ISSN- O: 2458 - 868X, ISSN –P: 2458 - 8687

International Journal of Medical Science and Innovative Research (IJMSIR)

IJMSIR : A Medical Publication Hub

Available Online at: www.ijmsir.com

Volume – 3, Issue –1, January - 2018, Page No. : 112 - 122

Effect of non-axial implant loading on maxillary anterior region with different bone density. A 3-dimensional finite element analysis.

Dr. Gulab Chand Baid[1], Dr. Vivek Lath[2], Dr. Nitin A Vaidya[3], Dr. Vidya Vaidya[4], Dr.Saumya Sharma[5], Dr.Neeraj Gupta[6]

[1,2]Reader, Department of Prosthodontics, Maitri College of Dentistry And Research Centre, Anjora, Durg, Chhattisgarh.

[3]Professor and Head, Department of Prosthodontics, Maitri College of Dentistry And Research Centre, Anjora, Durg, Chhattisgarh.

[4,5]Professor, Department of Prosthodontics, Maitri College of Dentistry And Research Centre, Anjora, Durg, Chhattisgarh.

[6]Senior Lecturer, Department of Prosthodontics, Maitri College of Dentistry And Research Centre, Anjora, Durg, Chhattisgarh.

Correspondence Author: Dr. Vivek Lath**,** Reader, Department of Prosthodontics, Maitri College of Dentistry And Research Centre, Anjora, Durg, Chhattisgarh.

Type of publication: Original Research Paper

Conflicts of Interest: Nil

Abstract

Background: Biochemical factors play a substantial role in implant success or failure. Application of occlusal forces induces stresses and strain within the implant prosthesis complex and affects the bone remodelling process around the implant. Off axis force which is common during normal mastication would appear to induce more stress than does axial force. In case where aesthetics require tooth overlap in the anterior region, off axis loading of implant is usually unavoidable.

Objectives: the purpose of this study is to evaluate the influences of stress and strain distribution in bone around anterior maxillary implant using 2 type of bone and 3 different loading angle

Materials and methods: A 3 D finite element model was made of the anterior maxilla using the details from a CT scan, using computer software (ANSYS 11). A simulated 13 x 4.2 mm implant was placed in the centre of the bone and it suprerstructure were created. Six different testing conditions incorporating 2 type of cancellous bone(high and low density) under 3 different loading angles(0, 30 , 60 degree) relative to long axis of the implant were applied.. The stress and strain generated in the cortical and cancellous bone around the implant was recorded and evaluated with the help of ANSYS.

Results: The maximum equivalent stress/strain increased linearly with the increase of loading angle. In addition to loading angle, bone quality also influenced resultant stress distribution. For low density bone model a substantial strain in the cancellous bone was found.

Conclusion: to achieve a favorable prognosis under off axis loading of an anterior maxillary implant, careful case selection for appropriate bone quality and precise occlusal adjustment should be attempted to optimally direct occlusal force toward the long axis of the implant.

Keywords: biomechanics, dental implants, bone quality, finite element analysis, premaxilla, off-axis.

Introduction

Biomechanical factors play an important role in maintaining the bone implant interface, in which one of the important factors is force direction. Occlusal force always features a transverse component in addition to a vertical component[1, 2].Osseointegrated dental implants are considered to be a viable treatment option for restoring partially and completely edentulous jaws. Long term clinical studies have reported 95% survival for mandibular implants and 65-85% survival for maxillary implants[3-5].

The higher rates of implant failures associated with maxillary therapy may be related to the biomechanical complications of dental implants and restorations, where mechanical stress may exceed the limits of physiological tolerance leading to loss of osseointegration. The stress/strain distribution may be influenced by various parameters such as the implant design, implant position and angulation, bone type and the magnitude and direction of the occlusal load[3, 4].To optimize osseointegration, a bone thickness of greater than 1mm around the implant is desirable[3]. The anatomy of the jaws and the morphology of the residual ridges determine the orientation and angulation of implant placement. Whereas, the position and morphology of the teeth are determined by esthetic and functional considerations. In the majority of situations, there is a difference between the long axis of the implant and the long axis of the planned tooth replacement[6, 7].The application of occlusal forces induces stress and strain within the implant prosthesis complex and affects the bone remodeling process around implants. The amount of strain/ stress on bone is directly related to the amount of occlusal force applied through the implant supported prosthesis. According to Frost's mechanostat concept, bone fractures at 10,000 to 20,000 microstrains. However, just 20% to 40% of the amount of strain required for fracture (i.e. 4,000 microstrains) may trigger cytokine to activate a resorptive response. The interaction of the mechanical and biologic factors in the oral environment is a critical determinant in the development of unfavorable loading conditions that may result in an undesirable bone response and predictable bone loss[1].

Off axis force which is common during normal mastication would appear to induce more stress than does axial force. In case where esthetics require tooth overlap in the anterior region, off axis loading implant is usually unavoidable. The bone quality in the premaxillary region is also not as good as that in the mandible[1].Photoelasticity provided good qualitative information on the overall location and concentration of stresses but produced limited quantitative information. The strain gauge measurement provided accurate data regarding strains only at the location of the gauge. The finite element method is capable of providing a detailed quantitative data at any location within a mathematical model[8, 9].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 element, its mechanical behavior 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].A load applied to a dental implant may induce deformation of both the implant and surrounding tissues. Biologic tissues may be able to interpret deformation and respond with the initiation of remodeling process[1, 9]. A relationship is needed between the force that is applied on the implant and surrounding tissues and the subsequent deformation experienced throughout the system. The closer the modulus of elasticity of the implant resembles that of biologic tissues, the less the likelihood of relative motion at the tissue to implant interface. The cortical bone is at least 5 times more flexible than titanium. As the stress magnitude increases, the relative stiffness difference between the bone and titanium increases. The viscoelastic bone can stay in contact with the more rigid titanium more predictably when the stress is low. Once a particular implant system is selected, the only way for an operator to control the strain is to control the applied stress or change the density of bone around the implant. Such stress may be influenced by the implant design, size, implant number, implant angulation, and the restoration. Generally, the greater the magnitude of force applied to a dental implant system, the greater the difference in strain between the implant material and bone. In such cases, the implant is less likely to stay attached to the bone, and the probability of fibrous tissue ingrowth becomes greater[12].From a review of the literature it would appear that most finite element analyses have assumed that occlusal load was directly applied on the abutment of the dental implant. Such studies fail to consider the effect of a prosthetic crown in a clinical setting. The application of load on a crown or implant results in the production of different bending moments; therefore, a more detailed premaxillary finite element analysis model with an implant and its superstructure is necessary. This study is designed to evaluate the influence of stress/strain distribution in bone around an anterior maxillary implant in high density and low density bone using a finite element modeling and analysis.The aim of this in vitro study is to analyze the influences of stress and strain in bone around an anterior maxillary implant in high density and low density bone using a finite element modeling and analysis.

The objectives of the study are:

- To record and evaluate the distribution and values of stress and strain generated within the cortical and cancellous bone around an anterior maxillary implant under loading angle of 0, 30, and 60 degree to the long axis of the implant on applying a load of 178 N in high density bone.
- To record and evaluate the distribution and values of stress and strain generated within the cortical and cancellous bone around an anterior maxillary implant under loading angle of 0, 30, and 60 degree to the long axis of the implant on applying a load of 178 N in low density bone.

Material and methods

Model geometry

A model of a maxillary segment in the incisal region featuring an implant and its superstructure was constructed using ANSYS Pre-processor (ANSYS 11). A CT scan was used as a reference to model the geometry of anterior maxillary region[1, 5, 8] . A simulated 13 mm × 4.2 mm tapered threaded implant made of titanium alloy (Ti-6Al-4V) was used for this study. The implant was opposed by cortical bone in the crestal region and by cancellous bone for the remainder of the implant bone interface. Overall dimension of crown is 10 mm in height, 6.7 mm in buccolingual width and 9 mm in mesiodistal length. Crown will be attached to a 6mm high implant abutment featuring a 1 mm collar and 5 mm profile (fig. 10, 11, 12). The abutment was made of same alloy as implant and crown was made of porcelain.Since the primary goal of this study was not to evaluate stress/strain distribution at the implant abutment or the abutment prosthesis interface, the implant abutment crown complex was modeled as one piece structure. The implant abutment complex was placed in the middle of the anterior maxilla. The platform of the implant was modeled as being flush with the alveolar ridge surface to mimic effectively a real clinical situation[13] (Fig.1).

Specifying material properties

For the accurate analysis of the problem and interpretation of the results, two material properties were utilized i.e.

Young's modulus (elastic modulus) and Poisson's ratio. The cortical bone, cancellous bone and implant with abutment and crown were presumed to be linearly elastic, homogenous and isotropic. Although cortical bone has anisotropic material characteristics and possesses regional stiffness variation, they were modeled isotropically due to the unavailability of sufficient data and difficulty in establishing the principal axis of anisotropy. 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 literature survey[1, 5, 10]

Material	Elastic modulus		Poisson's ratio
	High Density(GPa)	Low density(GPa)	
Cortical bone	13.4	13.4	0.30
Cancellous bone	1.37	0.8	0.31
Titanium alloy	102	102	0.33

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

Imposing boundary conditions

Before applying the boundary conditions, the system of equations is not completely defined. This is because, any model which is generated, has to be constrained depending upon the requirements of the study. Thus, boundary conditions are applied to have enough fixed nodal displacements to prevent the structure from moving in space as a rigid body when external loads are applied (Fig. 2,3,4).

Interface condition

The bone implant interface was assumed to be perfect, simulating complete osseointegration. The implant, abutment and crown were assumed to be connected as a single unit.

Load application

This is a part of the procedure where an attempt was made to simulate actual clinical situation the implant was assumed to be subjected to 3 different loading angles independently i.e 0, 30 and 60 degree relative to long axis of the implant. To ensure that the axial force were directed along long axis of the implant load was directly applied on the occlusal node of the implant at the centre of abutment (Fig. 2). For off axis loading, an occlusal load was applied on a node at palatal side of crown (Fig 3,4). A 2 mm overbite was simulated to mimic clinical conditions[1]. The magnitude of the force used was 178 N which is also within the range of mean values reported in the literature[1, 5, 8]. 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.

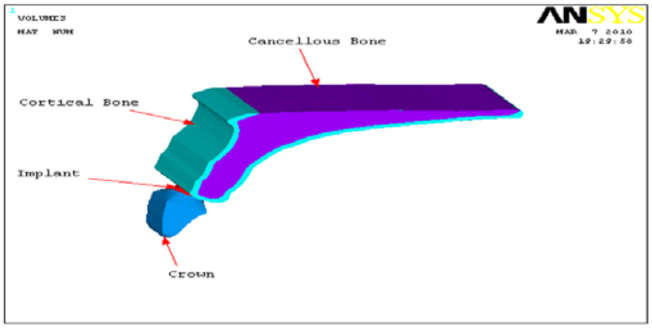

Figure 1: Three dimensional meshed structure of cortical bone.

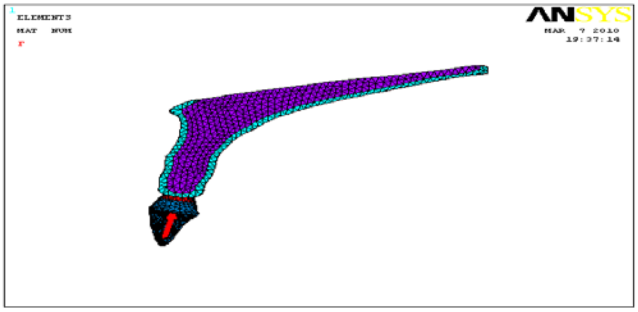

Fig 2: Boundary conditions of the problem and applied force.(zero degree)

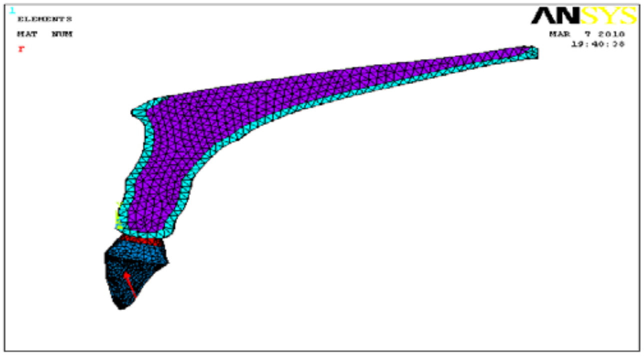

Fig 3: Boundary conditions of the problem and applied force.(30^0 degrees)

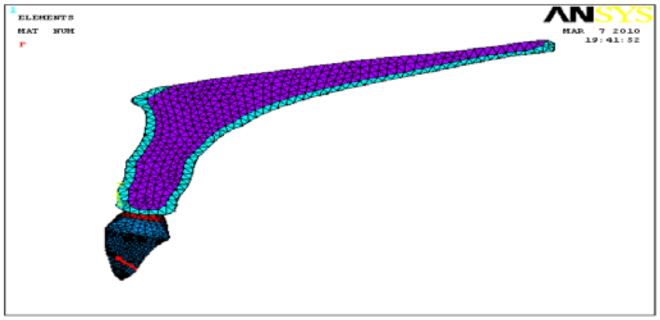

Fig 4: Boundary conditions of the problem and applied force.(60^0degrees)

Results

The study was conducted to analyze the distribution and values of stress and strain generated in the bone around an implant placed in anterior maxilla under loading angle of 0, 30, and 60 degree to the long axis of the implant in high and low density bone. Stress and strain were calculated using Von Mises criteria

Cortical Bone:

- Compared with cancellous bone substantial stress was observed in cortical bone .
- Under zero degree off axis loading stress was seen to develop around implant apex for both high and low density bone.
- Under 30 degree off axis loading stress was seen to develop not only around the implant neck but also near the implant apex in both high and density low bone.
- Under 60 degree off axis loading stress was seen to develop around implant neck for both high and low density bone.
- For the high density bone model, the maximum Von Mises stress were 8.11 Mpa, 26.9 Mpa and 48.12 Mpa at angles of 0, 30 and 60 degree respectively.(Table-2)
- For the low density bone model, the maximum Von Mises stress were 9.252 Mpa, 29.188 Mpa and 49.8 Mpa at angles of 0, 30 and 60 degree respectively.(Table-3)
- The stress develop under off axis loading conditions was significantly greater than that produce under axial loading
- For each 30 degree increase in loading (from zero degree), the maximum von mises stress develop in cortical bone increases on average 3 to 4 times compared with axial loading.
- For high density bone model the maximum Von mises strain were 707 µ strain, 2212 µ strain and 3724 µ strain at angles of 0, 30 and 60 degree respectively.(Table-4)
- For low density bone model the maximum Von mises strain were 799 µ strain, 2420 µ strain and 3919 µ strain at angles of 0, 30, and 60 degree respectively.(table-5)
- Regardless of load direction, the maximum von mises strain for low density bone was always greater than corresponding figure for high density bone.
- For all loading condition apart, from axial loading, the maximum von mises strain was concentrated in the cortical bone and was observed on the labial side of implant neck.
- For axial loading maximum strain was located on labial side near implant apex.

Cancellous Bone:

- Under the same loading angles the maximum von mises strain in low density was higher than that in high density bone.(table-5)
- Von mises strain increases as the loading angle increases regardless of the bone quality.(Table- 4, 5)
- For high density bone the maximum von mises stress in cancellous bone were 4.814 Mpa, 10.483 Mpa and 13.719 Mpa at angles of 0, 30 and 60 degree respectively.(Table-2)
- For low density bone the maximum von mises stress in cancellous bone were 4.787 Mpa, 10.118 Mpa and 13.136 Mpa at angles of 0, 30 and 60 degree respectively.(Table-3)
- For high density bone the maximum von mises strain in cancellous bone were 2514 µ strain , 7522 µ strain , and 10083 µ strain at angles of 0 , 30 and 60 degree respectively.(Table-4)
- For low density bone the maximum von mises strain in cancellous bone were 5983 µ strain, 12547 µ strain and 15420 µ strain at angles of 0, 30 and 60 degree respectively.(Table-5)

Loading angle	Low density bone	
	Cortical bone	Cancellous bone
Zero degree	799	5983
30 degree	2420	12547
60 degree	3919	15420

- The maximum von mises stress and strain was observed near the implant apex in both high density and low density bone.

Table – 2Von Mises stresses in cortical and cancellous bone in high density bone

	High density bone	
Loading angle	Cortical bone	Cancellous bone
Zero degree	8.11	4.814
30 degree	29.906	10.438
60 degree	48.412	13.719

*** All values in MPa (Mega Pascal)**

Table - 3

Von Mises stresses in cortical and cancellous bone in low density bone

Loading angle	Low density bone	
	Cortical bone	Cancellous bone
Zero degree	9.252	4.787
30 degree	29.188	10.118
60 degree	49.8	13.136

*** All values in MPa (Mega Pascal)**

Table -4

Von Mises strain in cortical and cancellous bone in high density bone

Loading angle	High density bone	
	Cortical bone	Cancellous bone
Zero degree	707	2514
30 degree	2212	7552
60 degree	3724	10083

*** All values in micro strain**

Table – 5

Von Mises strain in cortical and cancellous bone in low density bone

*** All values in micro strain**

Discussion

The success rate for dental implants suggests that tissues are capable of sustaining a long term positive response to implant loading. This implies that bony architectural strength and the direction in which stresses are transferred to the surrounding bone are typically favorable as regards to bone survival and implant stability[1]. Lower survival rates were observed for implants placed in anterior maxilla. Long term clinical results studies have reported 95% survival for mandibular implants and 65-85% survival for maxillary implants[3-5].

When the teeth are lost in the anterior maxilla, the pattern of bone loss cannot be accurately predicted. This change in bone morphology often dictates placement of implants[5,14].

In some case however a clinician, may need to introduce implant supported prosthesis in areas of compromised bone morphology, which may result in the development of an unfavorable Off axis loading. An off axis force could induce a bending moment thus exerting stress gradients within the implant as well as the adjacent bone. The bone quality in the anterior maxillary region is not good (type 3), therefore it was important to investigate how these off axis force could affect the stress distribution in bone of different quality[1].

Different methods have been used to study the stress/strains in the bone and dental implants[8-10]. For example,

a) Photoelasticity, which provides good qualitative information pertaining to the overall location of stresses but only limited quantitative information.

b) Strain gauge measurements, which provide accurate data regarding strains only at specific location of the gauge.

c) FEA, which is capable of providing detailed quantitative data at any location within a mathematical model. It would therefore appear that FEA could be a complementary tool for exploring the detailed mechanical response at work in implant dentistry. Assumptions imposed on the FEA models (model geometry, load magnitude, load direction and material property) influence the relative accuracy of the FEA. In the present study, an occlusal force of 178 N was applied, which is within the range of mean values reported in the literature[1, 5, 8, 15]. This force was applied on implant supported prosthesis to simulate actual loading condition. Different investigators have reported that the maximum incisal bite force ranges from 50 to 370 N. The variation may be related to many factors such as muscle size, bone shape, age and sex, degree of edentulism, parafunction and type of food[16]. However, the application of functional forces induces stresses and strains within the implant prosthesis complex and affects the bone remodeling process around the implant. Excessive forces on implant supported prosthesis could impair osseointegration or induce bone resorption[1, 5, 8, 17]. Therefore, when evaluating the stresses and strains in the bone, it is essential to consider their source, the occlusal force.

Designing models that simulate clinical situations is also essential[5]. For the present study, a careful review of cephalometric norms was done to create a 3-D model of anterior maxilla using a CT scan. Saab et al[5] and Clelland et al[8] also created finite element model of anterior maxilla using CT scan. In the present study the cortical bone for the maxilla was modeled as a 1 mm layer[5, 8], which represents actual clinical situation.The palatal surfaces of the maxillary anterior teeth provide a vertical ramp for the mandibular anterior teeth to guide the mandible through protrusive and lateral excursions[5]. An Off axis which is common during normal mastication would appear to induce more stress than does axial force[18]. For incisal region, the direction of maximum incisal biting force is about 12 -15 degree towards the frontal plane, which means that lateral component of force on an anterior dental implant can be appreciable[1].Moreover, the placement of dental implants would be more likely to produce an unfavorable off axis load in the case of severe palatal resorption of the alveolar ridge following tooth extraction than in the case of a ridge without resorption[1].

In comparative analysis, the complexity of real life situations can be simplified, assuming that proportion and relative effect accurately related. Hsu et al varied the direction of the force to create an unfavorable loading situation in the anterior maxilla. In reality, unfavorable loading situations are more due to increased bone resorption after tooth loss. Since the reconstruction of multiple complicated bone models was very elaborate and difficult, certain assumptions needed to be made to simulate unfavorable loading angle of an implant. Thus, the direction of the load was changed instead of bone geometry[1]. However, a more valid assumption for the precise modeling of the bone implant system is needed for further study.

Based on the result, the maximum equivalent stress/strain elicited by a force on this implant model appeared to increase linearly with an increase in the angle of loading from 0 to 60 degrees. For each 30 degree increase in loading angle, the maximum equivalent stress developed within the cortical bone increased an average of 3 to 4 times compared with that of an imparted axial load. The maximum stress observed for 60 degree loading angle for high density bone was 48.412 MPa in the cortical bone and 13.719 MPa in the cancellous bone. For low density bone maximum stress for 60 degree loading was 49.8 MPa for cortical bone and 13.136 MPa for cancellous bone. Such a result would seem to indicate that load direction (upon implant) exerts great influence upon the distribution of stresses within the supporting cortical bone and higher for low density bone. The ultimate strength of human cortical bone ranges from 72 to 76 MPa in tension and 140 to 170 MPa in compression and for cancellous bone it ranges from 22 to 28 MPa[4, 12].Equivalent Von mises strain distribution in cancellous bone differed among the six different testing condition, although in each case the maximum Von mises strain was observed near the implant apex. Under the same loading angle, the equivalent von mises strain around the implant apex was higher in the low density model than in the high density model.

From the physiologic viewpoint, bone density is directly related to the strength and elastic modulus of bone[19]; thus, these results appear reasonable. The patterns of strain distribution within the bone were influenced not only by the load direction but also by bone quality. Furthermore, the maximum equivalent strain was observed near the implant apex and not found at the cervical area of the implant. If a higher density of the bone had been assumed in the study, the location of the maximum equivalent von mises strain distributed area would have changed, as it did in a previous study (Tada et al)[20].

Based upon the mechanostat concept, peak load magnitudes creating strains greater than 4000 µstrain would typically result in pathologic overload.For high density bone model investigated the maximum strain in the cortical bone was 3724µ stain and for low density bone the maximum strain in the cortical bone was 3919 µstrain, which is below the pathologic overload zone. For cancellous bone maximum strain was above the pathologic overload zone for 30 and 60 degree loading angle and was much higher in low density bone compared to high density bone. Pathologic overload in bone may result in marginal bone loss and/or implant failure; hence when off axis loading is unavoidable, specific case by case selection of an implant location of appropriated bone quality is critically important.

The stress distribution revealed was also consistent with the result of an FEA by Tada et al. for low density bone models utilizing a cylindrical implant, the maximum strain developed upon implant loading was observed around the implant apex[20]. The result of the present study showed that the force direction and bone quality were related to the stress/strain elicited along the implant bone interface and the implant abutment interface. Off axis loading and poorer bone quality produced much more stress/ strain than axial loading and better bone quality. Despite little evidence that overloading can cause loss of osseointegration or bone resorption, some problems in clinical cases were solved by equilibration to achieve optimal occlusion and to avoid contact in lateral and protrusive movement. Where possible, limiting the biomechanical effect of the provisional restoration by

a. Limiting occlusal contact in centric occlusion.
b. Removing all excursive contacts
c. Limiting the effect of cantilever and off axis laoding
d. Splinting implants together has been suggested[21].

In the present study, the bone implant inference was assumed to be completely osseointegrated and the bone was modeled as homogeneous and isotropic. But varying degree of osseointegration occurs clinically and bone is actually anisotropic[22]. Higher stress can be observed with decrease in the percentage of osseointegration because of poor bone quality in anterior maxilla. The stress/strain patterns differ if there is no complete osseointegration between the bone and Implant[23].

Van Oosterwyck[18] in his finite element analysis concluded that bone-implant interface, elastic properties of bone, degree of osseointegration and the presence of lamina dura affect the final outcome of the study. The results also suggested that bone stress patterns are highly sensitive to the considered characteristics. This stresses the importance of case dependent finite element models of human jaw.

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:

- Firstly no movement was allowed between the implant and the bone during loading from different directions which may not represent a real clinical situation.
- The implant was also assumed to be 100% osseointegrated, which is never found in clinical situation. This would alter forces transmitted to the supporting structures.
- Next, the cortical bone, cancellous bone and the implant were considered to be isotropic and homogeneous. The bone in reality is anisotropic and inhomogeneous The static loads that were applied differed from the dynamic loading encountered during function

Finite Element Analysis has been used extensively in the prediction of biomechanical performance of dental implant systems. Assumptions made in the use of Finite Element Method in implant dentistry should be more accurate. 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, modeling 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 that considered the bone homogenous and isotropic, the results of static load and linear analysis support the following conclusion:

- Under Off axis loading Maximum amount of stress concentration was observed in the cortical bone located buccally around the implant neck .The cortical bone plays a major role in the dissipation of stresses.

- Maximum amount of strain and strain concentration was observed in the cancellous bone near the implant apex .Higher strain was generated in the cancellous bone because of the lower modulus of elasticity and more in low density compared to high density bone
- The maximum von mises stress/strain imparted to bone increases linearly with an increase in the angle of off axis loading.
- The highest stress and strain were generated with 60 degree loading angle and more in low density bone.
- The highest stress was below the elastic limit of the bone (approximately 60 Mpa) and the highest strain was within the physiologic limit as described by Frost.(in cortical bone)
- The zero degree angle loading produced the least amount of stress and strain, so as far as possible the implants should be placed along the axial loading direction of the proposed prosthesis.
- Stress/ Strain generated in high density bone were lower than low density bone thus high density bone might ensure a better biomechanical environment for implants.

Further studies are required to evaluate whether differences in bone quality resulting from differences in strain distribution may affect different mechanisms of failure. 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.

Reference

1. 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.
2. 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: 50-55
3. 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.
4. 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.
5. 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.
6. 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.
7. 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.

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. Norton M.R and Gamble C. Bone classification: an objective scale of bone density using the computerized tomography scan. Clin Oral Imp Res. 2000; 12: 79-84.
14. 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.
15. Kao H.C, Gung Y.W, Chung T.F, and Hsu M.L. The influence of abutment angulation on micromotion level for immediately loaded dental implants: A 3-D finite element analysis. Int J Oral Maxillofac Implants 2008; 23: 623-630.
16. Waltimo A., Konomen M. Bite force on single as opposed to all maxillary front teeth. Scand J Dent Res 1994; 102: 372-375.
17. Oosterwyck V.H, Duyck J, Sloten J.V, Perre G, Cooman M and Lievens S et al. The influence of bone mechanical properties and implant fixation upon bone loading around oral implants. Clin Oral Imp Res 1998; 9: 407-418
18. David C.Holmes, Chris R.Haganman, Steven A. Aquilino. Deflection of superstructure and Stress Concentrations in the IMZ Implant system. Int J Prosthodont 1994; 7 : 239-246
19. Carl E.Misch, Zbimin and Martha W.Bidez. Mechanical Properties of Trabecular Bone in the Human Mandible: Implication for Dental implant Treatment Planning and Surgical Placement. J Oral Maxillofac Surg 1999; 57: 700-706
20. Shinichiro Tada, Roxana stegaroiu, Eriko Kitamura, Osamu Miyakawa, Haruku Kusakari. Influence of Implant Design and Bone Quality on Stress/Strain Distribution in Bone Around Implants: A 3-dimensional Finite Element Analysis. Int J Oral Maxillofacial Implants 2003; 18: 357-368
21. Dean Morton, Robert Jaffin, Hans-Peter Weber, Dr. Med Dent. Immediate restoration and loading of dental implants: Clinical considerations and Protocols. Int J Oral Maxillofac Implants 2004; 19 (suppl):103-108
22. Aisling M. O’ Mahony, John L. Williams, and Paulette Spencer. Anisotropic elasticity of cortical and cancellous bone in the posterior mandible increases periimplant stress and strain under oblique loading. Clin Oral Imp Res.2001; 12: 648-657.
23. 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