To Evaluate the Influence of Implant Length on Stress Distribution of Osseointegrated Implant: A Three-Dimensional Finite Element Analysis: An In Vitro Study

Neha Jindal¹ Manjit Kumar¹ Shailesh Jain¹ Raman¹ Navjot Kaur¹

¹Department of Prosthodontics, Bhojia Dental College, Solan, Himachal Pradesh, India

Address for correspondence Manjit Kumar, MDS, Department of Prosthodontics, Bhojia Dental College, Baddi 173205, Distt. Solan, Himachal Pradesh, India (e-mail: manjitkiran@yahoo.co.in.com).

Dent J Adv Stud 2018;6:97–105

Abstract

Finite element analysis is a technique for obtaining a solution to a complex mechanical problem by dividing the problem domain into a collection of much smaller and simpler domains (elements) in which the field variables can be interpolated with the use of shape functions. An overall approximated solution to the original problem is determined based on variational principles. Finite element analysis can provide a nondestructive system for quantifying stresses generated at the various interfaces of similar or dissimilar material. The finite element method also allows the study of the internal state of stress of components as well as stress patterns in two or more dissimilar materials adjacent to each other without affecting their independent behavior. This method is therefore ideally suitable for the biomechanical analysis of orthopedic, cardiovascular, and dental structures. In this study, implants of different length were numerically analyzed using bone-implant models developed from computed tomography-generated images of the mandible with osseointegrated implants. The impact of various lengths on stress distribution was examined using implants with a length of 8, 10, and 13 mm in mandibular first molar region under axial load of 100 N and buccolingual load of 50 N. All materials were assumed to be linearly elastic and isotropic. The Statistical Package for the Social Sciences software package was used for statistical analysis. Maximum von Mises stresses were located around the implant neck. It was demonstrated that there was statistically nonsignificant decrease in von Mises stress as the implant length increased. Within the limitations of this study, there was statistically nonsignificant decrease in von Mises stress as the implant length increased.

Keywords

- ▶ dental implant
- ▶ finite element analysis study
- ▶ implant length
- ▶ masticatory load
- ▶ stress analysis
- ▶ von Mises stress

Introduction

It was possible to fix dental prostheses with intraosseous implants after the initiation of the concept of Osseo integration.¹ Success rate for implants in the mandible and maxilla have been reported as 95% and 90%, respectively, along with continued marginal bone loss over two decades. Marginal bone loss is seen to be due to poor oral hygiene and biomechanical factors.²

Dental implant transfers masticatory forces within jaws to the supporting osseous tissues. Many authors have emphasized the significance of various factors such as loading type, length and diameter of implants, topographic characteristics of implant, prosthesis type, and other biomechanical factors.³

Finite element analysis (FEA) divides the problem domain into a collection of smaller and simpler domains in which the field variables can be interpolated by using shape functions.

received
October 30, 2018
accepted after revision
November 30, 2018
published online
January 29, 2019

DOI <https://doi.org/10.1055/s-0038-1677646>
ISSN 2321-1482.

©2018 Bhojia Dental College and Hospital affiliated to Himachal Pradesh University

License terms

A comprehensive approximated solution to the problem is determined based on ever-changing principles.⁴

FEA helps measure stresses generated at the various interfaces of similar or dissimilar material. It also allows the study of the internal state of stress of components as well as stress patterns in two or more dissimilar materials adjacent to each other without affecting their independent behavior. This method is therefore ideally suitable for the biomechanical analysis of cardiovascular and dental structures.⁵

In this process, the structure of interest is divided into nodes and elements to create a mesh that is referred to as discretization, and computationally a system of simultaneous equations is generated to relate all the forces and displacement at nodes. Hence, the problem is converted into matrix algebra, and its solution is represented via displacements, strains, and stresses within a structure. The axi-symmetric techniques along with two- and three-dimensional (3D) models can be used although the 3D model can provide a more accurate prediction of stress distribution.⁶

Previously substantial extensive researches have been attempted on the characteristics of stress distribution in bone surrounding implants. However, FEA has not been extensively used to investigate the role of various implant dimensions on stress distribution in mandibular molar areas and biomechanical behavior of the mandibular cancellous bone when in contiguity to the cortical bone.⁷

The aim of this article is to evaluate the stress in peri-implant area of the cancellous bone by different lengths of implant in the molar region of the mandible, to understand the impact of implant length on stress distribution, and to understand the overload of clinically evidenced crestal bone loss at the implant neck in mandibular molar peri-implant region using 3D FEA.

Materials and Methods

Materials

- MIS implants (length 8, 10, 13 mm)
- Dried specimen of human partially edentulous mandible (with missing first molars) used for coordinate measuring machine scanning
- Finite element software ANSYS v. 18.1 (CADD Centre, Chandigarh, India)

Methodology

To accomplish this, solid models of tapered threaded dental implants with lengths of 8, 10, and 13 mm were developed and studied. These dimensions were selected on the basis of most routinely used implants. The finite element software ANSYS v. 18.1 was used to merge the bone-implant model and to generate and solve the discrete finite element meshes. Computed tomographic (CT) images of the human mandible were used to develop 3D models of the mandibular molar segment. In the model, implant was positioned in vertical direction in the molar region. The whole bone was considered to be a homogeneous, isotropic material with the character of cortical bone. Structure of cancellous bone is altered after osseointegration, and bone-implant interface resembles cortical bone.

The geometric models of implant and mandibular bone were generated (►Figs. 1, 2) using CMM (►Fig. 3) and then imported. The process of converting geometric model into finite element model called *meshing* was performed. The finite element model consisted of nodes and elements as shown (►Tables 1, 2).

The assembled finite element model of inserted implant in bone (►Fig. 4), then imported into ANSYS 18.1 (ANSYS 18.1, Inc., Canonburg, PA) software for analysis. The material properties of implant and bone were entered in the pre-processing stage (►Table 3).

The applied force and boundary conditions were applied (►Fig. 5). Postprocessing the results and capturing the von Mises stress and strain contours of each individual section in the system was done (►Figs. 6–9). Preprocessing, solving, and postprocessing are three separate stages in the ANSYS software. The force simulated was in the same direction as in occlusion that is in axial and buccolingual directions of 100 N and 50 N, respectively, which was loaded to the head of implant.

After application of force, the magnitude was analyzed by ANSYS Workbench (ANSYS 18.1, Inc., United States) a 3D FEA program. A color scale was used to assess quantitatively the stress distribution in the bone (cortical and cancellous) and implant interface. Red and blue indicate areas with the highest and lowest stress and strain, respectively (►Figs. 5–9).

Constraints and Loads

The models were constrained in all directions at the nodes on the distal-end surface of the bone segment. Forces of 100 N and 50 N were applied axially and buccolingually, respectively, to the occlusal node at the center of the implant. The maximum and minimum equivalent von Mises (EQV) stresses contour at cortical bone, cancellous bone, and implants were calculated.

Results

This study was aimed to assess the amount of stress/strain field developing at implant and bone interface with implant length of 8, 10, and 13 mm in mandibular first molar

Fig. 1 MIS implants (8, 10, 13 mm).

Fig. 2 Dried specimen of human partially edentulous mandible (with missing first molars).

Fig. 3 Coordinate measuring machine.

Table 1 Number of element and nodes generated for each model

Model	No. of elements	No. of nodes
Implant	9,756	17,706
Bone	80,651	145,973

region under axial and buccolingual loads of 100 N and 50 N, respectively. A color scale showing von Mises stress distribution values were used to assess quantitatively the stress distribution in the bone and implant interface (► **Figs. 6–9**). Red and blue indicate areas with highest and lowest stress, respectively.

Table 2 Number of elements and nodes used for model generation of different length of implant and bone

Model	Elements	Nodes
8-mm implant	4,283	7,760
10-mm implant	5,473	9,946
13-mm implant	6,663	12,132
Cortical bone	18,427	36,566
Cancellous bone	23,330	38,604

Fig. 4 Implant model inserted in the bone model.

Table 3 Physical properties of component of bone model

Material	Cortical bone	Cancellous bone
Elastic modulus (MPa)	14700	1470
Poisson's ratio	0.30	0.30

The maximum von Mises stress values obtained from scale in cortical and cancellous bones and implant interface were recorded and compared with the help of ► **Tables 4–8** and **Graphs 1–4**.

Further, it was concluded that when implant length was changed from 8, 10, and 13 mm in the mandibular first molar region with the help of ANSYS 18.1 software and data were analyzed statistically, the stresses with axial and buccolingual loads generated in all body model were within the tolerable limit whereas the effect of implant length was nonsignificant.

Statistical Methods

Repeated measures ANOVA (analysis of variance) test was used to statistically assess the impact of implant length on stress distribution of osseointegrated implant. The result was

Fig. 5 Finite element model and definition of load direction. **(A)** Axial load of 100 N. **(B)** Buccolingual load of 50 N.

Fig. 6 **(A)** An 8-mm implant model showing stress distribution on implant in cancellous bone when axial and buccolingual forces are loaded. **(B)** Cortical bone model showing stress distribution on cortical bone when axial and buccolingual forces are loaded on 8-mm implant. **(C)** Cancellous bone model showing stress distribution on cancellous bone when axial and buccolingual forces are applied on 8-mm implant.

confirmed by ANOVA for all the parameters. To determine whether variance in implant length had an effect on clinical parameters, stepwise ANOVA was used. In this study, it was accomplished that these stresses were within tolerable limit, and thus the effect of different implant length in the mandibular first molar region was nonsignificant ($p < 0.05$). The maximum von Mises stress values with axial force on implants were observed to be 5.18 MPa for 8-mm, 5.07 MPa for 10-mm, and 5.07 MPa for 13-mm implant. The maximum von Mises stress values with buccolingual force on implants were observed to be 3.78 MPa for 8-mm, 4.92 MPa for 10-mm and 4.34 MPa for 13-mm implant (► **Table 4, Graph 1**). The maximum von Mises strain values with axial force on implants were observed to be 1.26 mm/mm for 8-mm, 1.33 mm/mm for 10-mm, and 1.10 mm/mm for 13-mm implant. The maximum von Mises strain values

with buccolingual force on implants were observed to be 0.893 mm/mm for 8-mm, 0.985 mm/mm for 10-mm, and 0.921 mm/mm for 13-mm implant (► **Table 5, Graph 2**).

The maximum von Mises stress values with axial force on cortical bone were observed to be 1.33 MPa for 8-mm, 1.08 MPa for 10-mm, and 1.29 MPa for 13-mm implant. The maximum von Mises stress values with buccolingual force on cortical bone were observed to be 2.23 MPa for 8-mm, 1.84 MPa for 10-mm, and 2.18 MPa for 13-mm implant (► **Table 6, Graph 3**).

The maximum von Mises strain values with axial force on cortical bone were observed to be 0.576 mm/mm for 8-mm, 0.845 mm/mm for 10-mm, and 0.924 mm/mm for 13-mm implant. The maximum von Mises strain values with buccolingual force on cortical bone were observed to be 0.177 mm/mm for 8-mm, 0.218 mm/mm for 10-mm, and 0.192 mm/mm for 13-mm implant (► **Table 7, Graph 4**).

Fig. 7 (A) A 10-mm implant model showing stress distribution on implant in cancellous bone when axial and buccolingual forces are applied. (B) Cortical bone model showing stress distribution on cortical bone when axial and buccolingual forces are applied on 10-mm implant. (C) Cancellous bone model showing stress distribution on cancellous bone when axial and buccolingual forces are applied on 10-mm implant.

Fig. 8 (A) A 13-mm implant model showing stress distribution on implant in cancellous bone when axial and buccolingual forces are applied. (B) Cortical bone model showing stress distribution on cortical bone when axial and buccolingual forces are loaded on 13-mm implant. (C) Cancellous bone model showing stress distribution on cancellous bone when axial and buccolingual forces are loaded on 13-mm implant.

Discussion

Implant biomechanics is a significant factor in longevity of dental implants. The ability of implant to transfer physiologic loads within a range of 100 to 3,000 μm strains to the surrounding bone depends on implant biomechanics.⁸ Clinical studies have stated remarkable bone loss around the neck of failed implants, and different theories have been stated to interpret this bone reaction. Unfavorable loading has been seen to cause increased stress in bone around implant leading to bone loss, and

thus implant failure as demonstrated by various animal experiments⁹ and clinical studies. Thus, it is important to evaluate the stress/strain in bone and their relation to various parameters of implant and bone. This study used the finite element method to evaluate the effect of implant length on stress in bone and implant.

In a comparative analysis, complex reality can be simplified assuming that proportions and relative effect accurately reflect reality. Because simulation of the whole mandibular body is very magnificent, smaller models have been offered for parameter studies.¹⁰ Here in this analysis, a portion of

Fig. 9 (A) All body model showing stress distribution on implant bone interface when axial and buccolingual forces are loaded on an 8-mm implant. (B) All body model showing stress distribution on implant bone interface when axial and buccolingual forces are loaded on a 10-mm implant. (C) All body model showing stress distribution on implant bone interface when axial and buccolingual forces are loaded on a 13-mm implant.

Table 4 Descriptive von Mises stress values with axial (100 N) and buccolingual loads (50 N) on different length of implants inserted in bone (without the effect of cortical and cancellous bone)

Variable	Implant (mm)	Mean	Standard deviation	Standard error mean
Stress—axial	8	5.18	3.45	1.09
	10	5.07	3.37	1.06
	13	5.07	3.39	1.07
Stress—buccolingual	8	3.78	2.54	0.804
	10	4.92	3.30	1.04
	13	4.34	2.92	.924

Table 5 Descriptive von Mises stress values with axial (100 N) and buccolingual loads (50 N) on cortical bone with different lengths of implants (without cancellous bone)

Variable	Implant (mm)	Mean	Standard deviation	Standard error mean
Stress—axial	8	1.33	0.898	0.284
	10	1.08	0.730	0.230
	13	1.29	0.867	0.274
Stress—buccolingual	8	2.23	1.14	0.360
	10	1.84	1.01	0.319
	13	2.18	1.11	0.351

bone was modeled to simulate the posterior region of the mandible.

The shape of the cancellous bone was simplified to a rectangular block to facilitate the modeling of the same amount of bone for the three types of implants, while only the outer edges of the cortical bone were rounded.

In the studies of the whole mandible, in which convergence tests with mesh refinements have been done, models with more than 13,720 or 10,420 nodes showed convergent

results.¹¹ A fine mesh is important to accomplish a precise model. The main advantage of small, simplified models is that they allow for finer meshing with hexahedron elements. Thus in this study, only a mandibular segment with an extremely fine mesh around the implant (areas of high stress) was modeled. This resulted in a model consisting of 105,008 nodes, a number that was considered to ensure a sufficiently fine mesh for the given geometry, and no further mesh refinement was performed.

Table 6 Descriptive von Mises stress values with axial (100 N) and buccolingual loads (50 N) on cancellous bone with different lengths of implants (without cortical bone)

Variable	Implant (mm)	Mean	Standard deviation	Standard error mean
Stress—axial	8	19.74	13.25	4.19
	10	14.46	9.713	3.07
	13	20.7	13.9	4.40
Stress—buccolingual	8	1.28	8.61	2.72
	10	9.46	6.36	2.01
	13	1.11	7.49	2.36

Table 7 Descriptive von Mises stress values with axial (100 N) and buccolingual loads (50 N) on all body models with varying lengths of implants (with cortical and cancellous bone)

Variable	Implant (mm)	Mean	Standard deviation	Standard error mean
Stress—axial	8	1.14	351.1	111.0
	10	1.45	450.1	142.3
	13	1.36	430.18	133.1
Stress—buccolingual	8	5.42	7.59	2.40
	10	5.62	8.17	2.58
	13	.921	2.26	0.716

Table 8 Intergroup comparison of stress of all body models of all the groups

Variable	Group	Sum of Squares	Mean square	F	Sig.
Stress—axial	Between groups	9,615.716	4,807.858	0.026	0.9 ^a
	Within groups	5,043,109.079	186,781.818		
	Total	5,052,724.794			
Stress—buccolingual	Between groups	8.847	4.424	0.060	0.1 ^a
	Within groups	1,978.032	73.260		
	Total	1,986.879			

^aNon significant **Graph 1** Descriptive von Mises stress values with axial (100 N) and buccolingual loads (50 N) on different lengths of implants inserted in bone.

When only cancellous bone was considered in contact with implant-bone interface, without the contribution of cortical bone, the maximum EQV stress came out to be

19.74 MPa with 8-mm implant that further decreased to 14.46 MPa with 10-mm implant, but on further increasing the length to 13 mm, the stress increased further

Graph 2 Descriptive von Mises stress values with axial (100 N) and buccolingual loads (50 N) on cortical bone with different lengths of implants.

Graph 3 Descriptive von Mises stress values with axial (100 N) and buccolingual loads (50 N) on cancellous bone with different lengths of implants.

Graph 4 Descriptive von Mises stress values with axial (100 N) and buccolingual loads (50 N) on all body models with varying lengths of implants.

up to 20.7 MPa that is almost to the level of 8-mm-long implant. These findings are in contrast of what Baggi et al¹² observed. In his research, with increased implant length, stress gradients seemed to decrease at the cancellous peri-implant region.

The cancellous region withstood the axial load less efficiently, and the load was remarkably displaced downward. Thus, high or moderate stresses were found over large areas at the bone-implant interface, and highest stresses were located around the implant base. In these instances,

the presence of threads had a favorable influence on force transmission to bone resulting in dissipating force.

Under buccolingual load, in most instances, stress was concentrated in the cancellous bone around the implant neck, near the fulcrum of implant bending. For this reason, differences in implant length in the lower part of the implant did not influence the strain distribution.

When only the cortical bone was considered in contact with implant-bone interface, without the contribution of the cancellous bone, the maximum EQV stress came out to be 1.33 MPa with 8-mm implant that further decreased to 1.08 MPa with 10-mm implant, but on further increasing the length to 13 mm, the stress increased further up to 1.29 MPa that is almost to the level of 8-mm-long implant.

Because the cortical bone had a much higher elastic modulus than the cancellous bone, it was the main load-carrying member for all cancellous bone qualities, regardless of load direction. Higher stress concentration around the implant neck was also found in many 3D FEAs. Implant length did not greatly influence stress distribution in the cortical bone, probably because the implant neck had the same shape and diameter in all models and transferred the stress similarly to the surrounding bone.

Summary

Under axial load, maximum EQV stress on implant decreased from 8 to 10 mm and remained constant from 10 to 13 mm, and under buccolingual load, maximum EQV stress on implant increased from 8 to 10 mm and again decreased from 10 to 13 mm. Under axial load, maximum EQV strain on implant increased from 8 to 10 mm and again decreased from 10 to 13 mm, and under buccolingual load, maximum EQV strain on implant increased from 8 to 10 mm, but it again decreased from 10 to 13 mm.

Under axial load, maximum EQV stress on the cortical bone decreased from 8 to 10 mm and again increased from 10 to 13 mm, and under buccolingual load, maximum EQV stress on the cortical bone increased from 8 to 10 mm and again decreased from 10 to 13 mm. Under axial load, maximum EQV strain on the cortical bone increased as the length of the implant increased, and under buccolingual load, maximum EQV strain on implant increased from 8 to 10 mm, but it again decreased from 10 to 13 mm.

Under axial load, maximum EQV stress on the cancellous bone decreased from 8 to 10 mm and further increased from 10 to 13 mm, and under buccolingual load, maximum EQV stress on implant increased from 8 to 10 mm, but it again decreased from 10 to 13 mm. Under axial load, maximum EQV strain on implant increased from 8 to 10 mm and again decreased from 10 to 13 mm, and under buccolingual load, maximum EQV strain on implant increased from 8 to 10 mm, but it again decreased from 10 to 13 mm.

Under axial load, maximum EQV stress on all body model (cortical and cancellous) increased from 8 to 10 mm, but it again decreased from 10 to 13 mm, and under buccolingual load also, maximum EQV stress on all body model (cortical and cancellous) increased from 8 to 10 mm and again decreased from 10 to 13 mm.

Conclusion

Maximum EQV stress and strain in the cancellous and in cortical bones occurred around neck of implants, confirming the importance of crestal bone loss due to increased occlusal load and importance of presurgical diagnosis for implant long-term prognosis.

Under axial and buccolingual loads, the influence of implant length on bone stress and strain appeared to be nonsignificant.

Within limitations of this study, the length of implant has minimal clinical significance as far as stress and strain are concerned as compared with other factors.

There is still further scope for improving the clinical significance of this study by additional parameters that are diameter of implant, number of threads of implant, splinted implants, and comparison between different densities of bone.

Funding

None.

Conflict of Interest

None declared.

References

- 1 Tripathi R. Application of Finite Element Analysis in Dentistry. 3rd ed. PA, NJ: Prentice Hall; 2002:1–22
- 2 Petrie CS, Williams JL. Comparative evaluation of implant designs: influence of diameter, length, and taper on strains in the alveolar crest. A three-dimensional finite-element analysis. *Clin Oral Implants Res* 2005;16(4):486–494
- 3 Geng JP, Tan KBC, Liu GR. Application of finite element analysis in implant dentistry: a review of the literature. *J Prosthet Dent* 2001;85(6):585–598
- 4 Sevimay M, Turhan F, Kiliçarslan MA, Eskitascioglu G. Three-dimensional finite element analysis of the effect of different bone quality on stress distribution in an implant-supported crown. *J Prosthet Dent* 2005;93(3):227–234
- 5 Kong L, Hu K, Li D, et al. Evaluation of the cylinder implant thread height and width: a 3-dimensional finite element analysis. *Int J Oral Maxillofac Implants* 2008;23(1):65–74
- 6 Guan H, van Staden R, Loo YC, Johnson N, Ivanovski S, Meredith N. Influence of bone and dental implant parameters on stress distribution in the mandible: a finite element study. *Int J Oral Maxillofac Implants* 2009;24(5):866–876
- 7 Thirupathi RC. Introduction to Finite Elements in Engineering. 3rd ed. PA, NJ: Prentice Hall; 2005:1–15
- 8 Carter DR, Van Der Meulen MC, Beaupré GS. Mechanical factors in bone growth and development. *Bone* 1996;18 (1, Suppl):5S–10S
- 9 Isidor F. Loss of osseointegration caused by occlusal load of oral implants. A clinical and radiographic study in monkeys. *Clin Oral Implants Res* 1996;7(2):143–152
- 10 Mericske-Stern R. Overdentures with roots or implants for elderly patients: a comparison. *J Prosthet Dent* 1994;72(5):543–550
- 11 Hart RT, Hennebel VV, Thongpreda N, Van Buskirk WC, Anderson RC. Modeling the biomechanics of the mandible: a three-dimensional finite element study. *J Biomech* 1992;25(3):261–286
- 12 Baggi L, Cappelloni I, Di Girolamo M, Maceri F, Vairo G. The influence of implant diameter and length on stress distribution of osseointegrated implants related to crestal bone geometry: a three-dimensional finite element analysis. *J Prosthet Dent* 2008;100(6):422–431