauge and photoelastic measurements. They also concluded that angled abutment increased the amount of stress and strain but within the physiologic limit.

Sethi et al[4] and Eger et al[5] compared the effect of angled and straight abutments on clinical outcomes. Implants were placed in different sites of maxilla and mandible and were restored using abutments of varying angulations. They concluded that endosseous implants placed at unfavourable angles may be restored with angled abutments without compromise of function or aesthetics.

Mandibular anterior teeth are guided by are palatal surfaces of the maxillary anterior teeth in their protrusive and lateral excursions. Approximately 130 degree is the inter-incisal mean angle. [3]

In the present study, a force of 178 N was applied at the cingulum area of the abutment at an angle of 130 degree to the long axis of the abutment, which stimulates clinical situation during function.

According to the obtained results, the maximum equivalent stress/strain elicited by a force of 178 N on this implant bone model appeared to increase linearly with an increase of the abutment angulation. Near the implant neck, it was observed that there was concentration of maximum Von Mises stresses. The maximum stress observed for the thread collar with 15 degree abutment was 122.6MPa in the cortical bone and only 17.25Mpa in the cancellous bone.

As the abutment angulation increases the maximum equivalent Von Mises strain also increased. The cancellous bone around the middle portion of the implant concentrated with maximum strain followed by implant apex. Threaded collar implant with 15 degree abutment depicted Maximum Von Mises strain of 0.006mm/mm. Each model had the same pattern of strain distribution.

Limitations of Finite Element Method:

- Even though Finite Element Method is an accurate and precise method for analyzing structures, the present study had certain limitations:
- Real clinical situation experiences some movement between the implant and the bone during loading from different directions which was not present in this study.
- It was considered that the implant was totally osseointegrated, which is not the case in the clinical situations.
- The bone clinically is non-homogenous and anisotropic but in the present study it was considered that the cancellous bone, cortical bone and the implant was isotropic and homogenous.
- To achieve more realistic situation, advanced digital imaging techniques can be used to model bone geometry in greater detail; the anisotropic and non-homogenous nature of the material needs to be considered; and applied boundary conditions must be refined. In addition, modelling of bone-implant interface should incorporate the actual osseointegration contact area in cortical bone as well as the detailed 3-dimensional trabecular bone contact pattern.

CONCLUSION

Within the limitations of the methodology, the results of static load and linear analysis support the following conclusion:

- Maximum amount of stress concentration was observed in the cortical bone around the implant neck irrespective of the abutment angulations and collar design.
- Maximum amount of strain concentration was observed in the cancellous bone around the crestal portion of the implant.
- An angulated abutment contributes in the 1.5-2.0 times more stress and strain than straight abutments.
- 15 degree abutments with threaded collar generated maximum stress and strain around the implant and zero degree abutment with smooth collar produced least amount of stress and strain.
- Combined with proper diagnosis and treatment planning, angulated abutments can be used in anterior maxilla with proper selection of crest module design to restore the implants placed at unfavourable angles without compromise of function.
- Angulated abutment increased the stress and strain and smooth collar reduced crestal stress and stain as compared to threaded collar. Reduced abutment diameter (platform switching) resulted in less stress translated to the crestal bone.

Further researches are required to demonstrate the stress/strain values with more variables such as bone density, interface conditions and degree of osseointegration. Finite Element Analysis is based on mathematical calculations, while living tissues are beyond the confines of set parameters and values since biology is not a computable entity. Therefore, Finite Element Analysis should not be considered as a sole means of understanding the behaviour of a geometrical structure in a given environment. Actual experimental techniques and clinical trials should follow Finite Element Analysis to establish the true nature of the biologic system.

Declaration by Authors

Ethical Approval: Not Required

Acknowledgement: None

Source of Funding: None

Conflict of Interest: The authors declare no conflict of interest.

REFERENCES

1. Chang S.H, Lin C.L, Chang W.J, and Kuo Y.C. 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; 115:408-416.
2. Papavasiliou G, Kamposiora P, Bayne S.C, and Felton D.A. Three-dimensional finite element analysis of stress distribution around single tooth implants as a function of bony support, prosthesis type, and loading during function. J Prosthet Dent 1996; 76:633-40.
3. Saab X.E., Griggs J.A, Powers J.M, and Engelmeier R.L. Effect of abutment angulation on the strain on the bone around an implant in the anterior maxilla: A finite element study. J Prosthet Dent 2007; 97:85-92.
4. Sethi A, Kaus T, and Sochor P. The use of angulated abutments in implant dentistry: Five-year clinical results of an ongoing prospective study. Int J Oral Maxillofac Implants 2000; 15:801-810.
5. Eger D.E, Gunsolley J.C, and Feldman S. Comparison of angled and standard abutments and their effect on clinical outcomes: A preliminary report. Int J Oral Maxillofac Implants 2000; 15:819-823.
6. Hsu M.L, Ching Chen F, Kao H.C, and Kung Cheng C. Influence of off axis loading of an anterior maxillary implant: A 3-dimensional finite element analysis. Int J Oral Maxillofac Implants 2007; 22:301-309.
7. Kaukinen J.A, Edge M.J, and Lang B.R. The influence of occlusal design on simulated masticatory forces transferred to implant retained prosthesis and supporting bone. J Prosthet Dent 1996; 76:50-5
8. Clelland N.L, Lee J.K, Bimbenet O.C, and Brantley W.A. A three-dimensional finite element stress analysis of angled abutments

for an implant placed in the anterior maxilla. J Prosthod 1995; 4:95-100.

9. Brosh T, Pilo R, and Sudai D. The influence of abutment angulation on strains and stresses along the implant/bone interface: Comparison between two experimental techniques. J Prosthet Dent 1998; 79:328-34.
10. Geng J.P, Tan K.B, and Liu G.R. Application of finite element analysis in implant dentistry: A review of literature. J Prosthet Dent 2001; 85:585-98.
11. G. Vasudeva. Finite element analysis: A boon to dental research. The internet journal of dental sciences 2009, Vol 6, No. 2.
12. Misch C E. Contemporary Implant Dentistry. Mosby. 3rd edition. Pages 548-50.
13. Helkimo E, Carlsson G.E and Helkimo M. Bite force and state of dentition. Acta Odont. Scand. 1976; 35, 297-303.
14. Borchers L and Reichart P. Three-dimensional stress distribution around a dental implant at different stages of interface development. J Dent Res 1983, 62(2):155-59
15. Rieger M.R, Mayberry M, and Brose M.O. Finite element analysis of six endosseous implants. J Prosthet Dent 1990; 63: 671-6.
16. Meijer H.J.A, Kuiper J.H, Starmans F.J.M, and Bosman F. Stress distribution around dental implants: Influence of superstructure, length of implants, and height of mandible. J Prosthet Dent 1992; 68:96-102.
17. Waltimo A., Konomen M. Bite force on single as opposed to all maxillary front teeth. Scand J Dent Res 1994; 102:372-5.
18. Razavi R., Zena R.B., Khan Z. and Gould A.R. Anatomic site evaluation of edentulous maxillae for dental implant placement. J Prosthod 1995; 4:90-94.
19. Van Zyl Paul P, Grundling N.L., Jooste C.H, and Terblanche E. Three-dimensional finite element model of a human mandible incorporating six osseointegrated implants for stress analysis of mandibular cantilever prosthesis. Int J Oral Maxillofac Implants 1995; 10:51-57.
20. Benzing U.R, Gall H., and Weber H. Biomechanical aspects of two different implant prosthetic concepts for edentulous maxillae. Int J oral Maxillofac Implants 1995; 10:188-198.
21. Sertgoz A. Finite element analysis- Study of the effect of superstructure material on stress distribution in an implant supported fixed prosthesis. Int J Prosthodont 1997; 10:19-27.
22. Oosterwyck V.H, Duyck J, Sloten J.V, Perre G, Cooman M and Lievens S et al. The influence of bonemechanical properties and implant fixation upon bone loading around oral implants. Clin Oral Imp Res 1998; 9: 407-418.
23. Teixeira E.R, Sato Y., Akagawa Y, and Shindoi N. A comparative evaluation of mandibular finite element models with different lengths and elements for implant biomechanics. Journal of Oral Rehabilitation 1998; 25:299-303.
24. Stegaroiu R, Sato T., Kusakari H, and Miyakawa O. Influence of restoration type on stress distribution in bone around implants: A three-dimensional finite element analysis. Int J Oral Maxillofac Implants 1998; 13:82-90.
25. Kenney R, and Richards M.W. Photoelastic stress patterns produced by implant retained overdentures. J Prosthet Dent 1998; 80:559-64.
26. O' Mahony A.M, Williams J.L, and Spencer P. Anisotropic elasticity of cortical and cancellous bone in the posterior mandible increases periimplant stress and strain under oblique loading. Clin. Oral Impl. Res. 2001; 12:648-657.
27. Norton M.R, and Gamble C. Bone classification: an objective scale of bone density using the computerized tomography scan. Clin. Oral Impl. Res. 2000; 12:79-84.
28. Gross M.D, Nissan J, and Samuelc R. Stress distribution around maxillary implants in anatomic photoelastic models of varying geometry. Part I. J Prosthet Dent 2001; 85:442-9.

How to cite this article: Dushyant Soni, Nitin Modi, Dron Lakhani, Rajshree Bhandari, Amrit Assi, Anuva Aseri et.al. A 3-D finite analysis of stress and strain around a dental implant with different abutment angulations and two crest module design. *Int J Health Sci Res.* 2024; 14(5):1-9. DOI: *10.52403/ijhsr.20240501*

******