Mathematical & Computational Applications, Vol. 3, No. 2, pp. 83-91, 1998

STRESS ANALYSIS AROUND DENTAL IMPLANT IN HUMAN MANDIBLE

Mehmet ZOR* and Sami AKSOY*
*Dokuz Eylül University, Department of Mechanical Engineering , Bornova, Izmir, Turkey

Abstract- The stresses around the hole of an implant located in the anterior of a human mandible have been analyzed by using the Finite Elements Method. Three different models, each of which consists of different regions of mandible have been examined. In addition, variations between the implant and apical and coronal sections in different bone tissue have been included in this study. Sixteen different variations have been obtained. As a result, it has been observed that implants with straight abutments provide safer results than ones with angled abutments. Since the strength values of spongy bone are much lower than those of cortical bone, the stresses can exceed the strength limits in the cases where coronal upper section consist of spongy bone or in the cases where the cortical bone in the upper surface is very thin. So implants with straight abutments should be preferred, particularly in these cases

1. INTRODUCTION

The resorption of mandibular ridges remains important in dentistry despite medical and technological advances increases and progresses at present. It is a problem of universal scope, the fact shown from which the trouble is common either in developed countries or non-developed ones. Today the studies that are convenient to be applied, by placing artificial materials as oral implant for dislocated teeth have a feature to be an alternative to compensate this unoccupied field.

The implants used in the case of tooth absence associated with natural or unnatural tooth loss have been studied, and produced in accordance with almost all anatomic and protetic cases. Producer firms provide all technical possibilities available for oral implants to be used more commonly and to place easily by the dentist. Big difficulty, however, begins after this, which means that it is required to reciprocate the forces either vertical or parallel to tooth axis and to transmit them to the mandible. The forces can cause collapse or sometimes piling up of the natural tissue. A tissue placed around tooth rests and called periodontal membrane warns the brain by reflection to prevent these actions on mandible. But great amount of this tissue is lost around an implant hole in the case where implant is applied to the mandible. Therefore, it is very important to determine the stresses that occur around the holes where dental implant is applied to the mandible. Geometric and biological properties of the mandible, types and materials of the implants are the most important factors having influence on the stresses.

In a study alternative materials for three endosseous implants have been examined [2]. But today the material most used in dental implant are titanium and its alloys. Stress absorbing elements used in dental implants in order to decrease the magnitude of stresses and obtaining uniform distribution of them have been examined [4]. The numbers of studies concerning cases three dimensional studies are less than ones concerning two dimensional ones. The stress distribution in an edentulous mandible provided with two implants in the interforaminal region has been examined by means of different finite element models [1]. A study in which a two dimensional finite element model of the mandible with two implants was used, demonstrated that the place of support of the model has a large influence on the results [7]. It is known that the cases of single tooth absence have induced some insufficiencies in terms of fonation, function, and esthetic, and studies at large scale are carried out on this subject. Recently, single tooth implants that can be used as his/her own tooth and do not affect the near by teeth are often utilized, and the studies on their applications are conducted. The results of the study used finite elements method to examine the distribution patterns of bone stresses and displacements around one commercially available implant and on experimental implant with a resilient layer material incorporated under its superstructure, showed that the new modification is a simple and efficient way to mimic the structural natural tooth unit. It also showed that it allowed movement of the superstructure without of the implant three times that of the nonresident model [6]. Eleven different post type endosseous implants were biomechanical analyzed by finite elements method to compile a list of features that could be used to design an optimal post-type endosseous implant [8]. Implant features causing high stresses and low stresses, possibly contributory to pathologic bone reposition and bone atrophy were nodded. This study reports a finite element survey of 11 different post type endosseous implants, to compile a list of features that could be used to design an optimal post-type endosseous implant. First low stresses can be as problematic as high stresses. Partial or complete bone resorption can occur under both circumstances. Second, larger implants do not make better implants. Whereas it is clear that small implants increase the stress transmitted to bone, it has been shown that larger and larger implants produce more and more benefits. In another study, stress magnitudes and contours in bone, surrounding six endosteal post-type dental implants were calculated by using finite elements method. [5].

In this study oral implants are evaluated from this aspect; the stresses produced by forces transmitted on the mandible are calculated for three different finite element models of mandible by using Sap90 and Ansys finite element programs, based on the height and border conditions subject to the cases of different type of prosthesis, and conditions during the assembling these prosthesis on the mandible; and the stresses subject to these cases are compared.

2. PROBLEM DEFINITION

The stresses that occur on teeth and consequently on mandible relate to periodontal membrane. Periodontal membrane is located around roots of teeth and it supplies force transformation between teeth and mandible or maxilla. It resists to the increase of the forces that can cause unwanted positions on mandible by warning to brain and reflex mechanism.

During making a hole where implant is located, periodontal membrane and tissues around root are completely or partly destroyed and functions of extinction of forces and warning brain can not be performed. Therefore, unwanted results caused by the extreme stresses around the hole in which implant is located can not be prevented. The control of deformations of bone tissue depends on knowing the stress distributions around this hole.

3. MATERIALS AND METHODS

In this study, a human mandible in front of which an implant is located has been examined and the stresses around the hole of the implant have been analyzed by using Finite element method. According to regions of mandible, different types and standards of implants are used. Also geometric form of the mandible and the teeth have important influence on choosing implant. Certain types of implants used in front region of mandible have been examined and their type given in fig.1.Depending on the length of the mandible in vertical axis, implants can remain in spongy bone or can pass this bone and lean against the cortical bone. This situation also depends on the length of the implant.

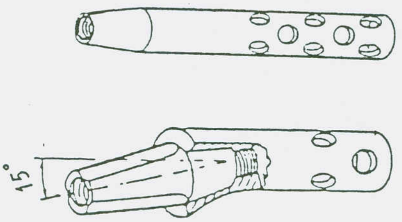


Figure 1. The types of the implants

Furthermore, some cases where the cortical bone in the upper region wears out and spongy bone in the region may be exposed cortical bone becomes thinner, which can be met in daily life. Therefore these cases are also included in this study. By this way the implant can contact to one of the two bones on the lower and upper parts. These connection forms are demonstrated in table 1.

Thus, implants have been grouped separately in modeling process as follows:

a- Their mouth-inside parts do not make angle with the axis of implant i.e. straight-headed.

b- Their mouth- inside parts make 15° with the axis of implant i.e. inclined-headed.

c- The same type implants have connections with lower and upper sections in different bone tissue.

d- Their inside sections contain spongy one for a certain height from lower section.

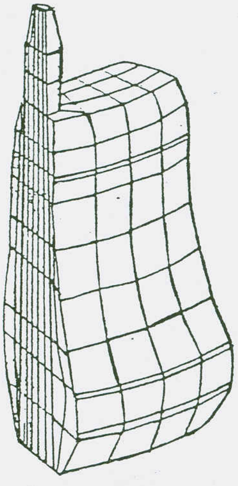
e- Their inside sections are filled.

Each of these cases is presented in table 1.

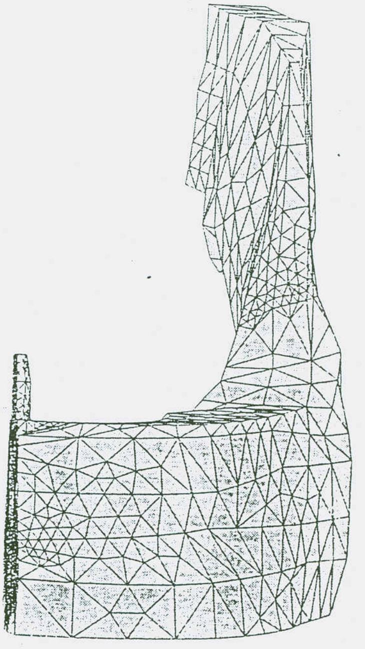
In this study the mandible belonging to an adult person has been modeled. Mandible's geometry changes from person to person. The geometry of some regions of the mandible are simplified in finite element model. The front area of the mandible has been primarily modeled, called a type model, by using SAP90 and stress analysis has been carried out for 16 different variations according to the length and type of prosthesis and also according to the case of mandible.

Because of examining only front area of the mandible, some errors may exist in the stress analysis. Therefore, the whole mandible has been modeled by using Ansys program and stress analysis has been made for only one variation. The large size of the front and stem parts of mandible have been also modeled in Ansys program. The stress distribution is calculated for the whole mandible model and this model is approximately same in the same variation. So models containing the front and stem part of mandible have been established for 16 variations in this program (Fig. 2).

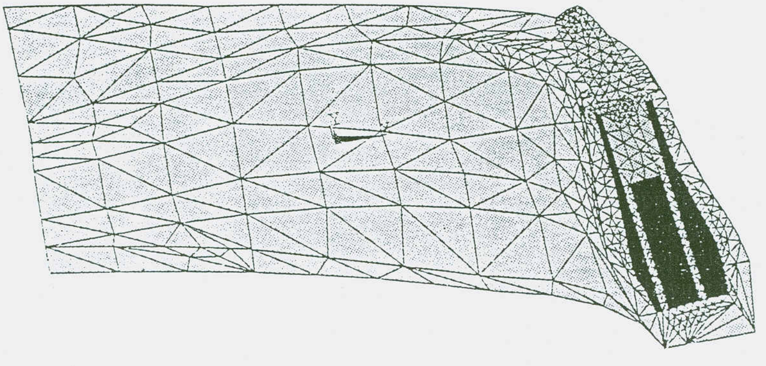
In all of the finite element models of the mandible, the cortical bone that is harder and has high density on outer surface of the mandible and the spongy bone that has low density in the inner section have been considered. It has been assumed that the attachment between the cortical bone and spongy bone is obtained by continuous bone fibers and they are not separated in any case.



Type a-the front part of the mandible



Type b-The whole mandible



Type c- The front and stem part of the mandible

Figure 2. The Finite Element Models

Table 1. The variations and their codes

Head of implant	Inside section of implant	Upper and lower sections are cortical	Upper and lower sections are spongy	Upper sect. is spongy, Lower sect. is cortical	Uppersect. is cortical Lower sect. is spongy
Straight-Headed	Emptied	1	2	3	4
	Filled	5	6	7	8
Inclined-headed	Emptied	9	10	11	12
	Filled	13	14	15	16

Due to the symmetrical geometry and loading, the displacements of the nodes on symmetry surface, along the normal of the surface are assumed to be zero. During biting, the joint region of the mandible and the skull acts like a cantilever joint. Therefore, the closest region to the skull is taken as cantilever in all models. The b type model which includes the whole mandible gives the most realistic results. Since the cantilever is taken from the middle and front parts of the mandible in a and c type models, the moment effects of the reaction forces arise at there regions and therefore, the results are effected to a certain degree.

There are various types of projections and indentations on dental implants. These geometric forms and the material of implant allow implant to coalesce with bone tissue. So the joint surfaces of implant and bone tissue are not separated in ideal case. Therefore, the displacements of the points on the joint surfaces are considered as the same. In practice there is an angle between implant axis and vertical axis in order to obtain harmonious geometry and function between prosthesis tooth and the other teeth. In this study this angle is taken as 7° according to the geometry of mandible and teeth examined Mandible has a symmetry axis. So one of its symmetric parts has been modeled. Displacements of the points in symmetry plane become zero in the vertical direction of this plane. The mandible joint where mandible is connected with skull and so cross section of the model, close to this joint has been assumed to be cantilever.

There are 700 nodes and solid elements, each of them has 8 nodes, in the model set by SAP90. Rotations of all nodes have been taken zero. Modules of elasticity of cortical bone and spongy bone have been accepted to be 19000 N/mm^2 and 1000 N/mm^2, and their poison ratios are taken 0.35 and 0.28 respectively. Tetrahedral elements with 4 nodes have been used in the model set by Ansys program. b type model has been established by using 12226 elements and 2876 nodes and c type model by using approximately 9800 elements and 2400 nodes. The nodes have been allowed to rotate Modulus of elasticity of cortical bone and spongy bone have been considered as, 19000 N/mm^2 and 1500 N/mm^2 and their poison ratio is 0.3. Titanium has been preferred as implant material, for each of the three types of models. Its elasticity modulus is 190000 N/mm^2 its poison ratio is 0.3

Although the direction and magnitude of the forces occurred during chewing process depends on mandible and tooth shape, upper limit value of chewing force for front teeth has been accepted 150N [9] and it has been applied on the upper surface of the implant as uniformly distributed load along implant axis. In all models, mandibular canal has been ignored. As it is shown in figure 3, a cylindrical coordinate system has been used at the hole where implant has been attached, the graphical analysis of stress distributions in the hole points has been performed.

An example of the symbols under the figures is "c1"

c: type of model, 1: number of case showed at table1.

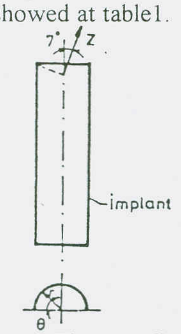


Figure 3. The coordinate system

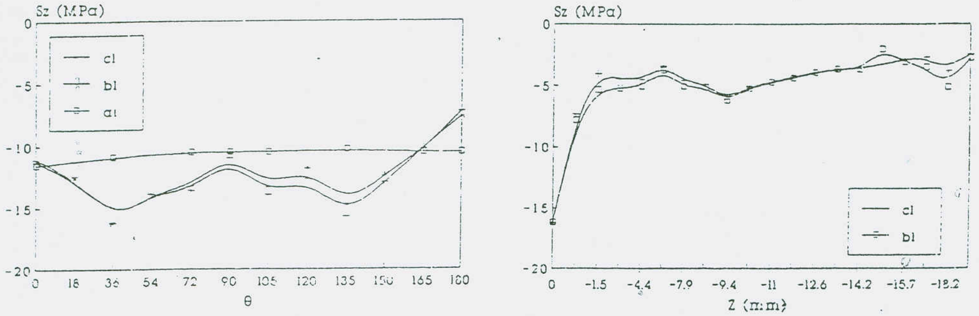
4. RESULTS

It has been consequently determined that values of normal stresses are higher than the values of shear stresses and the highest values of normal stresses are in the z directions. For all cases it is observed that σ_z normal stresses around the hole on the upper surface where the implant enters the mandible are higher than the stresses that occur in the other areas of the hole. Structure failures, such as cracks in the material and sudden differences in the cross section (notch), and other reasons as those maintained can result in stress concentrations. Different mechanical properties of the materials, between which there are connections, can cause the stress concentrations in interfaces. It can be said that this form of stress concentration is an anticipation case through either the results from clinical studies or reasons like those mentioned. In all cases for the models set by SAP90 and in the cases, where only inclined-headed implants have been used, for the models set by Ansys, the highest value of the stress around the hole on the upper surface is at the point where θ equals to zero. It has been observed that the highest stress around the hole on the upper surface occurred at the point where θ is approximately 36°, from the Ansys program for the cases of straight-headed implants.

For the 1st case, the stresses obtained from b type model set by Ansys have been compared with the stresses obtained from a type model set by SAP90 and from c type model set by Ansys (Fig. 4).It has been observed that the results obtained from b and c models are in accordance

The results obtained for the cases where the inclined-headed implants have been used, the stress distributions obtained from the a and c type models for 1^{st} and 9^{th} cases have been compared to each other and it has been found that the desired harmony exist as shown in fig. 5. For the cases where the implant head is straight and the inner part of it is empty (in the mandible, this inner part is filled with spongy bone) the stress values on the upper section of the spongy bone that is in the inner part of the implant are higher than those of the other areas. It can be said that this case arises because of the reduction of the implant cross sectional area and/or higher values of the force acting upon the spongy bone than those acting upon the implant and cortical bone. In the lower level sections the stresses in the implant and mandible decrease. For these cases, the stresses along the hole axis, particularly in the middle areas.

have greater values than those for the cases where the inner part of the implant is not empty (It contains its own material).



a) around the upper surface of the hole **b)** along the hole axis

Figure 4. The stress distribution in all models for the 1st case

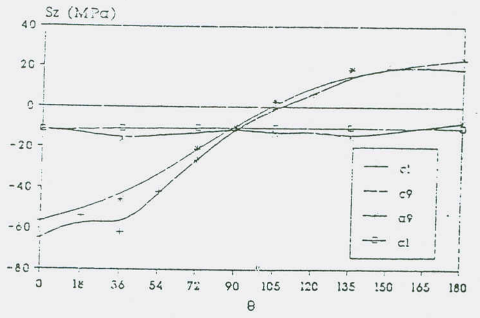
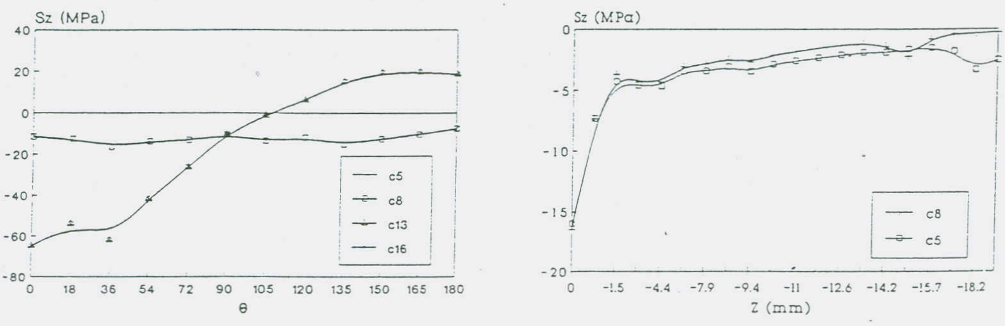


Figure 5. The stress distribution around the upper surface of the hole in the model a and c

For the cases where head of the implant is straight, the stresses occurring at the points around the hole on the upper surface have approximately the same value for the model a. For same cases, for the models b and c, the stress distributions at the same area, however, have not the same form, and they reach peak values at the point where θ is approximately 36°. It can be assumed that these results for model a, only the front part of the mandible is taken into consideration and consequently the effects occurred by the geometry of bone tissue in other section are neglected.

The stresses determined in the cases where implant contacts to the cortical bone at the bottom have been compared with the stresses in the cases where implant contacts to the spongy bone at the bottom. Although the stresses in both cases have the same distribution form on the upper sections, it has been determined that the stresses in cases where implant contact cortical bone at the bottom have greater values because of the different boundary conditions in the lower sections (Fig. 6).



a) around the upper surface of the hole **b)** along the hole axis

Figure 6. The stress distribution in the model c for some cases.

The stresses occurred in cases where the used inclined-headed implants are greater than those occurred in cases where straight-headed implants are used particularly on the upper sections: It can be said that, this is induced by the bending moment occurring because of loading form. The stresses observed on the upper sections, particularly for the model c in some cases have been 6.5 times as great as the stresses in the cases where implant head is straight. (Fig. 7).

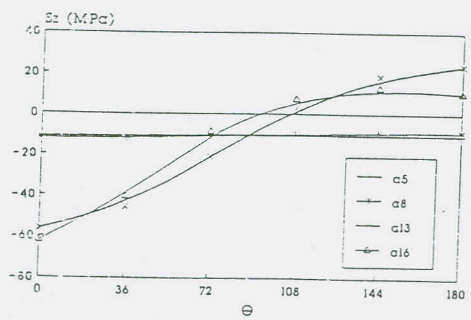


Figure 7. The stress distribution around the upper surface of the hole in the model a

The effect of the bending moment is reduced and so the stress values are decreased in the cases where inclined-headed implants are used. In these parts, the stress values approximate the stress values at the same areas in the cases where straight-headed implants are used because of the effect of compressive forces and they maintain the same negative values at points around hole.

The stresses in the cases where the implant contacts the spongy bone on the upper section of mandible, particularly on the upper sections, have lower values than those occurred in the cases where the upper section consists of cortical bone.

5. CONCLUSIONS

Finally, it can be said that, straight-headed implants provide safer results than inclined-headed ones. Strength values of spongy bone are much lower than those of cortical bone. Therefore in the cases where upper sections consist of spongy bone, the stresses exceed the strength limits, so especially in these cases or in the cases where the cortical bone in the upper surface is very thin, straight-headed implants should be preferred.

The maximum values of σ_z stresses in the cases where the inside section of the implant is filled with spongy bone or own material are approximately the same. So it can not be said that, there is more advantage between these types of implants. But according to the method of medical treatment or any other factors, one of these implants can be preferred. The same comment can be made for the cases where the lower section of the implant contacts the spongy bone or the cortical bone.

REFERENCES

1. H.J.A. Meijer, F.J.M. Starmans, F. Bosman and W.H.A. Steen, A comparison of three finite element models of an edentulous mandible provided with implants, *The Journal of Oral Rehabilitation* **20**, 147-157, 1993.

2. M.R. Rieger, W.K.Adams, G.L. Kinzel, M.O.Brose, Alternative materials for three endosseous implants, *The Journal of Prosthetic Dentistry* **61**, 717-722, 1989.

3. V.K. Goel, S.C. Khera, J.L.Raison, K.H. Chang, Stresses at the dentinoenamel junction of human teeth-A finite element investigation, *The Journal of Prosthetic Dentistry* **66**, 451-459, 1991.

4. L.P. Van Rossen, L.H. Braak, C.de Putter, K. de Groot, Stress-absorbing elements in dental implants, *The Journal of Prosthetic Dentistry* **64**, 198-205, 1990.

5. M.R. Rieger, M. Mayberry, M.O.Brose, Finite element analysis of six endosseous implants, *The Journal of Prosthetic Dentistry* **63**, 671-676, 1990.

6. H.G. El Charkawl, M.T. El Wakad, M.E. Naser, Modification of osseointegrated implants for distal-extension prostheses, *The Journal of Prosthetic Dentistry* **64**, 469-472, 1990.

7. H.J.A Meijer, J.H. Kuiper, F.J.M. Starmans, F.Bosman, Stress distribution around dental implants; influence of the superstructure, the length of the implants; and the height of the mandible, *The Journal of Prosthetic Dentistry* **68**, 96-100, 1992.

8. M.R.Rieger, W.K.Adams, G.L.Kinzel, A finite element survey of eleven endosseous implants, *The Journal of Prosthetic Dentistry* **63**, 457-465, 1990.

9. İ Çuhadaroğlu, *Kuron Köprü Protezi*, 29, İstanbul, 1983.