Effect of Variation of Thread Parameters of Dental Implant on Stress Intensity on the Cancellous Bone

Janhavi Gharate[1], Rushikesh Hadole[2], Kunal Kshirsagar[3], Priyanka Choudhari[4]
[1,2,3,4] *Student, Mechanical Engineering, MIT Pune*

Abstract- **A dental implant (also known as an endosseous implant or fixture) is a surgical component that interfaces with the bone of the jaw or skull to support a dental prosthesis such as a crown, bridge, denture, facial prosthesis or to act as an orthodontic anchor. Success or failure of depends on the health of the person receiving the treatment, drugs which affect the chances of osseointegration, and the health of the tissues in the mouth. The amount of stress that will be put on the implant and fixture during normal function is also evaluated. Planning the position and number of implants is key to the long-term health of the prosthetic since biomechanical forces created during chewing can be significant.**

This paper focuses on study of impact of parameters like depth, pitch and thickness of the implant profile on stress intensity. The mechanical aspects of the implant are studied through analysis. The paper includes selection of implant, Finite element analysis to find stress intensity and representation of effect of parameters on the value of stress intensity using MINITAB software.

The aim is to study the effect of variation of various parameters on the stress intensity on the bones. This aim is to be achieved by varying the mechanical parameters (depth, pitch and thickness) of the dental implant and performing static stress FEA analysis. The effect is studied, and results are interpreted and analyzed by using MINITAB software.

Index Terms- Dental implant, cancellous bone, implant profile, FEA analysis.

I. INTRODUCTION

Over the past several decades, dental rehabilitation with implants has been widely accepted by dentists and patients because of its reliable functional and aesthetic results. So far, dental implant has turned out to be a great success in long-term clinical applications with a survival rate of over 90%. However, in the posterior mandible with poor bone quality, the survival rate of implants is much lower. It is difficult to estimate the optimal primary stability in the posterior mandible, which leads to high implant failure rates. The success of dental implant is related to the quality and quantity of jaw bones, implant design, implant surface texture, surgical procedures and so on. Among the implant designs, implant depth, thickness and pitch have been intensively studied and well accepted as key factors, since they directly influence the primary stability, placement, and removal torque values of dental implant.

The finite element method in the analysis of implant biomechanics provides many advantages over other methods in simulating the complexity of clinical situations. It can be used to predict stress distributions in jaw bones and displacements in implants. However, most of the previous finite element studies examined the effect of implant parameters discretely and independently. Therefore, the information about implant parameters was not accurate, and some important information might have lost. This study aimed to evaluate continuous and simultaneous variations of dental implant depth, thickness and pitch of the threads and to identify their relatively optimal combination of these parameters to minimize the stress intensity on the cancellous bone.

II. MODEL DESIGN

A posterior mandibular segment with a screwed dental implant and a superstructure was modeled on a personal computer with a 3D program (CATIA). An original dental implant profile was measured on a profile projector. Once the dimensions are measured the results were used to create a CAD model using CATIA.

A. Material properties

All materials used in the modelling were assumed to be isotropic, homogeneous, and linearly elastic. Table 1 shows the parameters of elastic properties.

Materials	Young Modulus E (GPa)	Poisson's Ratio
Cortical Bone	13.7	0.26
Cancellous Bone	1.37	0.31
Titanium	110	0.33

Table 1. Mechanical properties of bones and Implant material

B. Interface conditions

An osseointegrated implant was simulated with a screwed rough surface. During the simulation, a bond condition was set at its interface with the mandibular bone.

C. Elements and nodes

The models were meshed with quad and tria elements for 2D elements and 10-node tetrahedral elements. A refinement mesh was generated around the implant as shown in Fig. 4. On average, one model consisted of 260,000 elements and 370,000 nodes.

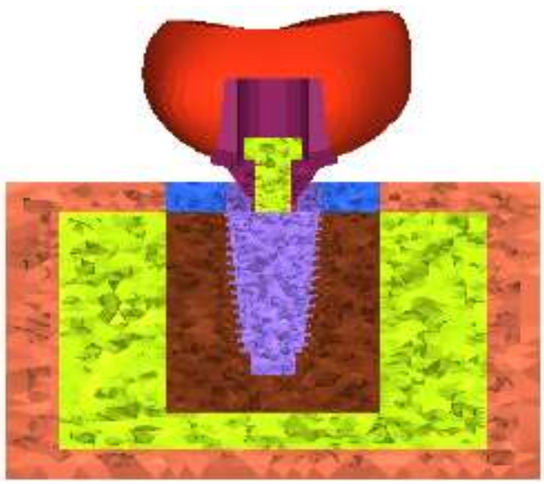

Figure 1. Meshed model of the assembly (Bones and implant)

D. Constraints and loads

The models were constrained at the nodes on the mesial and distal bones in all directions. Forces of 100N were applied axially (AX). The maximum stress intensity (maximum equivalent stress, abbreviated Max EQV stress) in cancellous bone was set as output variables to evaluate the effect of input variables on cancellous bone and implant. Sensitivities of output variables to input variables were also evaluated.

E. *Finite Element Analysis:*

Finite Element Analysis FEA is a computational tool for performing engineering analysis. It includes the use of mesh generation techniques for dividing a complex problem into small elements, as well as the use of software program coded with FEM algorithm. The FEA is used to calculate the deflection, stresses, strains, temperature, buckling behavior of the member. The basic idea of FEA is to make calculations at only limited (finite) number of points and interpolate the results for the entire domain (surface or volume). Any continuous object has infinite degrees of freedom and it's just not possible to solve the problem in this format. Finite element method reduces the degrees of freedom from infinite to finite with the help of discretization or meshing (nodes and elements).

Finite element analysis of implant model was done using Abaqus software.

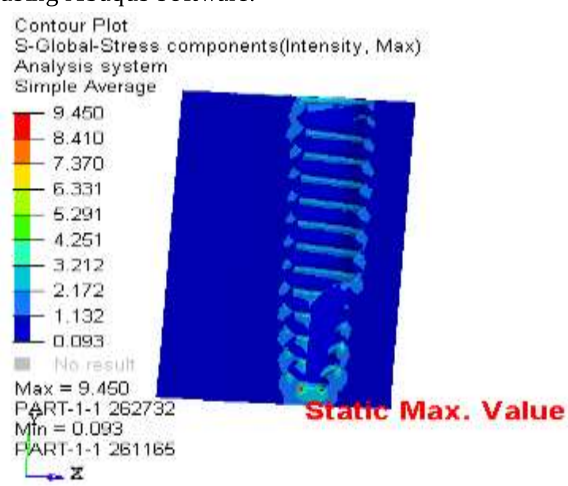

Figure 1. FEA model of the Cancellous Bone

III. L8 ARRAY DESIGN

The following design for 2 level and 3 factors was developed using MINITAB software. The CAD model of these iterations were developed and FEA analysis was carried out on the following, results of which are mentioned below.

Depth (mm)	Pitch (mm)	Thickness (mm)
0.21	0.85	0.07
0.21	0.85	0.19
0.41	1.20	0.07
0.41	1.20	0.19
0.21	1.20	0.07

0.21	1.20	0.19
0.41	0.85	0.07
0.41	0.85	0.19

Table 2. Two level Three factor L8 array Design

IV. RESULTS AND DISCUSSION

The below results are obtained after FEA analysis of all the above iterations.

Depth	Pitch	Thickness	Stress Intensity
0.21	0.85	0.07	7.27
0.21	0.85	0.19	2.34
0.41	1.20	0.07	15.69
0.41	1.20	0.19	15.68
0.21	1.20	0.07	7.29
0.21	1.20	0.19	7.27
0.41	0.85	0.07	15.66
0.41	0.85	0.19	15.69

Table 3. Stress intensity values obtained by FEA analysis

Graphical representations:

The following plots are obtained by statistical analysis on the MINITAB software.

The graphs to be obtained are:

1. Main effect plot for SN ratio
2. Main effect plot for Means

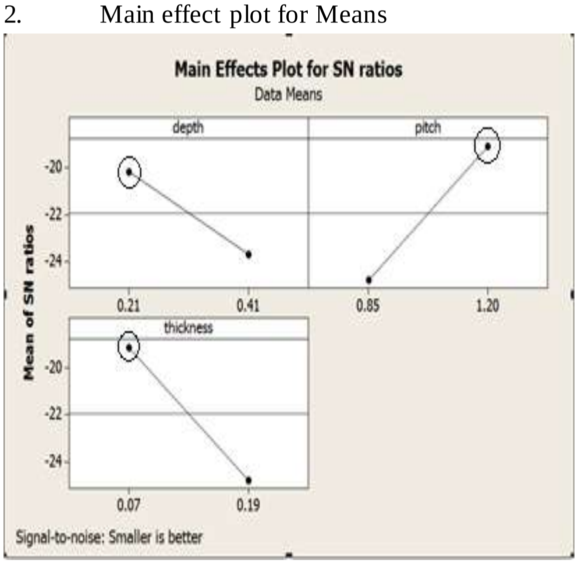

Figure 2. Main effect plot for SN ratio

Since the objective is to minimize stress intensity the function for SN ratio is "Smaller the Better". Hence lesser the value of SN ratio (magnitude) better are the results.

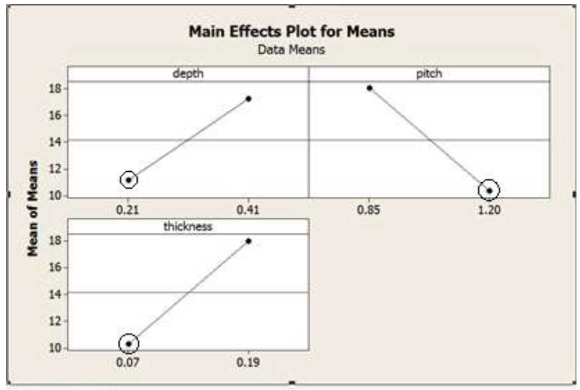

Figure 3. Main effect plot for Means

The objective is to minimize the stress intensity. The above graph shows the trend in the value of stress intensity when the parameters are varied.

The stress intensity on the cancellous bone will be minimum when we take following levels of the parameters:

- Depth: Low level (0.21mm)
- Pitch: High level (1.20mm)
- Thickness: Low level (0.07mm)

V. CONCLUSION

In the above study, the effect of parameters like Depth, Pitch and Thickness on stress obtained in cancellous bone were studied. Since the amount of stress intensity directly results in pain stimulation it is important to consider the individual effect of each parameter on the stress intensity. From the graphical representations we can conclude that, variation in depth, pitch and thickness of implant profile have considerable impact on the value of stress intensity. From the graphs, following trends are seen:

- Stress intensity increases with increase in depth and thickness.
- Stress intensity decreases with increase in pitch.

REFERENCES

[1] Dauglas Montgomery, Design and Analysis of Experiments 8th edition, 2012

[2] Abuhussein H, Pagni G, Rebaudi A, Wang HL. The effect of thread pattern upon implant osseointegration.

[3] Taguchi G, Konishi S, Taguchi Methods, orthogonal arrays and linear graphs, tools for quality American supplier institute, 1987.

[4] Hansson S, Werke M. The implant thread as a retention element in cortical bone: the effect of thread size and thread profile: a finite element study.

[5] Brunski JB. In vivo bone response to biomechanical loading at the bone/dental-implant interface.

[6] Eraslan O, Inan O. The effect of thread design on stress distribution in a solid screw implant: a 3D finite element analysis.

[7] Chun HJ, Cheong SY, Han JH, Heo SJ, Chung JP, Rhyu IC, Choi YC, Baik HK, Ku Y, Kim MH. Evaluation of design parameters of osseointegrated dental implants using finite element analysis.