Scopus®

Original Article

IJCRR

Section: Healthcare
ISI Impact Factor
(2019-20): 1.628
IC Value (2019): 90.81
SJIF (2020) = 7.893

Copyright@IJCRR

Analysis of Stress Pattern in the Bone Around Variable Thread Root form Implant of Different Diameters Under Axial and Non-Axial Loading

Mohammed Sadique[1], Jayashree Mohan[2], Sunantha Jayachandran[3], Sasikala[3], Santhosh Kumar V[4], Sasikanth Venkatesan[5]

[1]Clinical Practitioner, Kingdom of Saudi Arabia; [2]Professor and Head, Department of Prosthodontics, Vinayaka Missions Sankarachariyar Dental College, Salem, Tamilnadu, India; [3]Reader, Department of Prosthodontics, Vinayaka Missions Sankarachariyar Dental College, Salem, Tamilnadu, India; [4]Senior Lecturer, Department of Prosthodontics, Vinayaka Missions Sankarachariyar Dental College, Salem, Tamilnadu, India; [5]PG Student, Department of Prosthodontics, Vinayaka Missions Sankarachariyar Dental College, Salem, Tamilnadu, India.

ABSTRACT

Introduction: Unsuitable loading on implant causes extreme stress in the bone around the implant and may result in bone resorption.

Objectives: This study was carried out to analyze the stress pattern in the bone around the variable thread root form implant of different diameters under axial and non-axial loading.

Method: A three-dimensional Finite Element Method was used to study the influence of diameter of an implant on strain and stress distribution design in the cancellous and cortical bone through non-axial and axial loading. Using a computed tomography (CT) a geometric model of posterior mandibular area was created. Two implants of different diameters ie 3.3mm and 3.75mm but having the same length of 11.5mm was modelled and embedded in the section of the bone. The vertical load of 100 N and a horizontal load of 50 N from buccolingual and mesiodistal directions were applied on to the abutment.

Results: Implant with increased diameter appeared to distribute stress in a more uniform pattern around the implant. Implant with lesser diameter showed high-stress pattern during axial as well as non-axial loading. The highest quantity of stress concentration was detected in the cortical bone regardless of the magnitude and direction of loading. Higher stress was generated in the bone-implant system during non-axial loading. There was the favourable distribution of stress and strain pattern during axial loading.

Conclusion: It was concluded that axial loading of an implant appeared to be a favourable direction of loading and does not hamper longevity.

Key Words: Dental implant, Stress, Strain, Cortical bone, Cancellous bone

INTRODUCTION

An osseointegrated dental implant creates the beginning of a new era in oral rehabilitation.[1] The osseointegrated dental implants showed a higher success rate for more than 20 years worldwide.[2] Irrespective of high success rates reported to date, implant failures do occur. Implants can fail during both the unloaded healing phase as well as the functional phase but usually for different reasons.

How stress is transferred to the surrounding bone is a major factor for the accomplishment or failure of a dental implant. Unsuitable loading creates extreme stress in the bone around the implant and may result in bone resorption. It is this biomechanical interaction at the implant-bone interface, which studies have shown that the close apposition of bone to the titanium implant is an essential feature that allows the transfer of stress from the implant to the bone without any considerable relative motion or abrasion.[3] In case of a smooth surface implant, a strong bond between the implant surface and bone is not present for satisfactory performance. However, an implant surface with thread geometry may have a beneficial interlocking effect to achieve osseointegration, resulting in favourable stress distribution.[3]

Thus in this study surface topography of a root form variable thread implant is chosen as a parameter to understand the effect of loads on the bone through the implant. The finite element method (FEM) agreements numerous advantages including the precise symbol of complex geometries, easy model modification and depiction of the internal state of stress and other mechanical quantities. FEM can simulate the intricate details of the bond between bone and implant interface thereby giving us the understanding of the loads transferred from the implant to the bone through the interface.[4,5]

This study was done to understand the behaviour of bone around variable thread root form dental implant with two different diameters when it is loaded under axial and non-axial directions. Three-dimensional finite element analysis was used to carry out the study.

MATERIALS AND METHODS

Implants with abutments were modelled at Chennai, using a computer with specifications Pentium IV, 256 MB RAM. A finite element program, ANSYS version 8.0 was used for the study. The implant was assumed to be placed in the region of the first molar of the mandible.[6] The models were provided in close approximation to the in vivo geometry.

Modelling of the bone
Initially, computerized tomography (CT scan) of five different regions of actual human mandible figure 1 was obtained.[4,7] In this study, a section of the mandible of the lower first molar region was taken from the CT scan to generate the model of bone. The Bone is scanned at various sections at regular intervals of 0.5 mm. These scanned images are then imported into Pro/E sot wares to various offset planes.

Modelling of the implant with an abutment
A three-dimensional finite element model of root form thread type Implant System (ADIN dental implant system, Israel) was generated. The dimensions of the implant were 3.3 mm diameter and 11.5 mm in length. The implant considered was the swell type which is a straight parallel walled slightly tapered implant with variable thread design. The implant is positioned halfway between the mesiodistal length of the section of the mandible. The implant was inserted according to the section of the bone with a 5-degree lingual inclination.[8-10]

Suitable abutment was screwed on to the implant. The Finite Element Model of implant with the abutment, which could be created within the limitations of the study, is shown in figure 1.
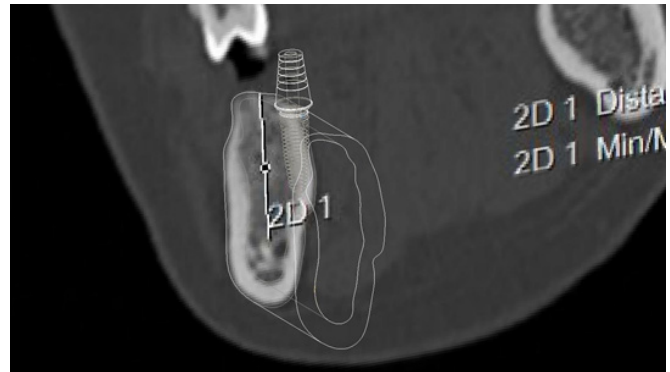

Figure 1: Modelling of bone and implant.

Another implant with 3.75 mm diameter and 11.5mm length was also modelled and placed in a similar section of the mandible for comparison. The section of bone containing implant with diameter 3.3mm was considered as group I and the one with diameter 3.75mm as group II.

Mesh generation
The 3-dimensional finite element model represents the geometric model was meshed using Ansys Pre-processor (ANSYS version 8.0 software). The type of meshing is free meshing because the model is not geometrically symmetric. The element size (SOLID 187) was designated rendering to default settings. The type of element appropriate for this specific study was 10 noded tetrahedron element which was allocated 3 degrees of freedom per node, namely translation in the x, y and z angulation. The elements were created so that their size aspect ratio would produce sensible solution accuracy.[11,12]

Specifying material properties
For the execution and accurate analysis of the programme and interpretation of the results, two material properties were utilized i.e. Young's modulus and Poisson's ratio. The corresponding elastic properties such as Young's Modulus (€) and Poisson's ratio (δ) of cortical bone, cancellous bone and implant were determined according to the literature survey (Table 1).[8,9]

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

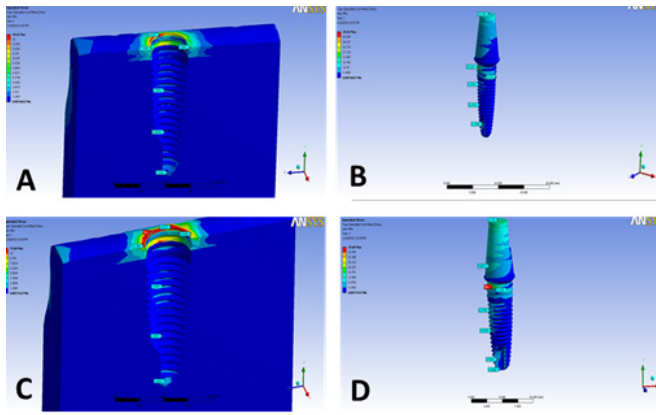| Material	Young's modulus (Mpa)	Poisson's ratio
Cortical bone	13700	0.30
Cancellous bone	1370	0.30
Implant (Titanium alloy)	110000	0.35

Figure 3: Fine element analysis of model 1 (Von Mises Stress during Axial Loading (Load 1) Group 1: A. Cortical and Cancellous Bone, B. Implant; Group 2: C. Cortical and Cancellous Bone, D. Implant.

Statistical analysis

A patient-level statistical analysis was performed for each of the parameters. A Kolmogorov Smirnov test was used to test normal distribution of the data and revealed evidence for normality. The mean values and standard deviations (mean ± SDs) for the clinical variables were calculated for each treatment, based on the subject as the statistical unit. The significance of the difference between the group was evaluated with the student's t-test. Data were analyzed with the unpaired observations. The level of significance was set at 0.05 and 0.01.

RESULTS

LOAD 1

The results of the study showed that at Cortical bone a significant difference in the mean stress was found between group 1 (7.18±0.60) and group 2 (5.84±0.78). Similarly, significant difference in the mean strain was found between group 1 (6.38±1.70) and group 2 (4.22±0.98) at Cortical bone. Hence it was concluded that Group 2 is most effective than Group 1 regarding stress and strain with respect to Cortical Bone with regard to Load 1 - Axial - (100N).

It was also found from the analysis that Group 2 is having lesser stress and strain than Group 1 in all the three parts of cancellous Bone. Hence it is concluded that Group2 is most effective than Group 1 regarding stress and strain with respect to all the three parts of cancellous Bone with regard to Load1 - Axial - (100N).

The significant difference in the mean stress and strain among the two groups with respect to Coronal part, Middle part and Epical part of Implant was analyzed. In the Coronal part, significant difference in the mean stress and strain among the two groups was found as their p-value is less than 0.05. Similarly, significant difference in the mean stress and

strain was found between group 1 and group 2 as their p-value is less than 0.05 at Middle part as well as in Apical part of Implant.

It is found from the analysis that Group 2 is having lesser stress and strain than Group 1 in all the three parts of the Implant. Hence it is concluded that Group2 is most effective than Group 1 regarding stress and strain with respect to all the three parts of the Implant concerning Load1 - Axial - (100N).

LOAD 2:

In the Cortical bone, the significant difference in the mean stress was found between group 1 (22.30±4.87) and group 2 (15.88±3.03). Similarly, a significant difference in the mean strain was found between group 1 (10.30±3.08) and group 2 (5.68±2.57). Hence it is concluded that Group2 is most effective than Group 1 regarding stress and strain with respect to Cortical Bone with regard to Load 2- Bucco- Lingual -(50N).

Similarly, significant difference in the mean stress and strain was found between group 1 and group 2 at the Middle part as well as in the Apical part of Cancellous Bone. Hence it was concluded that Group2 is most effective than Group 1 regarding stress and strain with respect to all the three parts of cancellous Bone with regard to Load 2- Bucco- Lingual -(50N)

The significant difference in the mean stress and strain among the two groups with respect to Coronal part, Middle part and Epical part of Cancellous bone was analyzed and the results are given in Table 5. In the Coronal part, significant difference in the mean stress and strain among the two groups was found as their p-value is less than 0.05.

The significant difference in the mean stress and strain among the two groups with respect to Coronal part, Middle part and Epical part of Implant was analyzed. In the Coronal part, significant difference in the mean stress and strain among the two groups was found as. Similarly, significant difference in the mean stress was found between group 1 and group 2 at the Middle part in addition to the Apical part of Implant. Significant difference in the mean strain was found between group 1 and group 2 at implant as their p-value is less than 0.05 but not at Middle part as its p-value is greater than 0.05.

It is found from the analysis that Group 2 is having lesser stress than Group 1 in all the three parts of the Implant. Hence it is concluded that Group2 is most effective than Group 1 regarding stress and strain with respect to all the three parts of the Implant with regard to Load 2- Bucco- Lingual -(50N).

LOAD 3:

In Cortical bone, the significant difference in the mean stress was found between group 1 (31.8 ±6.95) and group

2 (19.08±5.38) as their p-value is less than 0.05. Similarly, significant difference in the mean strain was found between group 1 and group 2 as their p-value is less than 0.05 at Cortical bone. It is found from the analysis that Group 2 is having lesser stress and strain than Group 1. Hence it is concluded that Group2 is most effective than Group 1 regarding stress and strain with respect to Cortical Bone with regard to Load 3- Mesio-Distal -(50N).

In the Cortical bone, the significant difference in the mean stress was found between group 1 and group 2 as their p-value is less than 0.05. Similarly, significant difference in the mean strain was found between group 1 and group 2 as their p-value is less than 0.05 at Cortical bone. It is found from the analysis that Group 2 is having lesser stress and strain than Group 1. Hence it is concluded that Group2 is most effective than Group 1 regarding stress and strain with respect to Cortical Bone with regard to Load 3- Mesio-Distal -(50N).

The significant difference in the mean stress and strain among the two groups with respect to Coronal part, Middle part and Epical part of Cancellous bone was analyzed In Coronal part significant difference in the mean stress among the two groups was found as their p-value is less than 0.05. and not on strain as its p-value is greater than 0.05. Similarly, significant difference in the mean stress among the two groups was found as their p-value is less than 0.05. and not on strain as its p-value is greater than 0.05 at Middle part as well as in Apical part of Cancellous Bone

It is found from the analysis that Group 2 is having lesser stress and strain than Group 1 in all the three parts of cancellous Bone. Hence it is concluded that Group2 is most effective than Group 1 regarding stress and strain with respect to all the three parts of cancellous Bone with regard to Load 3- Mesio-Distal -(50N).

The significant difference in the mean stress and strain among the two groups with respect to Coronal part, Middle part and Epical part of Implant was analyzed. In the Coronal part, significant difference in the mean stress and strain among the two groups was found as their p-value is less than 0.05. Similarly, significant difference in the mean stress and strain was found between group 1 and group 2 as their p-value is less than 0.05 at Middle part as well as in Apical part of Implant

It is found from the analysis that Group 2 is having lesser stress and strain than Group 1 in all the three parts of the Implant. Hence it is concluded that Group2 is most effective than Group 1 regarding stress and strain with respect to all the three parts of the Implant with regard to Load 3- Mesio-Distal -(50N).

DISCUSSION

For various implant types, the probability of implant treatment is reinforced by numerous clinical studies reporting success rates greater than 90%.[13,14] However, implant failures do occur predominantly because of biomechanical factors. These factors depend on: a) the mechanical stiffness of the implant and bone shape' length, diameter and surface topography of implant,[15] the magnitude, directions, locations and modes of action of the functional and parafunctional forces on the restored implant.

For complicated geometries, such as mandible it is very difficult to achieve an analytical solution. Therefore, the use of numerical methods such as Finite Element Method is required. Vertical and transverse loads from mastication persuade axial forces and bending moments that result in stress inclines in the implant as well as in the bone. Finite Element Method allows us to safely assess stress delivery in the contact area of the implants with cortical bone and surrounding the apex of the implants in trabecular bone.[16,17]

To suit the aims of this study 3-D Finite Element Method, which can be used appropriately in asymmetric situations such as in the mandible, was used to evaluate the stresses/strains in the bone around the implant.[18] In our study, a segment of bone was modeled to represent the posterior area of the mandible because imitation of the whole mandibular body is very extravagant. The cortical bone, cancellous bone and implant with abutment were presumed to be linearly elastic, homogenous and isotropic.[19] In this study, the constraints at the end of the bone segment and force application on top of the abutment approximated only roughly the complex balance between masticatory forces and their reactions.

The tendency of stress concentration around the implant neck at the cortical bone level was evident in all of the models. The reason may be because of great modulus of elasticity of cortical bone (E=13,700MPa), which delivers more rigidity and thus the additional capability to withstand stress. On vertical loading, the stress generated in group 2 was comparatively less than stress generated in group 1. The probable reason could be the increase in diameter. Stress distribution in the surrounding bone increases with the increased osseointegrated surface area. A minimum amount of stress was developed during axial loading as compared to the horizontal direction of since load is applied parallel to the long axis of the implant, and capacity of the cortical and cancellous bone to withstand stress raises.[20,21]

During buccolingual loading, stress seen in the cortical bone was less with group2 compared to group1. This is because of the decrease in the cross-sectional diameter of group1 implant. When a load is applied perpendicular to its long axis, the short implant will deform more when compared to the

implant of greater diameter. Thus causing generation of greater stress in the cortical bone.

During buccolingual and mesiodistal loading, the stress was found to be more in group1 on the lingual and distal sides. The probable reason could be the direction of force perpendicular to the long axis.[22,23] Because of the low modulus of elasticity of cancellous bone, the load-bearing capacity decreases while elasticity increases. Thus, more strain can be seen especially during horizontal loading. Von Mises strain rate of cancellous bone through axial loading is fewer associated to the value acquired during horizontal loading even with double the load i.e. 100 N. This is because during axial loading the stress was distributed to all sides of the cancellous bone whereas in horizontal loading the stress was concentrated on one side of the cancellous bone ie opposite to the direction of the force.

Regardless of the course and magnitude of loading, an implant with abutment withstood extreme amount of stress equated to any other component of the model. The likely reason could be its great elastic modulus (E=110,000 MPa), which is 8 times the elastic modulus of cortical bone (E=13700 MPa) and 80 times the elastic modulus of cancellous bone (E=1370 MPa). During axial loading, the stress generated within the implant was least as compared to the stress generated during buccolingual and mesiodistal loading. The reason is that the direction of load along the long axis of the implant provides a maximum cross-sectional area to withstand the stress. There was a slight reduction in the stress levels in group 2.

Loading of an implant fixed with an abutment in a horizontal direction induces a certain amount of deformation (force-length) into the system and causes bending of the abutment. This bending of abutment reduces with increasing distance from the loading point. Because of the continuum of abutment and implants, the abutment bending causes displacement of the implants. The displacement of implant based on the magnitude and resistance to bending of all components including the bone, implant and abutment.[22] In mesiodistal loading the resistance obtainable by the supportive bone was slighter than during loading and buccolingual loading hence additional stress was seen on the mesiodistal side.

Clinical implications
It is clear from the results of the study that implant with lesser diameter showed high stress around the bone and implant. Therefore the diameter of the implant should be considered as an important factor for implant longevity. Non-axial loading has been related to marginal bone loss, failure of osseointegration and even failure of the implant.[22,23] Hence, occlusal contacts that distribute axial stress should be incorporated in the prosthesis. Through eccentric movements, the implant-supported prosthesis should allow only negligible functional contact to evade forces from the non-axial direction.

CONCLUSION

Thus, it can be concluded that increased diameter of an implant improves the stress transfer homogenization along with the bone-implant interface by providing the greater surface area as compared to the implant with lesser diameter. It was also concluded that axial loading of an implant appeared to be a favourable direction of loading and does not hamper longevity.

Authors contribution

1. Dr. Mohammed Sadique- investigation

2. Dr. Jayashree Mohan- data collection

3. Dr. Sunantha Jayachandran- investigation

4. Dr. Sasikala- analaysis, evaluation

5. Dr. Santhosh Kumar- manuscript writing

6. Dr. Sasikanth Venkatesan,- editing

REFERENCES

1. Davis DM. The shift in the therapeutic paradigm: Osseointegration. J Prosthet Dent 1998; 79:37-42.

2. Wyatt CCL, Zarb GA. Treatment outcomes of patients with implant-supported fixed partial prostheses. Int J Oral Maxillofac Impl 1998; 13:204-211.

3. Skalak R. Biomechanical considerations in the osseointegrated prosthesis. J Prosthet Dent 1983; 49:843-848.

4. Geng JP, Tan KBC, Liu GR. Application of finite element analysis in implant dentistry: A review of the literature. J Prosthet Dent 2001;85:585-598.

5. Lin CL, Wang JC. Non-linear FEA of a splinted implant with various connectors and occlusal forces. Int J Ort Med 2003;18:331-340

6. Zyl PPV, Grundling NL, Jooste CH. Three-dimensional finite element model of a human mandible incorporating six osseointegrated implants for stress analysis of mandibular cantilever prostheses. Int J Oral Maxell Impl 1995;10:51-57.

7. Pierrsnard L, Hure G, Barquins M and Chappard D. Two dental implants designed for immediate loading: An FEA analysis. Int J Oral Maxill Impl 2002;17:353-362.

8. Miejer HJA, Starmans FJM, Steen WHA, Bosman F. A three dimensional Finite Element Analysis of bone around dental implants in an edentulous human mandible. Archives Oral Bio 1993;6:491-496.

9. Sato Y, Teixeira ER, Tsuga K, Shindoi N. The effectiveness of element downsizing on a three-dimensional finite element model of bone trabeculae in implant biomechanics. J Oral Rehab 1999; 26: 640-643.

10. Rieger HR, Mayberry M, Brose MO. Finite Element survey of six endosseous implants. J Prosthet Dent 1990; 63:671-676.

11. Vollmer D, Meyer U, Joos U, Vegh A, Piffko J. Experimental and finite element study of a human mandible. Eur Ass Cranio-Maxill Sur 2000; 28:91-96

12. Brosh T, Pilo R and Sudai D. The influence of abutment angulation on strains and stresses along with the implant/bone interface: Comparison between two experimental techniques. J Prosthet Dent 1998; 79:328-334.

13. Richter EJ. In vivo vertical forces on implants. Int J Oral Maxillfac Impl 1995; 10:99-107.

14. Himmlova L, Dostalova T, Kacovsky A, Konvickova S. Influence of implant length and diameter on stress distribution: A Finite Element Analysis. J Prosthet Dent 2004; 91:20-25.

15. Lang LA, Byungsik K, Wang RF, Lang BR. Finite Element Analysis to determine implant preload. J Prosthet Dent 2003;90:539-546.

16. Cehreli MC, Iplikcioglu H. In vitro strain GA of axial and off-axial loading on implant-supported FPD. Implant Dent 2002;11:286-292.

17. Tada S, Stegaroiu R, Kitamura E, Miyakawa O, Kusakari H. Influence of implant design and bone quality on stress/strain distribution in bone around implants: A three-dimensional Finite Element Analysis. Int J Oral Maxil Impl. 2003; 18:357-368.

18. Occlusal forces during chewing and swallowing as measured by sound transmission. J Prosthet Dent 1981; 46:443-449.

19. Cochran DL. Comparison of Endosseous Dental Implant surfaces. J Period 1999;70(12):1523-1539.

20. Cook SD, Weinstein AL, Klawitter JJ. Three-dimensional finite element analysis of a porous rooted Co-Cr-Mo alloy dental implant. J Dent Res 1982;61:25-29.

21. Hobkirk JA, Havthoulas TK. The influence of mandibular deformation, implant numbers and loading position on detected forces in abutments supporting fixed implant superstructures. J Prosthet Dent 1998;80:169-174.

22. Rho JY, Ashman RB, Turner CH. Young's modulus of trabecular and cortical bone material: ultrasonic and microtensile measurements. J Biomech 1993;26(2):111-119.

23. Wadamoto M, Akagawa Y, Sato Y, Kubo T. The 3D bone interface of an osseointegrated implant I: A morphometric evaluation in initial healing. J Prosthet Dent 1996;76:170-175.