ISSN 2395-1621

Stress Analysis and Optimization of Dental Implants

#1R.S.Jadhav, #2Dr.S.Y.Gajjal, #3Prof.V.K.Kulloli, #4Prof.P.N.Dhatrak

[1]jadhav.rsa@gmail.com

[2]shekhar.gajjal@sinhgad.edu

[4]pankajd123@gmail.com

Department of Mechanical Engineering
NBN Sinhgad School of Engineering, Pune , India
MIT College of Engineering pune, India

ABSTRACT

Finite element analyses were performed for various shapes of dental implant to study effects on stress distribution generated in the surrounding jaw bone and to determine an optimal thread shape for even stress distribution.The purpose of this study was to derive alternative implant shapes which could minimize the stress concentration at the shoulder level of the implant. A topological shape optimization technique which mimics biological growth, was used in conjunction with the finite element (FE) method to optimize the shape of a dental implant under loads. Shape optimization of the implant was carried out using a 2-dimensional (2D) FE model of the mandible

Keywords—Dental implants, Effective Stress Distribution ,Finite Element Analysis Osseo integration Shape Optimization

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

I. Introduction

Dental implant is a prosthetic replacement for a missing tooth. Our natural teeth consist of crown and a root. The crown is the visible part that is covered with white enamel. Supporting the crown is the tooth root which extends into the jaw bone and this root is effectively replaced by an implant. Implant is inserted directly into the jaw bone.

Branemark introduced the concept of osseointegration. Osseointegration is process in which bone cells attach themselves directly to implant surface, locking the implant into the jaw bone. The application of occlusal forces induces stresses and strains within the implant-prosthesis complex and affects the bone remodeling process around implants. To achieve optimized biomechanical conditions for implant-supported prostheses, i.e Stress distribution along the jaw bone should be even important consideration of the biomechanical factors that influence prosthesis success is essential

Many different methods have been used to study the stress/strains in bone and dental implants.Some of them arePhoto-elasticity, Strain-gauge measurements which provide accurate data regarding strains only at the specific location of the gauge. Finite element analysis (FEA) is capable of providing detailed quantitative data at any location within mathematical model. Thus FEA has become a valuable analytical tool in the assessment of implant systems in dentistry. The principal difficulty in simulating the mechanical behavior of dental implants lies in the modeling of human maxilla and mandible and its response to applied load. Certain assumptions are needed to make the modeling and solving process possible and these involve many factors which will potentially influence the accuracy of the FEA results: (1) detailed geometry of the implant and surrounding bone to be modeled, (2) boundary conditions, (3) material properties, (4) loading conditions, (5) interface between bone and implant, (6) test, (7) validation

II. TYPICAL DENTAL IMPLANT

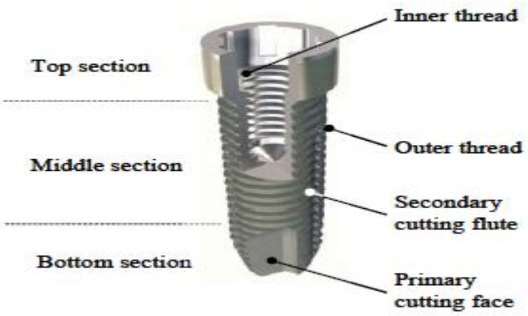

Figure 1

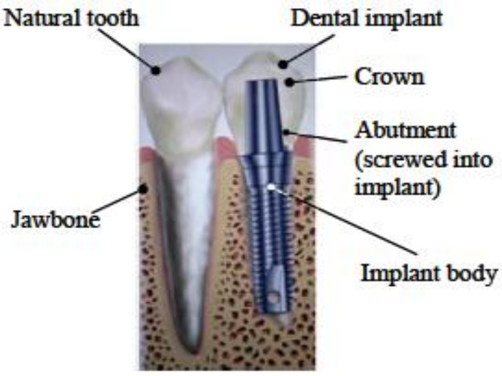

Figure 2

III.MATERIAL PROPETIES

Titanium and titanium alloys are the most preferred materials for dental implants because of their good biocompatibility, excellent corrosion resistance and suitable mechanical properties. The existing titanium implants still have several drawbacks. Firstly, the bonding strength at the interface between the implant and the bone is not high enough and the biological fixation has not been achieved. Secondly, there exist mismatches between the elastic modulus of the implant and of the bone. A stress shielding or concentration can be easily induced on the interface and results in a potential risk to the long-term stability of the implant.

Current dental implants are mainly fabricated using dense titanium and titanium alloys, which have no features representing the difference between the inner and outer layers of the mandible or that between their elastic modulus. The use of porous metal implants for medical applications has two main advantages. One is the similar elastic modulus to the bone, which helps to prevent the stress shielding effect at the bone interfaces. The other is that it can provide a structural condition for the bone ingrowth to achieve biological fixationHowever, the low mechanical strength limits their further applications in the implanting industry, the table below shows the material Properties

Table 1

	Young's Modulus(Gpa)	**Poission's ratio**
Cortical bone	13.7	.3
cancellous bone	1.37	.3
Implant(Titanium)	110	.33

IV. APPLICATION OF FINITE ELEMENT ANALYSIS

4.1 MODDELLING OF AN IMPLANT

The attractive feature of finite element is the close physical resemblance between the actual structure and its finite element model. Excessive simplifications in geometry will inevitably result in considerable inaccuracy. The model is not simply an abstraction; therefore, experience and good engineering judgment are needed to define a good model. Whether to perform a two-dimensional (2-D) or three-dimensional (3-D) finite element model for the study is a significant query in FEA. It is usually suggested that, when comparing the qualitative results of one case with respect to another, a 2-D model is efficient The first step in FEA modeling is to represent the geometry of interest in the computer. In some 2-D FEA studies, the bone was modeled as a simplified rectangular configuration with the implant (Fig. below). The mandible was treated as an arch with rectangular section or a simplified segment as cancellous core surrounded by a 1.3-mm cortical layer with the overall dimensions of this block were 23.4 mm in height, 25.6 mm in mesiodistal length, and 9.0 mm in buccolingual width in 3-D FEA models

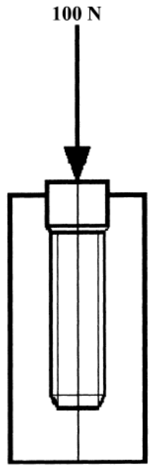

Figure 3

4.2 BOUNDARY CONDITIONS

Finite element model should be fully constrained to get the proper results. Zero displacement constraints must be placed on some boundaries of the model to ensure an equilibrium solution. The constraints should be placed on nodes that are far away from the region of interest to prevent the stress or strain fields associated with reaction forces from overlapping with the bone-implant interface.

Symmetry boundary conditions can be employed at the nodes on the symmetry plane. Models are to be constrained in all directions at the nodes on the mesial bone surface.

4.3 LOADING CONDITIONS

Different types of loads are been applied on the implant .We know that mastication (biting of food) involves a repeated pattern of cyclic impacts that causes loading to the implant components and distributes the force to the bone interface. When we apply FE analysis to dental implants, it is important to consider not only axial loads and horizontal forces but also a combined load (oblique occlusal force) because the latter represents more realistic masticatory pattern and will generate considerable localized stresses in compact bone. Bite force studies indicated considerable variation from one area of the mouth to another and from one individual to the next. In the premolar region, reported values of maximal bite force range from 181-608 N.

V. PROPOSED WORK

The models of the different implant shapes are to be modeled in the PRO-E software package. Five different models of the same implant company are been chosen and modeled. This implants has parameters like diameters and lengths. This parameters are been varied to reduce the maximum stress along the jaw bone. These implants are to be optimized i.e. by varing the diameter and length of the implant , the stress distribution along the jaw bone is to be found out. A proper relevant optimization technique will be used for the optimization process

ACKNOWLEDGEMENT

it gives me great pleasure to present a paper on ‘stress analysis and optimization of dental implants’. In this worknumber of hands are helping me directly and indirectly. Therefore it becomes my duty to express my gratitude towards them.I thank my guide, Prof. V. K. Kulloliand co-guide Prof. P. N. Dhatarkforhelping me with the well concept- utilization and for being a constant source of help and guidanceFinally I would like to thank our HOD Dr. S.Y. Gajjal and Principal Dr. S.D. Markande and the entire staff members of mechanical engineering department for their help

REFERENCES

1. Siegele D, SolteszU.Numerical investigations of the influence of implant shape on stress distribution in the jaw bone. Int J Oral Maxillofac Implants 1989;4:333–340.
2. H.-j. Chun, s.-y. Cheong, j.-h. Han, s.-j. Heo, j.-p. Chung, i.-c. Rhyu, Y.-c. Choi, h.-k. Baik, y. Ku & m.-h. Kim Evaluation of design parameters of osseointegrated dental implants using finite element analysis
3. Holmgren, e.p., seckinger, r.j., kilgren, l.m. &mante, f. (1998) Evaluating parameters of osseointegrated dental implant using finite element analysis a two-dimensional comparative study examining the effects of implant diameter, implant shape,and load direction. Journal of Oral Implantology, 24, 80.
4. Van Osterwyck H, Duyck J, Vander S, Vander PG, Decoomans M, Lieven S, Puers R, Naert L. The influence of bone mechanical properties and implant fixation upon bone loading around oral implants [J]. Clin Oral Implants Res, 1998, 9(6):407-412.
5. Geng J, Tan K B C, Liu G. Application of finite element analysis in implant dentistry: a review of the literature [J]. J Prosthet Dent, 2001, 85(6):585–598.

Authors

R.S.Jadhavis a research student of M.E in Design Engineering at NBN Sinhgad School of engineering ,AmbegaonBk, pune

(e-mail: Jadhav.rsa@gmail.com).

V.K.Kulloli is working as Asst.Professor atNBN Sinhgad School of engineering ,AmbegaonBk, pune

Dr. S.Y.Gajjalis working as PG Head at NBN Sinhgad School of engineering ,AmbegaonBk, pune

(Email:- shekhar.gajjal@sinhgad.edu)

P.N.Dhatrak is presently working as Asst. Professor at MIT College Of Engineering Pune

(Email:- pankajd123@gmail.com)