J Indian Prosthodont Soc (July-Sept 2011) 11(3):165–171
DOI 10.1007/s13191-011-0095-4

ORIGINAL ARTICLE

Evaluation of Design Parameters of Dental Implant Shape, Diameter and Length on Stress Distribution: A Finite Element Analysis

M. Mohammed Ibrahim · C. Thulasingam · K. S. G. A. Nasser · V. Balaji · M. Rajakumar · P. Rupkumar

Received: 31 July 2010 / Accepted: 4 August 2011 / Published online: 20 August 2011

Abstract The aim was to evaluate the design parameters of dental implants shape, diameter and length on stress distribution by finite element analysis (FEA).The objectives of the study was to compare the influence of stress distribution in the implants of screw-vent tapered and parallel design by varying the implant diameter with a standard implant length. Six dental implant models have been simulated three-dimensionally. The influence of diameter and length on stress distribution was evaluated by *Group I*: for screw-vent tapered design (Zimmer Dental Implant Carlsbad, CA, USA) (1) Dental implant model with diameter 3.7 mm and length 13 mm. (2) Dental implant model with diameter 4.1 mm and length 13 mm. (3) Dental implant model with diameter 4.7 mm and length 13 mm. *Group II*: for parallel design (Zimmer Dental Implant Carlsbad, CA, USA) (4) Dental implant model with diameter 3.7 mm and length 13 mm. (5) Dental implant model with diameter 4.1 mm and length 13 mm. (6) Dental implant model with diameter 4.7 mm and length 13 mm. The 3-D model of the implant was created in the pro-e wildfire 4.0 software by giving various commands. This model was imported to the *ANSYS* software through *IGES* (initial graphic exchange specification) file for further analysis. All six models were loaded with a force of 17.1, 114.6 and 23.4 N in a lingual, an axial and disto-mesial direction respectively, simulating average masticatory force in a natural oblique direction, to analyze the stress distribution on these implants. The increase in implant diameter in Group I and Group II from 3.7 to 4.1 mm and from 4.1 to 4.7 mm with constant 13 mm length for screw-vent tapered and parallel design implant resulted in a reduction in maximum value of Von Mises stress in the bone surrounding the implant was statistically significant at 5% level done by student "*t*" test. The overall maximum value of Von Mises stress was decreased in parallel design implant diameter of 4.7 mm with constant length of 13 mm when compared to screw-vent tapered design implant samples. The results of the FEA computation depend on many individual factors including material properties, boundary conditions interface definition and also on the overall approach to the model. The results depicted that the tapered shape implant design exhibited higher stress levels in bone than the parallel shaped implant design which seemed to be distributing stresses more evenly. The application of a 3-D model simulation with the non-symmetric loading by the masticatory force on a dental implant resulted in a more satisfactory modeling of "clinical reality" than that achieved with 2-D models used in other studies.

Keywords Finite element analysis · Von Mises stress · Optical comparator

M. Mohammed Ibrahim (✉)
Department of Prosthodontics, Sri Venkateshwara Dental College & Hospital, OMR, Thalambur, Chennai, Tamilnadu 603103, India
e-mail: md_ibrahim_2003@yahoo.co.in

C. Thulasingam · K. S. G. A. Nasser · V. Balaji · M. Rajakumar · P. Rupkumar
Department of Prosthodontics, Tamilnadu Govt. Dental College & Hospital, Chennai, Tamilnadu 600003, India

Introduction

The direct bone-to-implant interface without intervening connective tissue was described as early as 1939 by Strock [1]. This direct contact of the implant to the bone is known as osseointegration. The concept of osseointegration was coined by Dr. Per Ingvar Branemark [2]. The success of a dental implant is the manner in which stresses are transferred to the surrounding bone. Load transfer from implants to surrounding bone depends on the type of loading, the bone to implant interface, the length and diameter of the implants, the shape and characteristics of the implant surface, the prosthesis type, and the quantity and quality of the surrounding bone.

From a bioengineering perspective, an important issue is to design the implant with a geometry that will minimize the peak bone stress caused by standard loading [3]. The complex geometry of the implants prevents the use of closed-form solutions in stress-analysis, where simple formulas relate the effect of external loads to internal stresses and deformations [3].

Finite element analysis (FEA) 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 or elements in which the field variables can be interpolated with the use of shape functions. Weinstein et al. [4] were the first to use FEA in Implant dentistry in 1976. FEA is an effective computational tool that has been adapted from the engineering arena to dental implant biomechanics [4].

The close apposition of bone to the titanium implant is the essential feature that allows a transmission of stress from the implant to the bone without any appreciable relative motion or abrasion. The absence of any intermediate fibrotic layer allows stress to be transmitted without any progressive change in the contact between the bone and implant. The titanium implant and the bone may be regarded as having a perfect fit with no stress in either material prior to loading. The stress distribution after loading the implant by average masticatory force was computed by FEA in this study.

The objectives of the study included the following:

(1) To evaluate the influence of dental implants of screw-vent tapered design with varying implant diameter and standard implant length on stress distribution.
(2) To evaluate the influence of dental implants of parallel design with varying implant diameter and standard length on stress distribution.
(3) To compare the influence of stress distribution in the implant of screw-vent tapered and parallel design by varying the implant diameter with a standard implant length.

Materials and Methods

The finite element method is a computer aided mathematic technique for obtaining accurate numerical solutions used to predict the response of physical systems that are subjected to external stress. Essentially any problem can be split up into a number of smaller problems with finite element method. This is done by considering that a complex geometrical shape is made up of a number of simpler shapes. With each simple shape being known as an "element" and the whole collection of elements being known as "mesh".

Within each element the relevant property of the material is predicted, each element is given life by inducing into them the properties of original material which it represents. Material properties such as young's modulus and Poisson's ratio can be utilized by computer generated analysis to describe the mechanical behavior, induced stresses, or the relationship between forces and displacements for a structural element. This is done without any reference to other elements in the mesh [5].

Type of stresses in finite element studies are generally described by means of direction (shear, tension, and compression) or by an effective absolute magnitude of principal stresses (equivalent stress of Von Mises). The "equivalent stress of Von Mises" is an expression that yields an effective absolute magnitude of stresses, taking into account principal stresses in three dimensions. The basic step for conducting this study can be divided into three phases.

(1) Pre processing and modeling
(2) Processing and meshing
(3) Post processing analysis.

The implant was first observed for dimensions and structural formation through the optical comparator. The magnification was set to 10× for better observation. The thread profile was drawn by using the points that was obtained from the optical comparator (Fig. 1). The 3-D model of the implant was created in the pro-e wildfire 4.0 software by giving various commands. This model was imported to the *ANSYS* software through *IGES* (initial graphic exchange specification) file for further analysis. All the six implants of various dimensions mentioned above were observed through the optical comparator and were modeled (Figs. 2, 3) and imported (Fig. 4) in the same way as described above.

Material Property of Constituent Materials

FEA assumes the following mechanical properties of the materials comprising the structure.

Implant Properties

The selected 3-D implant model represented commonly available submerged titanium [elastic modulus (e) = 1.1 ×

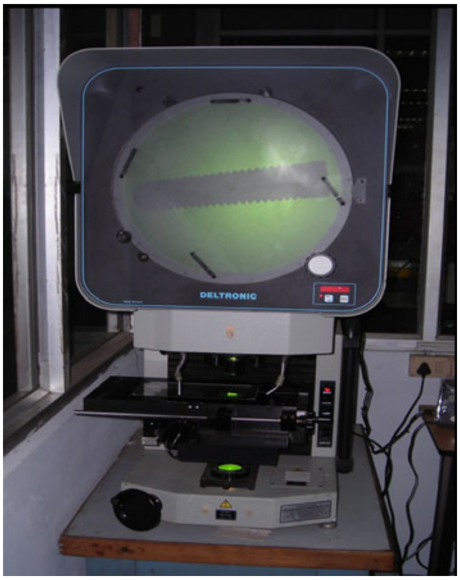

Fig. 1 Optical comparator

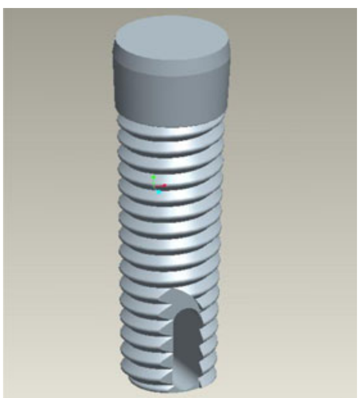

Fig. 2 Screw-vent tapered design dental implant sample A—diameter 3.7 mm, length 13 mm

10^5 MPa, Poisson's ratio (u) = 0.32] tapered and parallel shaped dental implants [5].

Bone Properties

The entire volume of bone was considered to be a homogeneous, isotropic material with the character of cortical bone [elastic modulus (e) 1.37×10^4 MPa, Poisson's ratio (u) = 0.3]. The shape of the bone was simplified to a prism having a quadrangular base and walls of an irregular octagon. The interface between the implant and the bone was assumed to be an immovable junction. For this a "fixed contact" option in the software was chosen [5].

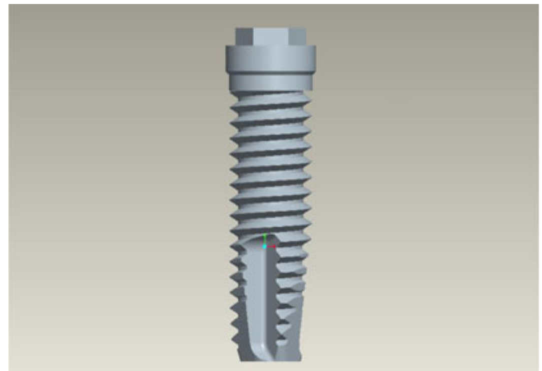

Fig. 3 Parallel design dental implant Sample D—diameter 3.7 mm, length 13 mm

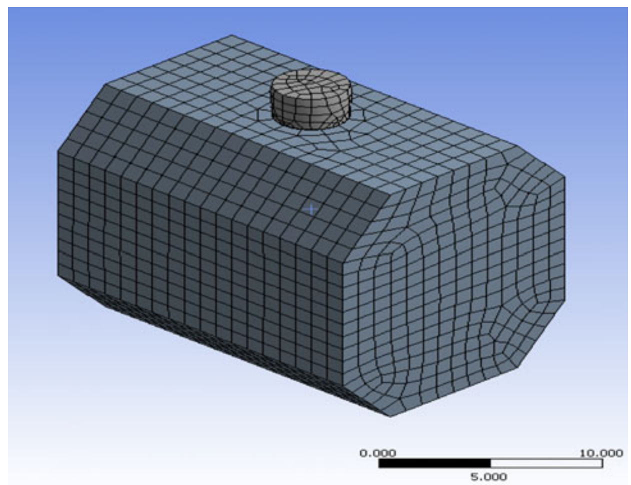

Fig. 4 Finite element model of dental implant with bone containing nodes and elements. Sample A—diameter 3.7 mm, Length 13 mm

Loading to Which the Implant Model is to be Subjected

Loading of the implant, three dimensionally was to be done with forces of 17.1, 114.6, and 23.4 N, in a lingual, an axial and disto-mesial direction respectively, simulated average masticatory force in a natural oblique direction (Fig. 5). The load applied to the implant was static type of loading [5].

Element Type

The ten-node tetrahedral type of element was selected the element size was 1 mm. The models consisted of 15,000–20,000 elements and 17,000–22,000 nodes depending on the implant size [5].

Stress distribution in the FE model comes in numerical values and in color coding. Maximum value of Von Mises

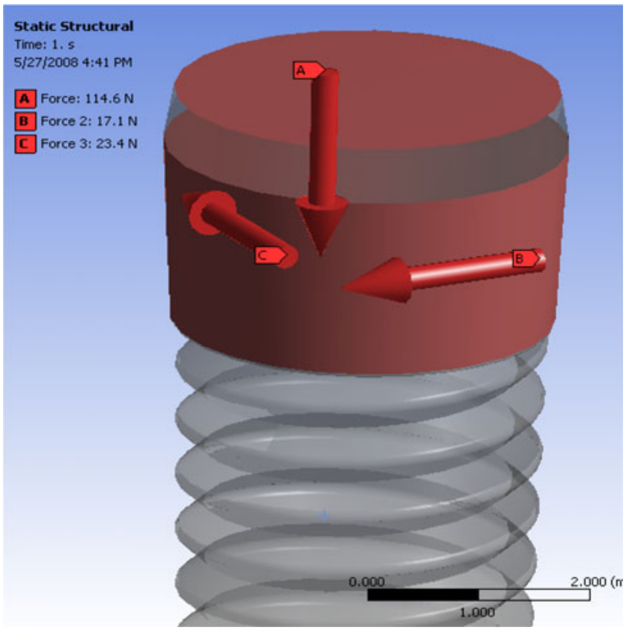

Fig. 5 Different types of structural forces applied to the dental implant. *A* indicates force acting in axial direction = 114.6 N. *B* indicates force acting in lingual direction = 17.1 N. *C* indicates force acting in disto-mesial direction = 23.4 N

stress = is denoted by red color. Minimum value of Von Mises stress = is denoted by blue color. The in-between values are represented by bluish green, green, greenish yellow and yellowish red in the ascending order of stress distribution.

Working Steps in Post Processing

This consists of:

(1) Analysis
(2) Interpretation of results both numerically and by color-coding

The Von Mises equivalent stress (MPa) at the implant–bone interface was computed using FEA software. All computations were performed on all the six 3-D implant models mentioned above and the values of maximum Von Mises equivalent stress on the implant and the bone was obtained (Figs. 6, 7).All the values of Von Mises equivalent stress on the implant and the bone obtained during this study were tabulated and analyzed for computation of the results.

Results

The maximum value of Von Mises stress in mega Pascal calculated in the bone surrounding the implant of commercially available marketed implants. Then, the results are analyzed using the following statistical analysis.

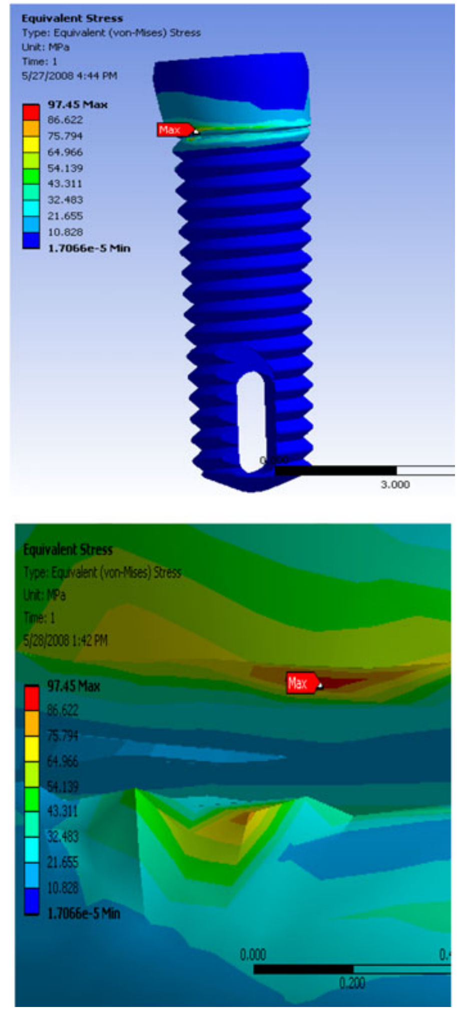

Fig. 6 Equivalent stress developed on dental implant after analyzing in Sample A. Maximum stress occurring region on dental implant Sample A

Table 5 shows the statistical evaluation of Student '*t*' test used to assess the significant difference between two group of implants. Tables 1 and 2 shows the number of elements and nodes used in the finite element models for screw vent tapered and parallel design respectively. Tables 3 and 4 shows the maximum value of Von Mises stress calculated in the bone surrounding the implant for screw-vent tapered and parallel design after loading the implant by simulated average masticatory forces in a natural, oblique direction. The basic data obtained after FEA for this study is presented in Tables 3 and 4.

Inference from Table 3

While loading the implants for screw-vent tapered design with constant implant length of 13 mm, the maximum value of Von Mises stress were calculated in the bone

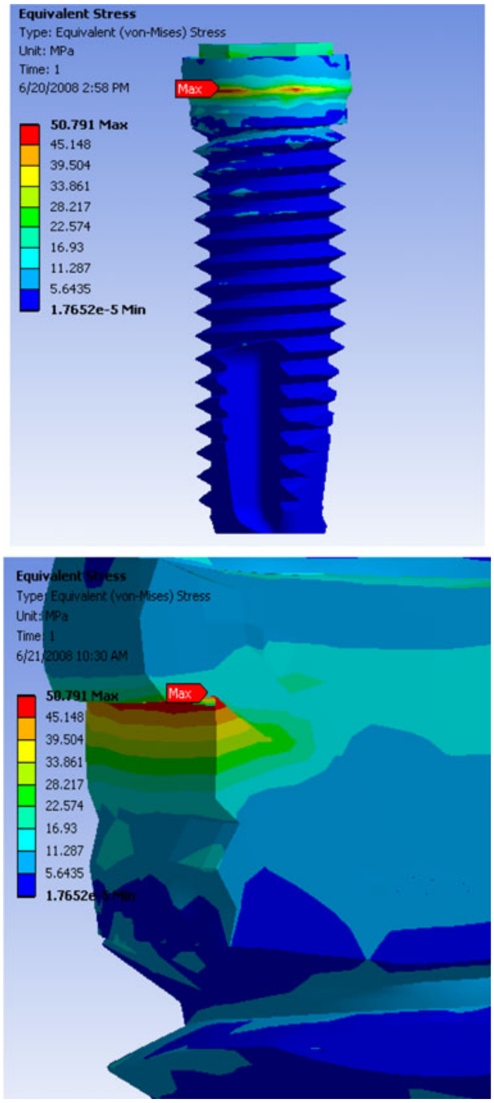

Fig. 7 Equivalent stress developed on dental implant after analyzing in Sample D. Maximum stress occurring region on dental implant Sample D

surrounding each implant of Group I was found to be decreasing as the implant diameter was increased from 3.7 to 4.1 mm and from 4.1 to 4.7 mm (Fig. 8).

Inference from Table 4

While loading the implants for parallel design with constant implant length of 13 mm, the maximum value of Von Mises stress were calculated in the bone surrounding each implant of group II was found to be decreasing as the implant diameter was increased from 3.7 to 4.1 mm and from 4.1 to 4.7 mm (Fig. 9).On comparing Tables 3 and 4, the overall maximum value of Von Mises stress is decreased in Sample F of parallel design implant diameter 4.7 mm with 13 mm length when compared to screw-vent tapered design implant samples.

Statistical Analysis

See Table 5.

Discussion

Since in clinical practice, the most frequently used implants are the screw and cylinder types, the stress/strain in bone around these implants were compared with parallel type of implant in FEA study. Currently available implants vary in diameter from 3 to 7 mm. The requirements of implant diameter are based on both surgical and prosthetic requirements. Finite element studies suggest on implant with a wider diameter is more favorable in reducing the stress distribution in bone surrounding the implants [5].

Dental implants are available in lengths ranging from 6.0 to 20 mm the commonly used implant length ranges from 8 to 15 mm, which correspond closely to normal root length. As the length of the surface area increases, it has been suggested that the stress levels for a given applied load is reduced on longer implants because of greater surface area. This also improves the mechanical resistance to masticatory forces. The implant length depends entirely upon the amount of available bones [6].

Finite element studies suggest that an implant in the form of threaded shape is able to transmit axial tensile or compressive loads better than the smooth type implant [6]. In a study done previously, the screw shaped implants provided the greatest retention immediately after placement and maximize the potential area for osseointegration and provide good initial stability.

Hence, the loading of the implants, in 3-D, with forces of 17.1, 114.6 and 23.4 N in a lingual, an axial and a disto-mesial direction respectively, simulating average masticatory force in a natural, oblique direction. The force magnitude as well as the acting point was chosen based on the work on Merickse-Stern. Therefore, static type of loading was applied in this study [5].

An increase in the implant diameter decrease the maximum value of Von Mises equivalent stress occurs as a result of a more favorable distribution of the simulated average masticatory forces applied in this study [5]. Moreover, it was concluded that the stress distribution in jaw bone was more effective as the surface area transmitting a horizontal component of force applied to dental implant increased.

From a biomechanical standpoint, the use of wider diameter implants allows engagement of a maximal amount of bone, and improved distribution of stress in the

Table 1 The number of elements and nodes used in the finite element models for screw vent tapered design

Implant samples	Nodes	Elements
A	48212	16016
B	51369	17262
C	43705	16269

Table 2 The number of elements and nodes used in the finite element models for parallel design

Implant samples	Nodes	Elements
D	54046	18137
E	57610	18272
F	55290	17912

Table 3 The maximum value of Von Mises stress calculated in each implant for the given boundary conditions for screw-vent tapered design (Group-I)

Implant samples	Diameter (mm)	Length (mm)	Von Mises stress (MPa)
A	3.7	13	97.450
B	4.1	13	79.111
C	4.7	13	64.917

Table 4 The maximum value of Von Mises stress calculated in each implant for the given boundary conditions for parallel design (Group-II)

Implant samples	Diameter (mm)	Length (mm)	Von Mises stress (MPa)
D	3.7	13	50.791
E	4.1	13	49.915
F	4.7	13	36.806

surrounding bone. The use of wider components also allows for the application of higher torque in the placement of prosthetic components. The use of wide implants, however, is limited by the width of the residual ridge and esthetic requirements for a natural emergence profile [6].

The known advantages of using wide-diameter implants include providing more bone to implant contact, bicortical engagement, and immediate placement in failure sites and reduction in abutment stresses and strain. Therefore, more contact area provides increased initial stability and reduces the stresses. Improved implant strength and resistance to fracture can be attained by increasing the diameter of implant [6].

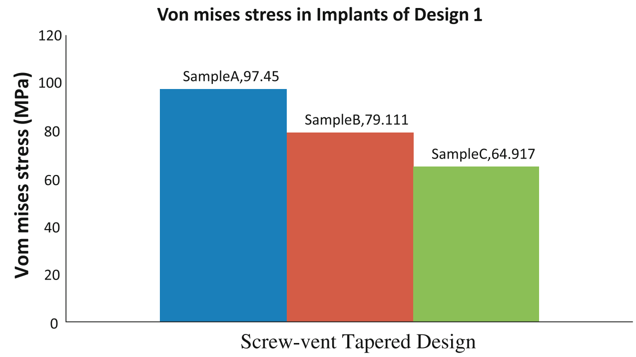

Fig. 8 Bar diagram Von Mises stress in Group I implants

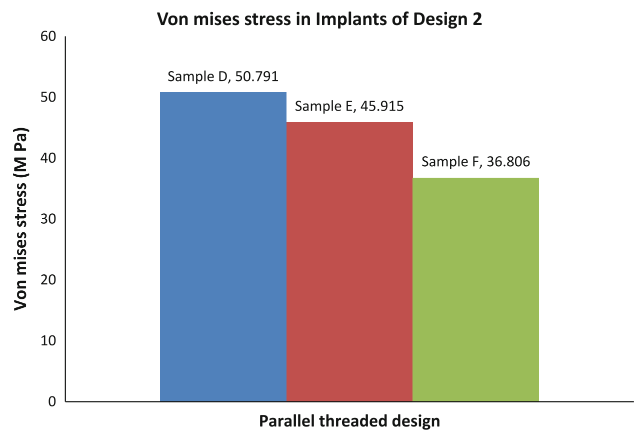

Fig. 9 Bar diagram Von Mises stress in Group II implants

When applying FEA to dental implants, it is important to consider not only axial load and horizontal loads/forces (moment-causing loads) but also a combined load (oblique occlusal force), because the latter represents more realistic occlusal directions and for a given force, will result in localized stress in cortical bone [3].

Rieger et al. [7] concluded that an endosseous implant with a high elastic modulus would be most suitable for dental implantology. Holmgren et al. [4] reported that a screw shaped implant design is most desirable from the stand point of stress distribution to surrounding bone. Also using FEA, Mailath et al. [4] compared cylindrical and conical implant shapes exposed to physiologic stresses and examined the occurrence of stress concentrations at the site of implant entry into bone. Patra et al. [4] finally reported that the tapered shape implant design exhibited higher stress levels in bone than the parallel shaped implant design which seemed to be distributing stresses more evenly. Factors such as inhomogeneous, non-linear, anisotropic properties of bone and the presence of a dynamic interface between the implant and bone were not taken into consideration in this study.

Table 5 Student '*t*' test

Design	Number of implant samples	Mean	SD	*P* value
Screw-vent tapered design	3	80.4927	16.310	0.029*
Parallel design	3	45.8373	7.834	0.048*

* Denotes significant at 5% level

Conclusion

Dental implants of screw-vent tapered and parallel design of three different diameter with constant implant length were used to evaluate the stress distribution in the bone surrounding the implant.

Within the limitations of this study, the findings of the present study support the following conclusion.

The increase in implant diameter in Group I and Group II from 3.7 to 4.1 mm and from 4.1 to 4.7 mm with constant 13 mm length for screw-vent tapered and parallel design implant resulted in a reduction in maximum value of Von Mises stress in the bone surrounding the implant was statistically significant at 5% level done by student "*t*" test. The overall maximum value of Von Mises stress was decreased in parallel design implant diameter of 4.7 mm with constant length of 13 mm when compared to screw-vent tapered design implant samples. The efficacy of this study under varied clinical condition also needs to be studied to enhance the results of this study.

References

1. Akagawa Y, Sato Y, Teixeira ER, Shindoi N, Wadamoto M (2003) A mimic osseointegrated implant model for three-dimensional finite element analysis. J Oral Rehabil 30:41–45
2. Branemark PI (1983) Osseointegration and its experimental background. J Prosthet Dent 50:399–410
3. Bozkaya D, Muftu S (2003) Mechanics of the tapered interference fit in dental implants. J Biomech 36(11):1649–1658
4. Geng JP, Tan KBC, Liu GR (2001) Application of Finite Element Analysis in implant dentistry: a review of literature. J Prosthet Dent 85:585–598
5. Himmlova L, Dostalova T, Kacovsky A, Konvickova S (2004) Influence of implant length and diameter on stress distribution: a Finite Element Analysis. J Prosthet Dent 91:20–25
6. Hoon LJ, Frias V, Woo LK, Wright RF (2005) Effect of implant size and shape on implant success rates: a literature review. J Prosthet Dent 94:377–381
7. Rieger MR, Fareed K, Adams WK, Tanquist RA (1989) Bone stress distribution for three endosseous implants. J Prosthet Dent 61(2):223–228