ISSN 2395-1621

Optimisation of Dental Implant

#1S.P.Gosavi, #2Prof.P.N.Dhatrak, #3Prof.K.M.Narkar

1g.shreyas@rediffmail.com
2Pankajd123@gmail.com
3knarkar@gmail.com

#123Mechanical Engineering Department, Dr.D.Y.Patil College of Engineering, Pune, India

ABSTRACT

Dental implants are used as restoration of lost teeth caused due to accidental injuries, deformation due to infections. Implant geometry is an important objective in biomechanical optimization of dental implant. The factors which determine success or failure of a dental implant is the manner in which stresses are transferred to the surrounding bone through the interface. The dental implant once osseointegrated behaves in a manner similar to that of natural teeth as it is exposed to static and dynamic loadings continuously. The forces transmitted during osseointegration of dental implant are directed on to the jaw bone as compared to the natural teeth where there is a healthy periodontium. These forces induce residual stresses which may result in microfracture at the interface of bone-implant, fracture of implant, components of implant system may get loose and unwanted bone resorption. Hence, it is essential to understand stress concentration on implants, in surrounding bone and functional parameters such as implant diameter, profile, length etc. affecting them. The aim is to optimize functional parameters such as diameter, length of dental implant system by using Non-Traditional Optimization Technique and analyse using Finite Element Analysis so as to withstand and minimize the stress induced as well as transferred along bone implant interface increasing the implant's success rate.

***Keywords*— Dental implant, Stress Analysis, Functional parameter reduction, FEA, 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

Dentistry plays a important role in replacing missing teeth in form of dental implants. Dental implants are insertions into a patients mouth's bone to which an artificial tooth crown is attached. The bonding between the implant and bone must be rigid enough to bear the stress due to different loading conditions. Design and Insertion technique are the two important factors which determine success of implant. Implant according to invention seeks mechanical properties and stress distribution in bone that are similar to those of natural tooth.

Excessive stress transferred from implant to bone may prone to damage .Hence for a better bone implant interfacing threaded part of implant must be self tapping for better integrity and increase in contact surface area. Many factors have been found to influence this interfacial bonding between the implant and bone and thus the success of implants.

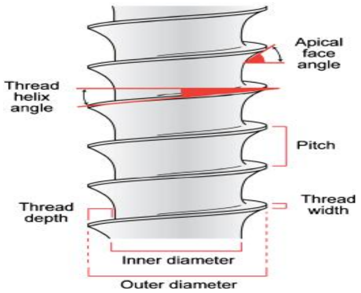

Fig.1 Basic Implant Geometry

LITERATURE REVIEW

Saluja et al studied the effect of length and diameter on stress distribution by creating a 3D solid model of Indigeneous titanium dental implant and mandible and subjecting to FEA using Ansys. Implant length does not affect Stress concentration and distribution. Increase in diameter increases surface contact area thus making implant stable and reducing stress pattern[1]. Arsalanlo et al investigated the effect of different thread types on loading in implant abutment interface**.** Models with varying diameter and length were analysed using FEA to understand stress distribution They found that increase in thread depth enhances stress distribution on the other hand decrease in thread pitch reduces implant stability.[2]

Zhihong Mao evaluated the effect of abutment & fixed screw on dental implant system by modeling of three screw-type dental implant systems with three fixed screw diameter sizes and subjected to static occlusal force of 100N with 15^o for FEA. Stress increases with improper size of abutment & fixed screw [3]. Giuseppe et al evaluated the influence of implant design parameters such as diameter, thread type, length on load transfer mechanism of oessointegration in dental implant by analyzing and comparing ten different types of implants (varying length, diameter, thread shape and geometry) .He concluded that such as implant diameter, length, thread shape highly influence design parameters in load transfer on Implant[4].

Ghorpade et al carried a literature survey to analyse role of design parameters related to bone –implant interface on stress distribution in FEA and experimental studies. Modeling approach and implant geometry are influential factors for the accuracy of analysis[5]. Ausiello et al investigated influence of implant design factors for implant longevity.DOE approach was adopted for automatic generation of different implant designs and analysis using FEA.Implant stability is dependent on thread-width and thickness by correlating design parameters with each other[6]. Desai et al determined stresses in bone implant interface using various dental implant designs.Modeling and analysis of eight types of implants with different thread designs to evaluate stresses and strains around the implants[7]. Bahrami et al evaluated stress distribution in bone implants interface and dental prosthesis. More thread depth to increase contact surface area can reduce stresses in surrounding bone[8]. Szajek et al Presentation of an optimization procedure for minimizing diameter of two-component implantology system. Two component implant model based on FE analysis was optimized and incorporated into the hybrid optimization procedure (genetic and Hooke-Jevees algorithm). The presented hybrid technique optimized diameter without disturbing the defined boundaries[9]. Mohammed et al studied stress distribution in dental implants by modeling and analysis of different implant designs by varying length and diameter subjecting to 50N tensile,100N compressive, 20N bending. Implant diameter and length, crestal bone geometry and placement site affect the load transfer mechanism[10].

Oğuz et al Evaluated effect of different implant thread designs on stress distribution at bone implant interface by modeling and analysis of 3D solid screw implants for four different thread forms V, Square, Butress , Reverse Butress with same implant design parameters number of threads, length, position, height (D), and pitch (P) to analyse the stress distribution[11]. Mansour et al studied stress distribution around tapered and cylindrical threaded implant geometries. The study was divided in modeling and stress distribution analysis. Tapered implants and triangular thread form showed high tensile and compressive stress than non-threaded implants, square thread and straight body form [12]. Abuhusein et al carried out review study to study the effect of implant macrodesign features such as thread factors (thread geometry) thread shape, thread pitch, depth, thickness, face, helix angle on implant analysis using FEA[13].

Kong et al evaluated the cylinder implant thread height and width using 3D finite element analysis. He concluded that in design of a screw type implant optimum thread height is a more important factor than thread width for reduction of stress in bone[14]. Mansour et al in his work to studied stress distribution around tapered and cylindrical threaded implant geometries using three-dimensional finite element stress analysis[15]. Baggi et al studied to analyze the influence of implant diameter and length on stress distribution. Implant shape, geometry and bone resorption possibly affect the load transfer mechanism leading to failure[16].

The success or failure of implant is determined by the manner the stresses at the bone implant interface transfer to surrounding bones. The long term success of implant, its reliability and stability strongly depends on implant implant bone interface.

Implant geometry is an important objective in biomechanical optimization of dental implants.It is necessary to evaluate thread design of dental implant.From the above literature review study the following conclusions can be drawn for the stability and longevity of Dental Implant:

i. The durability as well as the stability of Dental implant depends on oessointegration which is solely dependent on the bone implant interface. Hence bone implant interface is one of the regions to be considered.
ii. Secondly the use of the threaded implant may enlarge the contact area between the implant and bone, increase implant mobility, and help dissipating interfacial stresses
iii. Likewise, the stepped fixture is suggested to create favorable load distribution due to its nature of root form.
iv. There has been insufficient research focusing on the pattern of load transfer and the failures with the effect of implant length correlated with the force transmission area of implant bone interface.
v. Analyzing the corelating stress distribution between implant geometry and implant diameter may help in optimization.

Hence as a work study Optimisation Of Single Dental Implant Thread Profile for minimization of Stress Distribution is undertaken.

I. MATERIALS AND METHODOLOGY

This study was performed in following steps which included creating solid models of implants, creating finite element model and analyzing the process of load transfer and stress distribution.

i.Designing implant Models

CAD models of five screw-type dental implant systems

With diameter Φ 3.5 mm , Φ 3.8 mm, Φ 4.0mm, Φ 4.5 mm, Φ 4.8 mm and Φ 5.0mm were created. All the other implant parameters such as length ,type of thread pitch, depth, helix angle were kept constant for alll the implants. This was done to study the stress distribution in implant screw for varying values of diameter.

Implant Screw diameters sizes used are (M1- 3.5mm, M2- 3.8mm, M3- 4.0mm, M4 – 4.5mm ,M5-4.8mm and M6- 5.0mm). The dental implant system is treated as isotropic homogeneous elastic materials. A static vertical load of lOO N is applied on to the fixed screw.The implant models were constructed using the SOLIDWORKS version 12.0. .
The material of the implant was assumed to be isotropic and linearly elastic.

ii. ***Finite element analysis of the models under load***

The Finite Element Linear static analysis was performed by meshing software ANSYS version 15.0 and the elements used in meshing of all three-dimensional models are Tetrahedral element . Increasing density of mesh increases the accuracy of obtained stress levels in higher stress areas as well as increasing no of elements improves geometry representation.
Table 2 lists the number of nodes and elements of the 5 implant models with indication to the implant model geometry type. The material properties used in this study are listed in Table 1.

Table.No.1. Material Properties

Implant	Young's Modulus (MPa)	Poisson's Ratio ν
Titanium Ti6Al4V	114,000	0.35
Body Ti6Al4V	114000	0.35

Table.No.2 Five Models, Dimensions, No of Nodes

Model #	Diameter (mm)	Length (mm)	No. of nodes	No. of elements
M1	3.5	8.5	88748	52349
M2	3.8	8.5	92924	55601
M3	4.0	8.5	110881	66717
M4	4.5	8.5	114027	68780
M5	4.8	8.5	124093	75240
M6	5.0	8.5	128915	78408

Design curves were obtained from this study. Curve fitting was done using graphical as well as using MATLAB Least Square Method .

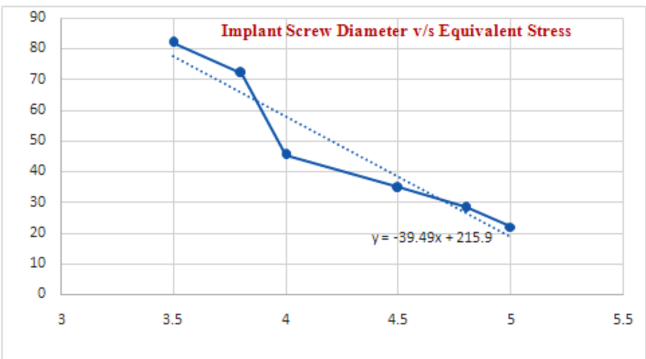

Fig.2 Graph of Implant Screw Diameter v/s Equivalent Stress

This analysis may help in selecting the suitable implant geometry to be used with patient jaw-bone conditions and limit of stresses can be withstand.
By design curve obtained from this study an equation was obtained as an interrelation between implant diameter and equivalent stress.

IV.OPTIMIZATION USING GENETIC ALGORITHM

The genetic algorithm (GA) is a search procedure based on the mechanics of genetics and natural selection that is routinely used to generate useful solutions to optimization and search problems. It generates solutions to optimization problems using techniques inspired by natural evolution, such as inheritance, mutation, selection, and crossover. Genetic algorithms are one of the best ways to solve a problem for which little is known.It works on the Darwins principle of " Survival of the fittest".
The steps involved in optimization using Genetic Algorithm to arrive at a fittest solution involve

i. Function Definition :
The global independent parameters were defined:
a. implant screw diameter,
b. implant length,
c. thread pitch,
d. thread depth,
e. thread angle,

As the main goal of this study was optimization of diameter of implant hence the objective function defined is a single Objective function obtained from curve fitting using least square method with length as a constraint for stress reduction.

$f(y) = -39.4947x + 215.9$ ………….. (1)

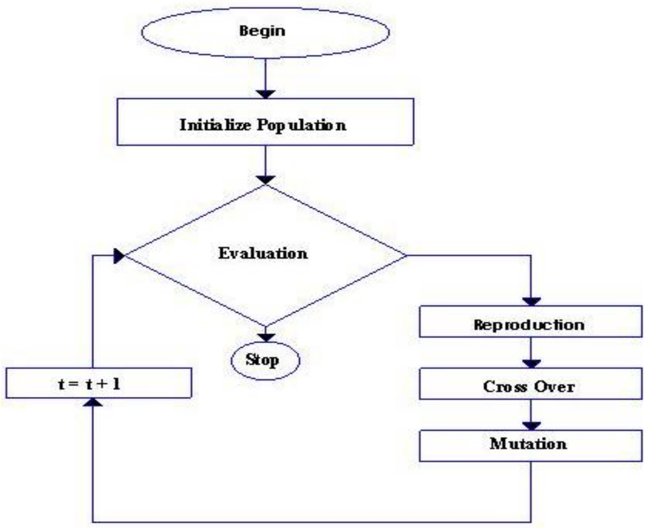

Fig.3 Flow Chart – Working Principle of GA

ii.Initialization :
It starts with random selection of parent chromosomes which are a combination of binary digits having some string length.
They serve as chromosome genes type.
The chromosomes selected comprised of 5 bits string.

Chromosome1 : 1 0 0 0 1

These individuals are in encoded form.

iii. Evaluation:
All the fitness values are calculated from the fitness function and are ranked as per the descending order of fitness value.
iv.Cross-over:
A percentage of the population is selected for combination and assigned random mates in between the parent chromosome. A random crossover point is selected and a pair of new solutions are produced.

Chromosome	Cross over
1] 1 0 0 0 1	1 0 0 1 1
2] 0 1 0 1 1	0 1 0 0 1

The selection occurs with a given probability on the base of fitness functions. The fitness function plays a role of the environment to distinguish between good and bad solutions. The recombination is carried out after selection process is finished. It combines, with predefined probability, the features of two selected parent chromosomes forming similar children. After recombination offspring undergoes to mutation.
v. Mutation:
A small change is made in chromosomes formed after crossover. Mutation refers to the creation of a new chromosome from one and only one individual with predefined probability.

Chromosome	Mutation
1] 1 0 0 0 1	1 0 0 1 1
2] 0 1 0 1 1	0 1 1 1 1

While cross over and mutation the newly formed chromosome i.e the individual should not be similar to parent chromosome.
After three operators are carried the offspring is inserted into the population, replacing the parent chromosomes in which they were derived from, producing a new generation. This cycle is performed until the optimization criterion is met.
The population is successively 'optimised' after a number of generations.
The first iteration has been solved manually which is to be implemented using MATLAB or ABAQUS.

V. RESULTS

The 6 cylindrical implants screws that were analyzed by finite element simulations exhibited different stress-based biomechanical behaviour, dependent on size parameters. The implant with smaller screw diameter showed highest value of Equivalent stress whereas the stress value kept on reducing with increase in diameter. The maximum value of stress obtained was 80MPa.

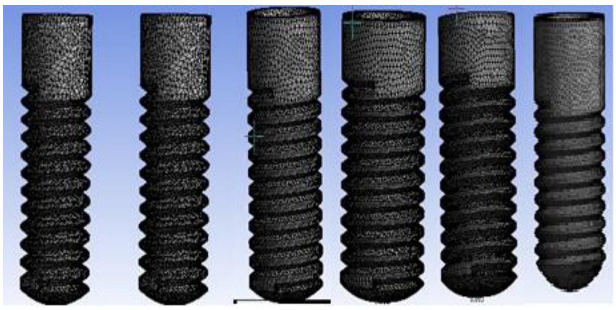

Fig.4 Meshing of M1,M2,M3,M4,M5,M6 Implant Diameter

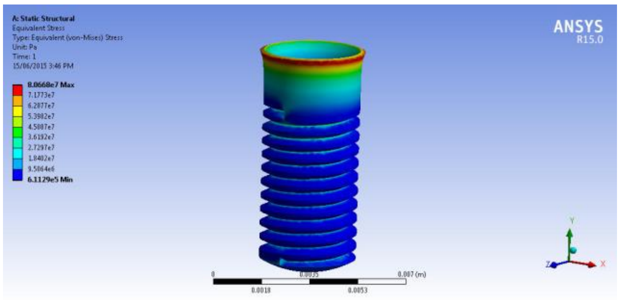

Fig.5. Equivant Stress Distribution for ϕ3.5mm.

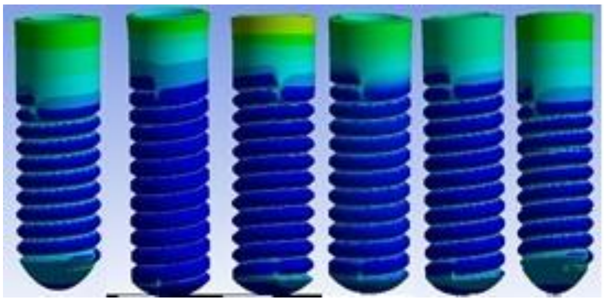

Fig.6 Equivalent Stress Ditribution of M1,M2,M3,M4,M5,M6 Implant Diameter

VI.CONCLUSION

Increase in the diameter increases the contact surface area between the implant and the bone which subsequently reduces the stress pattern thus enhancing the longevity of implant.

VII. ACKNOWLEDGEMENT

I would like to express my gratitude towards my Project Guide Prof.K.M.Narkar and Dr.D.R.Panchagade ,Head of Mechanical Engineering Department, D.Y.Patil College of Engineering, Akurdi , for their valuable input and support right through the project work and encouraged me in completing the dissertation work in scheduled time. I would also like to thank Prof.Dhatrak P.N and Prof. R.N.Ghorpade for their timely guidance and support.

REFERENCES

1. BobinSaluja, MasoodAlam, T Ravindranath, A Mubeen et al "Effect of length and diameter on stress distributionpattern of INDIDENT dental implants byfinite element analysis" , Journal of Dental Implants, Vol 2 , Issue 1, PP 19-25, 2012.
2. ZeinabArsalanloo, Reza Telchi, and Kambiz Ghaemi Osgouie, " Selection of Optimum Thread Type in Implants to Achieve Optimal Biomechanical Properties by Using 3D Finite Element Method" , International Journal of Bioscience, Biochemistry and Bioinformatics, Vol. 4, Issue 3, PP 185 – 190,May 2014
3. Zhihong Mao ,Nanning, " Design of dental implant system by FEA " 2014 IEEE Workshop on Electronics, Computer and Applications, 978-1-4799-4565-8, PP 675-677, 2014.
4. Giuseppe Vairo and GianpaoloSannino: "Comparative Evaluation of Osseointegrated Dental ImplantsBased on

Platform - Switching Concept : Influence of Diameter, Length, Thread Shape, and In-Bone Positioning Depth onStress-Based Performance ", Computational and Mathematical Methods in Medicine ,Volume 2013, Article ID 250929,PP 1-15,2013.
5. Ratnakar R. Ghorpade , Shailesh S. Yelekar, "Computational And Experimental Studies In Threaded Dental Implant Research ", e-Journal of Dentistry , Vol 3, Issue 4, PP 457 -465 , Oct - Dec 2013
6. PietroAusiello, Pasquale Franciosa , Massimo Martorelli , David C. Watts, " Effects of thread features in osseo-integrated titaniumimplants using a statistics-based finite element method " ,ELSEVIER, Dentalmaterials , Issue 28,PP 919–927(2012).
7. S.R Desai, MS Desai, G Katti, I Karthikeyan , "Evaluation Of Design Parameters Of Eight Dental Implant Designs: A Two-Dimensional Finite Element Analysis ", NJPC Vol. 15,Issue 2,PP 176-181, 2012
8. Babak Bahrami, Farzan Ghalichi, "Finite Element Analysis of Stress Distribution in Immediately Loaded Dental Implant " ,International conference of Biomedical Engineering,Tehran,Iran,21-22 Dec, 2012.
9. Krzysztof Szajek, MarcinWierszycki and Tomasz Łodygowski , "Reduction of the tooth-implant components dimensions by optimization procedure", CMM-2011 – Computer Methods in Mechanics, Vol.4,9–12 May 2011.
10. Mohamed I. El-Anwar , Mohamed M. El-Zawahry, "A Three Dimensional Finite Element Study On Dental Implant Design ",Journal of Genetic Engineering and Biotechnology, Issue 9, PP 77–82, 2011.
11. OğuzEraslan&Özgürİnan, "The effect of thread design on stress distribution in a solidscrew implant: a 3D finite element analysis ", Springer-Verlag, Clin Oral Invest,Issue 14,PP 411–416 , 2010.
12. Mansour Rismanchian, Reza Birang, Mahdi Shahmoradi, "Developing a New Dental Implant Design and Comparing its Biomechanical Features with Four Design" , Dental Research Journal ,Vol. 7, No. 2, PP 70 - 75 ,Summer -Autumn 2010).
13. Abuhussein H, Pagni G, Rebaudi A, Wang HL, "The effect of thread pattern upon implant osseo-integration ". Clinical Oral Impl.Res.21, 129-136,2010
14. Liang Kong,Kaijin Hu,Dehua Li,Yingliang Song, "Evaluation of Cylinder Implant Thread Height and Wdth – 3D Finite Element Analysis",The International Journal of Oral and Maxillofollical Implants,Volume 23,Number 1,PP 65-74,2008.
15. Luigi Baggi, Ilaria Cappelloni,Michele Di Girolamo, Franco Maceri, and Giuseppe Vairo, "The influence of implant diameter and length on stress distribution of osseointegrated implants related to crestal bone geometry: A three dimensional finite element analysis" , The Journal of Prosthetic Dentistry,Volume 100,Issue 6, pp 423-431,Dec 2008.

