STRESS DISTRIBUTION IN BONE BED INFLUENCED BY IMPLANT LENGTH AND DIAMETER

L. Himmlová*, T. Goldmann**

* Institute of Dental Research 1st MF and GMH, Prague 2, Czech Republic
** Czech Technical University in Prague, Faculty of Mechanical Engineering, Laboratory of Biomechanics of Man, Prague 6, Czech Republic

himmlova@seznam.cz, goldmann@biomed.fsid.cvut.cz

Abstract: Chewing forces acting on dental implants can result in an undesirable stress in adjacent bone, which in turn can cause bone defects and the eventual implant failure. Mathematical simulation of stress distribution around the implants was used to determine which length and diameter of implants would be best to dissipate these stresses. Computations were made with the finite-element method using 3D models of implants over a range of diameters and lengths. Models were loaded with forces representing an average chewing force in natural direction. Maximum stress areas were located around the implant neck. The stress decrease was the greatest (31.5%) in the diameter range from of 3.6 mm to 4.2 mm. Further stress reduction in case of the 5.0 mm implant was only 16.4%. The increase in the implant length also led to the decrease in the maximum von Mises stress values; this influence, however, was not as pronounced as that of diameter. Within limitations of this study, an increase in the implant diameter decreased the maximum von Mises stress around the implant neck more compared to an increase in the implant length, as a result of a more favorable distribution of the simulated chewing forces applied in this study.

Introduction

The lifetime of a submerged implant has two periods: the unloaded healing and the functional period when withstanding the chewing force. In both these periods implants can fail but for different reasons. The failure in the first period occurs within a short time after insertion of the implant and is associated primarily with inflammation.[1] The failure in the second period takes place after implant loading and is associated primarily with bone loss around the neck area of the implant.[2] Bone loss is thought to result from the magnitude and/or the direction of the load being incorrectly oriented along the long axis of the implant.[2-5]

The implant size influences the area of possible retention in the bone; the factors such as the occlusion, the chewing force, the number of implants and their position within the prosthesis affect the forces acting on the bone adjacent to implants.[3] Holmgren [6] state that implant diameter, shape and load direction influence the stress distribution. Processes accompanied by the reduction of alveolar bone and some anatomical structures (canine fossa, antrum, nasal cavity or mandibular canal), may limit the implant size and/or force their placement into positions where they need angulated abutments [1], and thus may cause their inadequacy to distribute effectively the chewing force.[7]

An applied mechanical force produces the stress and strain in the bone and deforms its structural arrangement.[2] Carter [4] described a hypothesis about the remodeling of cortical bone as a response to a mechanical loading. A bone with dental implants that demonstrates a higher turnover rate compared to the normal setting with teeth may result from repair stimuli caused by compressive and tensile loading damage in tissues adjacent to the implants.[2,8] Isidor [7] claimed that excessive force acting on the implant caused a bone decrease in the surrounding area followed by fibrointegration and resulted in the possible implant release from the bone socket.

On the basis of clinical observations, some authors state that during the first year after the implant loading the regular marginal bone loss around the neck ranges from approximately 0.5 to 1 mm or 1.5 mm.[2,9] Subsequently, the rate of the bone loss is considered either stationary or significantly reduced (bone loss approximately 0.1 mm)[2] or the resorption of the bone crest continues and the implant is lost within a few years. These findings are in accordance with recent 3D mathematical models of dental implants under the non-axisymmetric loading [10], which indicates the maximum stress around the implant cervix.[5,11]

This simulation study was performed with the objective to compare the influence of the diameter and the length of an implant on the stress distribution around the implant. For this purpose, situations were modeled using 3D graphics where single cylindrical dental implants of various diameters and lengths were vertically inserted into the molar part of mandibles. The stress distribution in the bone socket after loading by averaged chewing forces was computed by the finite-element analysis (FEA).

Materials and Methods

The finite-element method was used to analyze stress around cylindrical dental implants inserted in the

 ISSN: 1727-1983

molar part of the mandible. This method is an analytical tool that is widely used for mathematical modeling of real bodies.[12]

Geometrical 3D models of the implant and part of the mandible as well as material properties of the bone were simplified to save the computing time and memory. Geometrical simplifications used to reduce the computer time and memory did not relevantly affect the accuracy of the computation of the proposed parametric study from the point of view of the stress distribution.[13] The simplifications used within this research did not influence the results since all the models were subjects of the same simplifications.

The selected 3D models represented commonly available submerged titanium (E = 1.1e5 MPa, μ = 0.32) [14] solid cylindrical dental implants without threads (IMZ implants, similar to ITI Bonefit, etc.) with a bioactive coating inserted into the molar part of a simulated mandible in a vertical position (Fig. 1). For the purpose of this study, the implant shape was simplified to a plain cylinder and the bone to a prism having a quadrangular base and walls of irregular octagon (Fig. 1). The implant model with the diameter of 3.6 mm and the lengths of 8,10,12,14, 16,17, and 18 mm was applied to the investigation of the influence of the length factor. The influence of the diameter was modeled by using implants with the length of 12 mm and the diameters of 2.9 mm, 3.6 mm, 4.2 mm, 5.0 mm, 5.5 mm, 6.0 mm and 6.5 mm. The implant surface was modeled with a bioactive coating providing an immovable junction between the implant and the bone. For this reason the tied (term in users manual ABAQUS version 5.8) contact in ABAQUS software was chosen because of its firm connection between contact bodies (implant and bone socket surfaces).

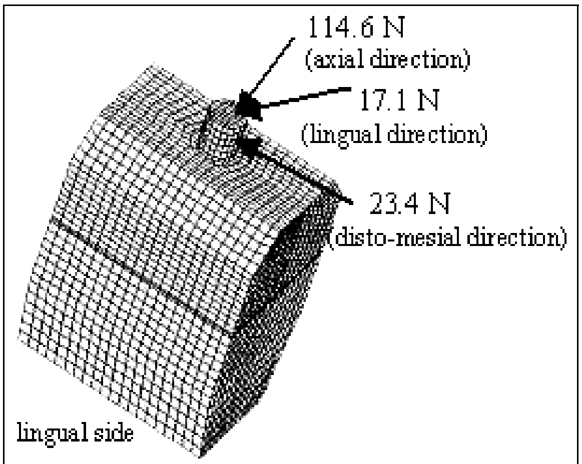

Figure 1: 3D model of cylindrical dental implant without threads in molar region of mandible under loading.

The mesial and distal borders of the end of the modeled section of the mandible were constrained so that the displacement of nodes in all directions was equal to zero.[12] The bone was considered a homogeneous, isotropic material with the character of cortical bone (E = 1.37e4 MPa, μ = 0.3)[14] in the whole volume. The models consisted of 15 000 - 20 000 elements, depending on the implant size.

To generate this 3D model, the pre- and post-processor ABAQUS CAE, which is a part of the FEM software ABAQUS, version 5.8 (HKS Inc., Pawtucket, Rhode Island, USA) was used. This pre- and post-processor enables a parametric definition of the geometry and the FE mesh (users manual ABAQUS version 5.8). The model parameterization was used to investigate certain property of the modeled subject in dependency with the parameter.

The implant loading in 3D with forces of 17.1 N, 114.6 N and 23.4 N, in a vestibulo-oral direction, an axial direction and a disto-mesial direction, respectively (Fig. 1), simulated average chewing force in natural, oblique direction. These components represented chewing force 118.2 N in the angle of approx. 75^{o} to the occlusal plane. This 3D loading acted on the center of the upper surface of the abutment at a distance of 4.5 mm from the upper margin of the bone. The force magnitudes, as well as the acting point, were chosen with respect to the measurement of Mericske-Stern.[15]

The pre- and post-processing were carried out on the Silicon Graphics Indigo II workstation (SGI, Mountain View, California, USA) of the Faculty of Mechanical Engineering, Czech Technical University (CTU) and all the computations on SGI Power Challenge L (SGI, Mountain View, California, USA) at the Center for Intensive Computation of CTU.

All computations were performed for the 3D models mentioned above. The effective (von Mises) stress (MPa) at the implant-bone interface was computed by FEA using ABAQUS version 5.8 (HKS Inc., Pawtucket, Rhode Island, USA). Values for the three most stressed model elements for each variation of the implant diameter and length were averaged to eliminate potential minor inaccuracies caused during computation. These averaged values of effective (von Mises) stress were compared with the value concerning the reference implant with the diameter of 3.6 mm and the length of 12 mm. The resulting values were the relative stress in % of the computed value for the reference implant (=100%). This relative stress (in per cent) was used to determine its dependence on both the diameter and the length of the implant.

Results

The mathematical analysis showed an uneven stress distribution inside the socket around the loaded implants. The elements exposed to the maximum stress were located around the neck of the implant on the mesio-lingual rim of the bone socket (area indicated by red colour in Fig. 2). This location was identical for all implant sizes.

A comparison of the areas with the maximum stress for implants of the same length but different diameters showed distinct differences. In the case of the 2.9 mm diameter implant and the 6.5 mm diameter implant, the area of maximum stress was not only reduced but the actual values computed for the same loading were smaller.

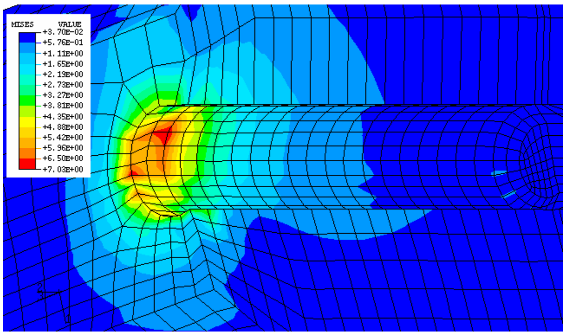

Figure 2: Uneven distribution of the stress around the loaded implant. View is from inside of the bone socket.

The plotting of relative stress (in per cent) for implant diameters varying from 2.9 mm to 6.5 mm showed an exponential regression curve, indicating a marked influence of the implant diameter on stress in the simulated bone (Fig. 3). The relative stress acting in the bone around the implant with a diameter of 4.2 mm was smaller by 31.5% than in the case of the reference implant (the diameter of 3.6 mm). Further stress reduction with the 5.0 mm implant represented only additional 16.4 % and continued to decrease for larger diameters. The use of an implant with the diameter 6.5 mm resulted in the reduction of the maximum values of stress by almost 60%.

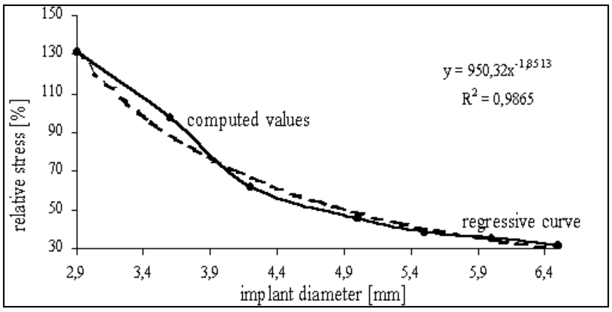

Figure 3: Exponential regression curve indicating a marked influence of the implant diameter on stress in the simulated bone.

The model for implants with the same diameter (3.6 mm) but with the different length showed a substantially lower effect of the length. As exemplified by the implants with the lengths of 8 mm and 17 mm, there was only a small difference in the area affected by maximum stress, and the values fall within a similar range.

The relation between relative stress (in percents) and the implant length showed a similar curve as in the case of the variable diameters. But compared with the curve concerning the diameter there was an evidently smaller effect of the implant length on stress in the bone indicated by a less steep curve (Fig. 4). The relative stress acting in the bone around the implant with the length of 17 mm was by 22.9% smaller than that around the reference implant (with the length of 12 mm). As shown by the examples of the implants with the length of 8 and 17 mm, there was a difference of only 7.3%.

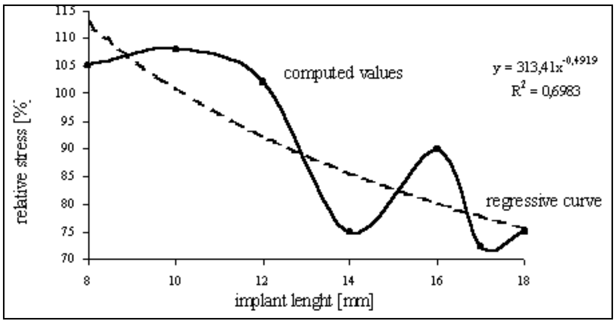

Figure 4: Exponential regression curve indicating an influence of the implant length on stress in the simulated bone.

Discussion

The finite element method is one of the most frequently used methods in stress analysis in many branches of industry and science.[12] In experimental medicine, it is used for analyzing hip joints, knee prostheses or dental implants.[5,10,11] The results of the FEA computation depend on many individual factors (material properties, boundary conditions, interface definition, etc.) and also on the overall approach to the model.[12] It is apparent that the presented model was only an approximation to reality. The application of a 3D model simulation with the non-symmetric loading by the chewing force on a dental implant resulted in a more satisfactory modeling of the "real state" than that achieved with 2D models used in other studies.[6]

This parametric model attempts to make possible the comparison of implants of various sizes, however, the absolute values of stress cannot be related to results computed under different conditions. The simplification of the model [13], for example, the implant in the shape of a cylinder rather than a screw or other shapes commonly used in clinical practice and the simplification of material properties (bone was homogeneous, isotropic with the character of cortical bone in the whole volume) made possible the required computer time to be reduced without affecting the purpose of this study – to establish the relative importance of the implant length and diameter.

The results of this study complement the already published facts that stress distribution in the bone around the implant depends on the shape and the size of the endosseal part.[3,6] The results of this simulation study have shown that the implant diameter was more important than its length. This probably results from the fact that the stress distribution inside the socket is uneven, the elements exposed to maximum stress are

ISSN: 1727-1983 © 2005 IFMBE

located around the neck5 and therefore the chewing force should be better dissipated by the wider area in the cervical part of the implant. Holmgren [6] concluded that larger diameters are not necessary. But implants used by him had diameters ranging from 3.8 mm to 6.5 mm. Within this range, similar results were obtained in this study.

It is known from clinical experience that it is not always possible to insert the implants into optimal position and place favorable from the loading point of view and it is necessary to use angulated abutments.[1] Carter's hypothesis [4] claims the bone strain above 3000 microstrains to be troublesome for the bone, leading to a hypertrophic response and, above 4000 microstrains to cause local overloading followed by bone loss in the locations of the acting force. The values obtained by computer-assisted simulation in this parametric study for size variations of the cylindrical dental implants cannot be directly compared with those of Carter's hypothesis due to the simplicity of the model. However, the location of zones with higher stress around the implant neck may indicate a danger of overloading in this area, as all size variations displayed maximum values of stress. The elements exposed to maximum stress were located at places to which most of non-axial chewing force was transferred (for example forces acting in vestibulo-oral and disto-mesial directions that are associated with grinding movements - in comparison with axial loading during chopping movements). This situation corresponds to non-parametric6 computerized models of loaded dental implants, where the utmost strain (and according to Carter's hypothesis this is where bone overloading occurs) acts in the surroundings of the implant neck.[5,11]

Generally, the assessment of the dental implants from physical, biological, and technological viewpoints made possible the time prognosis of implants in the oral cavity to be improved.

Conclusions

This simulation study showed that increased implant diameter better dissipated the chewing force and decreased the stress around the implant neck. The highest reduction in stress in comparison with the reference implant (100%, diameter of 3.6 mm) was obtained for the diameter of 4.2 mm (a decrease by 31.5 %). From a biomechanical perspective, the optimum choice was an implant with the maximum possible diameter allowed by the anatomy. In this study, the impact of implant length was less notable.

Acknowledgements

This study was supported by the Grant Agency Min. of Education of the CR grant No: 6840770012.

References

[1] SETHI A, KAUS T, SOCHOR P. (2000): 'The use of angulated abutments in implant dentistry: Five year clinical results of an ongoing prospective study', *Int J Oral Maxillofac Implants*, **15**, pp. 801-10

[2] WISKOTT HWA, BELSER UC. (1999): 'Lack of integration of smooth titanium surfaces: a working hypothesis based on strains generated in the surrounding bone', *Clin Oral Impl Res.*, **10**, pp. 429-44

[3] AKPINAR I, DEMIREL F, PARNAS L, SAHIN S. (1996): 'A comparison of stress and strain distribution characteristics of two different rigid implant designs for distal-extension fixed prostheses', *Quintessence Int.*, **27**, pp. 11-7

[4] CARTER DR, VAN DER MEULEN MCH, BEAUPRÉ GS. (1996): 'Mechanical factors in bone growth and development', *Bone*, **18**, pp. 5S-10S

[5] MEIJER HJA, STARMANS FJM, STEEN WHA, BOSMAN F. (1996): 'Loading conditions of endosseous implants in an edentulous human mandible: a three-dimensional, finite-element study', *J Oral Rehabil*., **23**, pp. 757-63

[6] HOLMGREN EP, SECKINGER RJ, KILGREN LM, MANTE F. (1998): 'Evaluating parameters of osseointegrated dental implants using finite element analysis – a two dimensional comparative study examining the effects of implant diameter, implant shape, and load direction', *J Oral Implantol*., **24**, pp. 80-8

[7] ISIDOR F. (1996): 'Loss of osseointegration caused by occlusal load of oral implants. A clinical and radiographic study in monkeys', *Clin Oral Impl Res*., **7**, pp. 143-52

[8] ROBERTS WE, TURLEY PK, BREZNIAK N, FELDER PJ. (1987): 'Bone physiology and metabolism', J Calif Dent Assoc., **15**, pp. 54-61

[9] DAVARPANAH M, MARTINEZ H,TECUCIANU JF. (2000): 'Apical-coronal implant position: Recent Surgical Proposals. Technical note', Int J Oral Maxilofac Implants, **15**, pp., 865-72

[10] HOLMES DC, LOFTUS JT. (1997): 'Influence of bone quality on stress distribution for endosseous implants', J Oral Implantol., **23**, pp. 104-11

[11] LAI H, ZHANG F, ZHANG B, YANG C, XUE M. (1998): 'Influence of percentage of osseointegration on stress distribution around dental implants', Chin J Dent Res., **1**, pp. 7-11

[12] BATHE K.J. (1996): 'Finite Element Procedures', Merrill Peterson (Ed), (Prentice-Hall, Inc., New Jersey).

[13] TEIXEIRA ER, SATO Y, AKAGAWA Y, SHINDOI N. (1998): 'A comparative evaluation of mandibular finite element models with different lengths and elements for implants biomechanics', J Oral Rehabil., **25**, pp. 299-303

[14] VALENTA, J. a kol. (1985): 'Biomechanika' (Academia, Prague).
[15] MERICSKE-STERN R, PIOTTI M, SIRTES G. (1996): '3-D in vivo force measurements on mandibular implants supporting overdentures', Clin Oral Impl Res., **7**, pp. 387-96

ISSN: 1727-1983 © 2005 IFMBE