Original Research Paper

Prosthodontics

FEM ANALYSIS OF STRESS DISTRIBUTION AROUND A DENTAL IMPLANT AND SURROUNDING JAW BONE.

Dr Bennett Atlin Correya*	Associate professor (CAP), Government Dental college Calicut *Corresponding Author
Dr Mehul R Mahesh	Associate Professor(CAP) , Government Dental College, Calicut
Dr Joy P T	Asst. Professor Government Dental College, Calicut
Dr.Iven Jose	Associate Dean, Christ university faculty of engineering, Kumbalagodu, Karnataka
Dr Dheeraj Kumar Koli	Asst professor in Prosthodontics, AIIMS, New Delhi

ABSTRACT

A new era of oral rehabilitation began with the introduction of Osseointegrated dental implants. This study analyses the stress around an implant and the jaw bone by applying a known load along the axial direction and non axial direction. Root form implants of 4.5 mm x 15 mm with screw retained abutment was created by ANSYS and Pro engineer software. A pressure of 200 Psi which is equivalent to biting force was applied on the occlusal surface of the Crown. An additional pressure of 50 Psi was applied on the crown in horizontal direction.

The distribution indicates that most of the stress is concentrated around the area surrounding the cortical bone near the implant abutment interface.

KEYWORDS :

Introduction

Introduction of osseointegrated dental implants opened a new era for oral rehabilitation . High success rates of 70- 85% for implants placed in the maxilla and over 90% success rate for the mandible, have been reported.[1]-[4]

Even though the success rates have been high, implant failures do occur. Excessive masticatory loads have been implicated in many implant failures.[5] Clinical studies have demonstrated that stress on implant-supported prostheses was comparable or lower than that for natural teeth.[6] Normal occlusal loads range from 15 to 50 N.[7]-[8]

The mechanisms by which tissue loading induces cellular control have not been identified.[9] Bone is "genetically programmed" to accept a particular amount and pattern of stress as normal and that deviations lead to bone resorption[10].

Mechanical instability may be a key factor in implant failure. Micromovements lead to bone resorption and to the development of fibrous connective tissue at the bone-implant interface[11]. Because of the flexibility of the mandible,[12] micromovement around the dental implants probably will always be encountered.

Bone is usually subjected to cyclic loads, with results that differ from static loads.[13] If a sufficient number of repetitive load cycles are applied, stress microfractures in bone may occur. After bone microfractures occur, microdamage caused by stress greater than normal levels may stimulate osteoclastic activity .[14]

If masticatory forces on implants can produce stresses at the bone-implant interface greater than the elastic limit of bone, fractures may occur. Although theoretical analyses of the stress distribution around implants have been made, the stress analysis studies (photoelastic and/or finite element analyses [FEA]) have focused primarily on the implant material itself.[14]-[16]

In this study, nonlinear stress analysis using the finite element method was performed to investigate the stress occurring in the jaw bone around an implant on cyclic loading.

MATERIALS AND METHODS

FEM model of the complete implant – jaw bone system was done, it was cut to half, and half symmetry conditions were applied for better display of results using Pro- Engineer software, SOLID-92. ANSYS software was used for the FEM analysis

MODELING OF THE IMPLANT

One root-form implant, 4.5 mm in diameter and 15 mm in length, with a screw-retained abutment was created using ANSYS and Pro Engineer software .

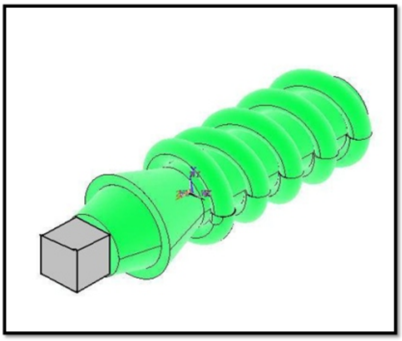


Fig 1: Implant modeled in Ansys v.10

Using ANSYS menu the implant was made with a cylinder of length 9 units, and radius 1.5 units . A toroidal volume of radius 1.5 units, smaller cross-sectional radius of 0.25 units and larger cross-sectional radius of 0.5 units was made. Using 'WP (working plane) rotation by increments', the working plane was rotated by an amount equal to the thread angle of the implant, i.e. 26°. Five turns of the threads were made. For the abutment, two frustums with smaller radii of 1.5 units, larger radii of 2 units and 2.25 units and length of 2 units and 3 units respectively were made. The dental crown was approximated by a cube of side 2 units, was created in order to reduce the complexity (and increase convergence rate) of the solution.

MODELING OF THE JAW BONE

The jaw bone consists of hard cortical bone and soft cancellous bone. The cortical bone (represented by a cuboid) has dimensions of 15×12×4 units, thus indicating that the thickness of this bone layer is taken to be 4mm. The cancellous bone had the dimensions

15×12×20 units.

These two volumes were glued, indicating that they are attached to each other.

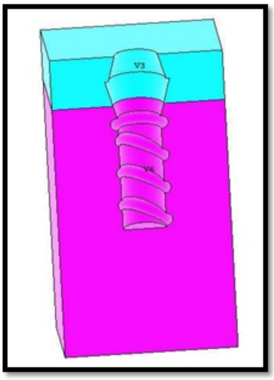


Fig 2: Two layered jaw bone, post drilling and implant mounting

MATERIALS AND PROPERTIES

Analysis was performed using a finite element method program (ANSYS Inc., Canonsburg, PA, USA). 4 types of materials used were

1) The titanium alloy implant
2) Titanium alloy abutment,
3) The cortical bone,
4) The cancellous bone.

Since the elastic modulus and Poisson's ratio of pure titanium is almost the same as that of titanium alloy[10], a model with unified abutment and implant areas (one-piece type) was produced for the purpose of performing analysis. The titanium implant and tooth crown were taken to be linear -› elastic -› isotropic.

Material	**Young's Modulus**	**Poisson Ratio**
Titanium alloy	113 GPa	0.3

As for the double bone layer, orthotropic properties were assigned. The assumed material characteristics of the jaw bones are linear, elastic and orthotropic.

Material Property	**Cortical Bone**	**Cancellous Bone**
E_x	13.9 GPa	4.5 GPa
E_y	14.6 GPa	5.8 GPa
E_z	15.3 GPa	4.9 GPa
v_{xy}	0.27	0.32
v_{yz}	0.33	0.34
v_{xz}	0.29	0.31
G_{xy}	1.6 GPa	0.8 GPa
G_{yz}	1.3 GPa	0.7 GPa
G_{zx}	1.7 GPa	1.3 GPa

E – young modulus , v – poison's ratio , G – shear modulus

ELEMENT TYPE

In this study, a finite element software (ANSYS) is used. In particular interest to this project is the element SOLID92 (Figure 3). It is a three-dimensional 10-node tetrahedral structural solid which has a quadratic displacement behavior and is well suited in modeling irregular meshes. The element is defined having three degrees of freedom at each nodes in x, y, and z directions.

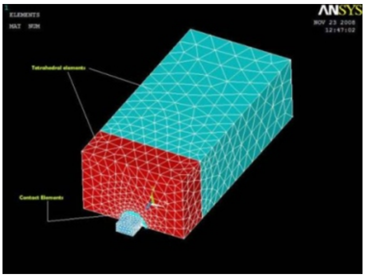


Fig 3: Tetrahedral and contact elements, after the model is meshed

The mesh density was kept more at the interface of the implant and the jaw bone, since it was the area of concern.

LOADING CONDITIONS

The major force acting on the implant-system is the non axial biting force. Its value ranges from 8N to 850N. This force has been replaced with an equivalent pressure. A pressure of 200 Psi (or around 1.4 MPa) is applied on the top area of the tooth crown. An additional pressure of 50 Psi was applied on one of the vertical faces of the crown (cuboid) in the horizontal direction.

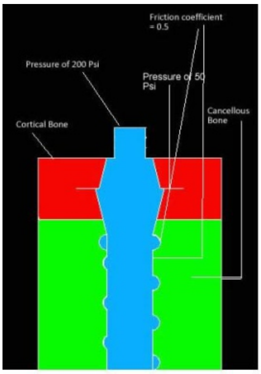


Fig 4: Loading conditions on various parts of the implant – bone system

For the fatigue calculations, it is necessary to define the character of load changeability in the shape of a curve load-time, the so called load signal. In an applied low-cycled scheme of 24-hour loads, the average values were 60 N.

A total of 6 load steps were taken into account. The first, third and fifth involved application of pressure on the tooth crown, while the second, fourth, and sixth did not involve any structural loads on the crown/ implant system.

BOUNDARY CONDITIONS

In all the stages of implant analysis, all degrees of freedom at the bottom, and side parts of jaw bone were fixed. This assumption seemed to have its explanation in dental practice, where no movements of implants under physiological load are acceptable.

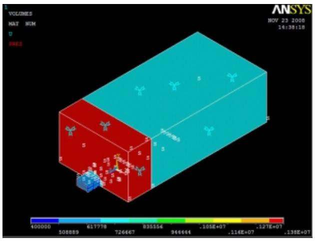


Fig 5: Implant – Jaw bone system, with all loads and boundary conditions applied

Von MISES STRESS

Von Mises stress is a scalar quantity that includes all the components of the stress tensor that represents the overall magnitude of the tensor in three-dimensional. It gives an impression of the overall magnitude of the stress tensors and allows a comprehensive comparison between the different components.

RESULTS

A total of 6 load steps were taken into account. The first, third and fifth involved application of pressure on the implant abutment, while the second, fourth, and sixth did not involve any structural loads on the implant abutment. The application of 200 Psi pressure on the horizontal face, and 50 Psi on the vertical face, gave a maximum *VON MISES STRESS* of 153.9 Mpa.

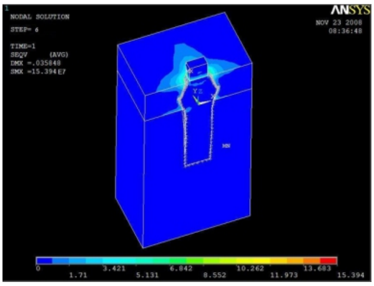
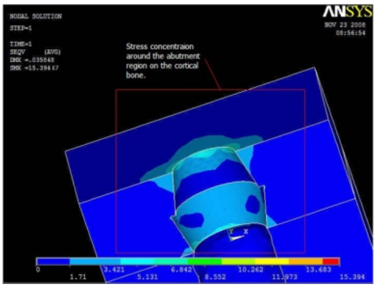


Fig:6 von Mises stress distribution in the implant – bone assembly

The distribution in fig: 6 indicates that most of the stress is concentrated around the area surrounding the cortical bone near the implant abutment interface. This bone being harder does not easily get damaged because of the resulting stresses. The level of equivalent stresses for cortex tissue in the area of implant entrance into the bone is dangerously high but values are located below the average cortex tissue strength.



Stress distribution

Determining the real level of hazardous stresses and scopes of their occurring depends significantly on the mesh density. The problem of the influence of division results on finite elements is commonly known; however the basic rule is to carry out comparative models researches always with identical mesh parameters. Nevertheless, the seeking for real stresses values requires a detailed examination of the mesh density influence. In the presented analysis, increasing mesh density results in an increased stress value at nodes adjacent to the edge.

DISCUSSION

An oral implant is a biologic or alloplastic material surgically inserted in to hard or soft tissues of mouth for functional or cosmetic purposes.

Osseointegration of the titanium implant in bone is considered an essential criterion for a successful dental implant. However, compared with natural tooth, a dental implant embedded in bone sometimes may be subjected to unacceptably high impact from occlusal forces. A dental implant and the surrounding bone are exposed to different stresses under occlusal forces due to lack of a periodontal ligaments around the dental implant. Excessive masticatory loads have been implicated in many implant failures. Bone is a relatively brittle material ,which is regularly subjected to cyclic loads of mastication Duyck et al reported that excessive dynamic loads caused crater like defects lateral to osseointegrated implants. Therefore, stress analysis under impact loading may be helpful for predicting implant stability in oral functional conditions.[17]

The purpose of this study was to simulate the biomechanical behavior, and to evaluate the resulting stress distribution on the jaw-bone, thus enabling the appropriate selection of, material and geometric characteristics of a dental implant part which creates a criteria to control the bone tissue loading. A basic tool that is commonly used for evaluation of bone loading state is the non-linear FEM analysis. Nonlinear stress analysis using the finite element method was performed here , to investigate the stress occurring around implants caused by biting forces.

Calculation of this stresses allow the investigator to determine the high stress areas or areas of deformations which may lead to fracture. The finite element (FE) method provides mechanical responses and alters parameters in a more controllable manner, driving its common use as an analytical tool in dental biomechanical studies.

In this study, nonlinear stress analysis using the finite element method was performed to investigate the stress occurring in implants caused by biting forces. In the present study, the distributions of the von Mises stress and the major principal strain in the bone surrounding the implant were investigated. The von Mises stress is a scalar variable that is defined in terms of all the individual stress components and, therefore, is a good representative of the state of stresses.

Six loading steps, were performed during the FEM study .A maximum von Mises stress of 153.9 MPa was calculated, which is well within the physiologic limits, which the bone tissue can with stand. The stress distribution indicates that most of the stress is concentrated around the area of the cortical bone, surrounding the implant-abutment interface.

In FE analysis studies, the assumptions made regarding the geometry, mechanical properties of the materials, and loads and constraints applied to the model have a key role in the accuracy of the experiment. Studies on FE analysis of dental implants suggests the importance of modeling bone as an anisotropic material. **O'Mahony et al**[18] compared a completely isotropic model of the mandible with a transversely isotropic model and found a 20% higher level of stress at the crestal level for the transversely isotropic model. **Clelland et al**[19] created a 3-dimensional (3-D) model of the anterior maxilla with a 1.5- and 3.0-mm-thick cortical layer with isotropic characteristics, which does not represent type 3 bone with a thin cortical layer. In the present study cortical bone of 4mm thickness was modeled to duplicate the clinical situation.

Cortical bone would absorb most of the stresses, while the reaction forces of the cancellous bone upon the loaded implant would be underestimated. By assigning to the cortical and cancellous bones properties that are not so different, it can be seen that the highest strains observed on the bone are at the coronal third of the implant-bone interface. However, they are not limited to the cortical layer; they are also shared with the cancellous core. This may motivate future investigators to be more thorough in the research of the mechanical properties assigned to the materials involved in FE analysis experiments.[20]

LIMITATIONS OF THE STUDY

- In biological tissue, a hard and fast rule regarding the homogenous nature cannot be followed. for the study purpose we were considering cortical and cancellous bone as tissues which are isotropic and homogenous all through their structure.
- Only simplified bony segments were modeled for parameter studies; the bending of the mandible during mastication has not been taken into account. Therefore, the FEA modeling results provide only a general insight into the biomechanical aspects of the system under average conditions.
- The ideal clinical conditions in an oral cavity and the diversity of forces to which an implant has been subjected has not been simulated.
- The fixed values of stress concentrations at the cortical bone have not been computed. Instead, the stress distributions have been mentioned as a range.
- The implant bone interface has been considered to be 100% osseointegrated and fixed with out any micro movements.

Research to explain the anisotropic behavior of the maxilla and mandible and to accurately calculate the material properties for cortical and cancellous bone is needed, and then conclusions drawn from future FE analysis studies may be more clinically relevant

CONCLUSION

Within the limitation of the study following conclusion were made that on applying axial and non axial load on the implant and abutment the stresses were found concentrated mainly in the cortical bone around the implant-abutment interface.

References

1. Branemark Pi, Zarbga, Albrektson T . Tissue-integrated prostheses. In: Osseointegration in clinical dentistry . (Chicago:Quintessence Pub Co, 1985:11,53,93-4).
2. Adell R, Lekholm U, Rockier B, Branemark Pi. A 15-year study of osseointegrated implants in the treatment of edentulous jaw. (Int J Oral Surg 1981;10:387-416).
3. Albrektson T, Bahl E, Enbon L, Et Al. Osseointegrated oral implants. A multicenter study of 8139 consequetively inserted nobel pharma implants. (J Periodontol 1988;59:287-96)
4. Zarb Ga, Schmitt A. The longitudinal clinical effectiveness of osseointegrated dental implants:the Toronto study. Part I: surgical results. (J Prosthet Dent 1990;63:451-7).
5. Ibbott Cg. In vivo fracture of a basket-type osseointegrating dental implant: a case report. (Int J Oral Maxillofac Implants 1989;4:255-6).
6. Falk H, Laurell L, Lundgren D. Occlusal force pattern in dentitions with mandibular implant-supported fixed cantilever prostheses occluded with complete dentures. (Int J Oral Maxillofac Implants 1989;4:55-62).
7. Anderson Dj. Measurement of stress in mastication I (J Dent Res 1956;35:664-73)
8. Anderson Dj. Measurement of Stress in Mastication II. (J Dent Res 1956;35:671-3)
9. Lanyon Le. Functional strain as a determinant for bone remodeling. (Calcif Tissue Int 1984;36(Suppl 1):556-61)
10. Rubin Ct, Lanyon Le. Osteoregulatory nature of mechanical stimuli: function as determinant for adaptive remodeling in bone. (J Orthop Res 1987;5:300-10).
11. Shreiber A, Jacob Ha. Loosening of the femoral component of the ICLH double cup hip prostheses. A biomechanical investigation with reference to clinical results. (Acta Orthop Scand 1984;27(SuppD:l-34.)
12. Korioth Tw, Hannam Ag. Deformation of the human mandible during simulated tooth clenching. (J Dent Res 1994;73:56-66)
13. Martin Ad, Mcculloch Rg. Bone dynamics:stress,strain and fracture (J Sports Sci 1987;5:155-63)
14. Smith El Raab Dm. Osteoporosis and physical activity. (Acta Med Scand 1986;711(suppl) :149-56).
15. Cook Sd, Klwittar Jj, Weinstein Am. The influence of implant elastic modulos on the stress distribution aroud LTI carbon and aluminium oxide dental implants. (J Biomed Mater Res 1981;15:879-87.)
16. Davis Dm, Rimrott R, Zarb Ga. Studies on frameworks for osseointegrated prostheses: part 2. The effect of adding acrylic resin or porcelain to form occlusal superstructure. (Int J Oral Maxillofac Implant 1988;3:275-80)
17. Tanimoto Y, Hayakawa T And Nemoto K, Mode Superposition Transient Dynamic analysis for Dental Implants with Stress absorbing Elements: A Finite Element Analysis (Dental Materials Journal 25 (3) :480-486,2006)
18. O'mahony Am Williams Jl, Spencer P. Anisotropic elasticity of cortical and cancellous bone in the posterior mandible increases peri-implant stress and strain under oblique loading (Clin Oral Implant Res 2001;12:648-57)
19. Clellant Nl, Lee Jk, Bimbenet Oc, Bratley Wa. A three-dimensional finite element analysis of angled abutments for an implant placed in the anterior maxilla. (J Prosthodont 1995;4:95-100)
20. Lucie Himmlova, Tatjama Dostalova, Alois Kacovsky And Svatava Konvickova, "Influence of implant length and diameter on stress distribution: A finite element analysis". (J Prosthet Dent 2004; 91:20-25)