International Journal of Analytical, Experimental and Finite Element Analysis
Vol. 7, Issue. 3, Aug 2020, pp 68 - 76
https://doi.org/10.26706/ijaefea.3.7.20200804

Evaluation of The Stress Distribution on Four Different Peri Implant Bone Types When Loaded with Three Different Implant Lengths Subjected to Vertical and Oblique Forces in the Mandible: A Three-Dimensional Finite Element Analysis

Dr. Pankaj. C. Londhe[1]
pankajlondhe28@gmail.com

Dr. Ashvini Padhye[2]
drashvini16@gmail.com

[1]Post Graduate Student, MGM Dental College and Hospital, Department of Periodontology, Navi Mumbai, India

[2]HOD and Professor, MGM Dental College and Hospital, Department of Periodontology, Navi Mumbai, India

Abstract— The purpose of this study was to evaluate the stress distribution on four different peri implant bone types when loaded with three different implant lengths subjected to vertical and oblique forces in the mandible. 12 three-dimensional finite element models of the edentulous mandible simulating Type 1, Type 2, Type 3 and Type 4 bone quality according to the classification system of Lekholm and Zarb were created from a computerized tomography image by using the Hypermesh 13.0. Software program, which were loaded with three different implant lengths of 8mm, 11.5mm and 13mm (with titanium abutment and screw retained zirconia crown) modelled in the first molar region. A total force of 300 N was applied in the locations of the central fossa (300 N) in a vertical direction and a total oblique force of 300 N with a 30-degree angle was applied in the locations of the mesiobuccal cusp (150 N) and the distobuccal cusp (150 N) of the first molar. Lesser stresses were created in the peri implant bone by shorter implant lengths in all the four bone types as compared to longer implant lengths when loaded in vertical direction as opposed to on oblique direction.

Keywords— bone-implant interface, cortical bone, mandible

I. Introduction

Understanding of implant biomechanics in depth is important to enhance treatment modalities and avert any possible functional adversities and thereby failures. Bone availability has been a crucial factor in implant restorative dentistry as it dictates the external structure or volume of the edentulous area.

Factors such as type of implant surface, bone quality affects the degree of osseointegration and adaptive responses to changes in bone metabolism and transmission of masticatory forces. Stress values and stress distributions are influenced by the various bone qualities of the implant supported bone.

Van Oosterwyck et al [1], in his study had discussed that the primary stability and survival rate of implants is affected by the quality of peri-implant bone which is found to be lowest in the posterior maxilla, which he had

Research Article
Published online – 20 Aug 2020

Cite this article – Dr. Pankaj. C. Londhe and Dr. Ashvini Padhye, “Evaluation of The Stress Distribution on Four Different Peri Implant Bone Types When Loaded with Three Different Implant Lengths Subjected to Vertical and Oblique Forces in the Mandible: A Three-Dimensional Finite Element Analysis”, *International Journal of Analytical, Experimental and Finite Element Analysis*, RAME Publishers, vol. 7, issue 3, pp. 68-76, 2020.
https://doi.org/10.26706/ijaefea.3.7.20200804

evaluated using numerical simulations. Using the same method, he had also concluded that bone implant interface, elastic properties of bone, fixations of implants and lamina dura is modified by different patterns of bone loading.

According to classification system proposed by Lekholm and Zarb [2], he classified the bone qualities into four types: -

a) In Type 1 bone quality, the entire jaw comprises of homogenous compact bone.

b) In Type 2 bone quality, a thick layer (2 mm) of compact bone surrounds a core of dense trabecular bone.

c) In Type 3 bone quality, a thin layer (1 mm) of cortical bone surrounds a core of dense trabecular bone of favorable strength.

d) In Type 4 bone quality, a thin layer (1 mm) of cortical bone surrounds a core of low-density trabecular bone.

Biomechanics of oral implants is strongly influenced by the surrounding bone. Enough literature is available on the effects of ridge diameters, implant designs, platform designs and direction of load applied to an implant on stress and strain patterns in peri-implant bone. However, published data on stress patterns around implants of different lengths on surrounding bone are few. Whenever an implant is loaded, the load is transferred to the abutment, which carries it to the fixture and through the implant abutment and implant bone connection to the surrounding peri-implant bone [3]. Therefore, in the present study we aim to examine the role of implant height on stress and strain distribution patterns within the implant system and surrounding peri-implant bone.

In multiple implant restorations, in order to achieve the best stress distribution, implants should be parallel to each other and to the adjacent natural teeth. For a better surgical outcome, achieving parallelism of implants is crucial. But such condition does not always exist. In these situations, implants may be inserted at angulations with each other or the adjacent teeth. So, clinicians have to use angulated abutments to compensate for implant angulation [4-14].

Implant stability and peri-implant stress and strain distribution is greatly influenced by the occlusal loading patterns. Accentuated loading angles increase the peri-implant stress and strain patterns [5].

Finite element analysis (FEA), initially developed in the late 60’s has made significant contributions in excelling engineering intelligence in dentistry to tackle structural problems and aiding in better understanding of implant biomechanics [6].

Three dimensional (3D) FEA has been widely used for the quantitative evaluation of stresses on the implant and its surrounding bone [7].

In this study, 3D FEA models were developed and the stress distribution on 4 different peri-implant bone types was visualized, evaluated and compared for combinations of different lengths of implant when subjected to vertical and oblique forces.

II. Materials And Methods

Distributions of stress values in the cortical and trabecular bone and on the implants were evaluated in the present study. Displacement under force was analyzed at three different implant lengths that were placed in four different bone types in the edentulous mandible. The 3-Dimensional tetrahedral structural solid finite elements were used to model the bone, implant abutment framework, and crown structure material. 4 Mandibular bone 3D FEA models with crestal bone height of 13 mm and buccolingual dimension of 7 mm were created, simulating Type 1,2,3 and 4 bone, according to the classification system of Lekholm and Zarb [2] as described earlier. Trabecular bone was simulated as a zone of solid structure in cortical bone. 3D FEA models of endosseous implant system (Nobel Replace® CC) were created of fixed diameter of 4.3mm and lengths of 8, 11.5 and 13mm for this study. The shoulder of the implant system was 4.3 mm, and the implant neck height was 2.8 mm. The thickness of the zirconia crown used in this study was 0.7mm. Geometry of all the implant models was simulated as described by Wheeler [8] [Fig. 1].

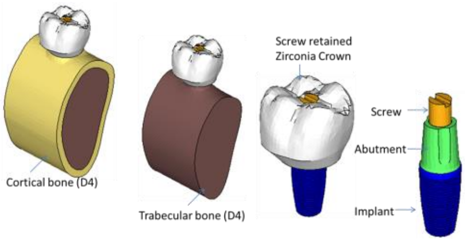

Fig. 1: Finite element model design

All the materials were presumed to be homogenous, isotropic and linearly elastic [9]. Elastic properties such as Young's modulus [E] and Poisson's ratio [μ] were determined from the literature [15-26] [Table I]. Elements and node numbers for the models were simulated. A fixed union between the bone and the implant along the interface was simulated. In total, the model consisted of 33,175 nodes and 175,776 elements [Fig. 1].

TABLE I

YOUNG'S MODULUS (E) AND POISSON'S RATIO (μ)

Material	Young's Modulus (MPa)	Poisson's Ratio
Cortical Bone	13400	0.30
Medullary Bone (D1, D2, D3)	1370	0.30
Medullary Bone (D4)	1100	0.30
Titanium (Implant, abutment)	110000	0.35
Zirconia	205000	0.30

Care was taken that there were no empty spaces within the implant-abutment and abutment-cylinder connections, or frictional coefficient, assuming a perfect fit situation among the implants, bone and prosthetic components.

Total vertical force of 300 N was applied on the central fossa of the simulated mandibular molar model. The total oblique force of 300 N was applied from the inner inclines of the mesiobuccal (150 N) and the distobuccal (150 N) cusps to simulate the mean off-axis interval in the clinical situation [Fig. 2 and 3]. The applied forces were assumed to be static.

Finite element analysis was done with following steps:

A. Designing of the 3D models

B. Assigning necessary material and behavioral properties

C. Generating the Finite Element Mesh

D. Loading of the models and Boundary conditions

E. Analyzing stage

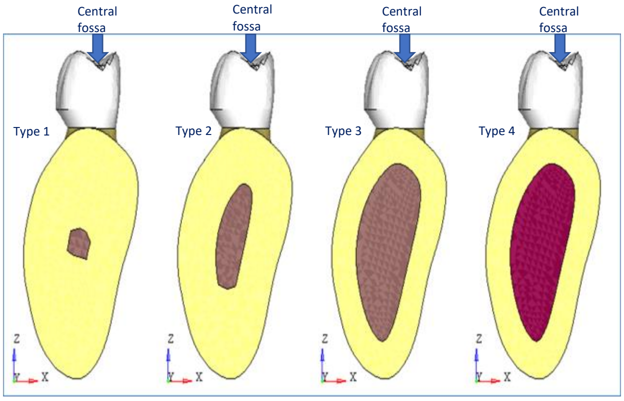

Fig. 2: Forces applied on the central fossa in vertical direction

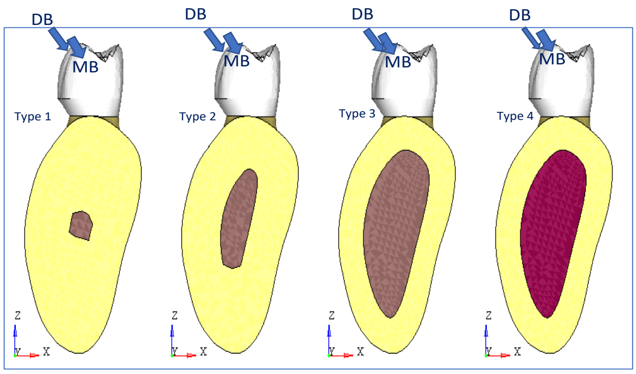

Fig. 3: Forces applied on the mesiobuccal and distobuccal cusps in oblique direction

A. Designing of the 3D models

Scanning was done for the implants (Nobel Replace® Tapered CC 4.3mm diameter and 8,11.5mm and 13mm length), titanium abutment and screw retained zirconia crown. Using a 3D modelling software (ANSYS 12.1 software), 3D FEA models of edentulous mandible, Type 1,2,3 and 4 bone quality and implant models with prosthetic restorations were generated.

B. Assigning necessary material and behavioral properties

Necessary material properties like 'Young's modulus of elasticity' and 'Poisson's ratio' were assigned to all model components after importing the designed models to ANSYS 12.1 software.

C. Generating the Finite Element Mesh

In this step, numerous elements and nodes were used to do meshing of the models.

• Elements: A complex geometric domain is represented as a collection of simple geometric subdomains called 'elements.

• Nodes: The points of connection between elements are called as 'nodes.

• Mesh: The collection of elements is called as 'Finite Element Mesh'.

D. Loading of the models and Boundary conditions

Allowing for the possibility of rational movements, vertical and oblique loads were applied. Occlusal loading of 300N magnitude was done in the vertical as well as oblique (30 degrees) directions along the long axis of the tooth.

Boundary conditions: To prevent a solid from moving in space like a rigid body on application of external loads, constrains are applied. These constrains are known as 'boundary conditions.

E. Analyzing stage

The solver, which is a part of ANSYS 12.1 software, was used to solve the problem equation of stresses in the bone, post load application on the models.

Mathematical models along with boundary conditions and loading were prepared with finite element software. The values were rendered to the ANSYS 12.1 (MSC Software) program to display stress values and their distributions. Stress data was conceived numerically and color coded.

In studies [27-30], the maximum (tensile) and minimum (compressive) principal stresses have been reported to be an adequate criterion to evaluate the occurrence of bone resorption in full osseointegration conditions, and the threshold ranges are 100 to 130 and 170 to 190 MPa, respectively. Von Mises stress values were defined as the beginning of deformation for ductile materials such as implants.

Elastic strain values were evaluated, as these values revealed the amount of deformation resulting from tensile and compressive stresses.

III. Results And Discussion

Stress analysis produced numerical values which had mathematical calculations without variance, thereby not requiring statistical analysis.

All designs showed similar distribution of Von mises stresses for all the implants. All the values obtained were far from the breaking point of titanium. Primarily, the concentration of Von mises stresses was seen on both the buccal and lingual aspects of the bone implant interface. The stress distribution on the Type 1,2,3 and 4 bone loaded with 8, 11.5 and 13mm implant and the abutment were evaluated [Fig. 4].

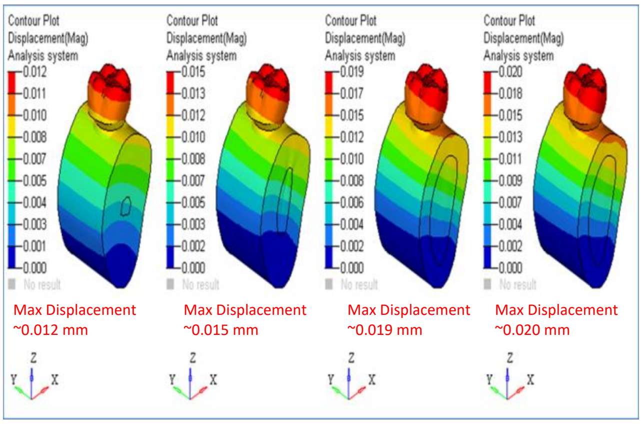

Fig. 4: Stresses generated when forces are applied in vertical direction on Implants

The maximum displacement of crown of 0.020mm was seen with implant length of 8mm in Type 4 bone whereas minimum displacement of 0.011mm was seen with implant length of 11.5mm in Type 1 bone when subjected to vertical loading conditions [Fig. 5]. As the implant length increased, the stress zone spread and the amount of displacement gradually decreased, resulting in the crown with 13mm implants showing least displacement. On the contrary on oblique loading, maximum displacement of 0.112mm was seen in implant length of 8mm in Type 4 bone and minimum displacement of 0.06mm was seen in implant length of 11.5mm in Type 1 bone [Fig. 6]. As compared to displacement of crowns seen in 8mm and 13mm implant lengths, in all four types of bone, the values were significantly lower in 11.5mm implant length.

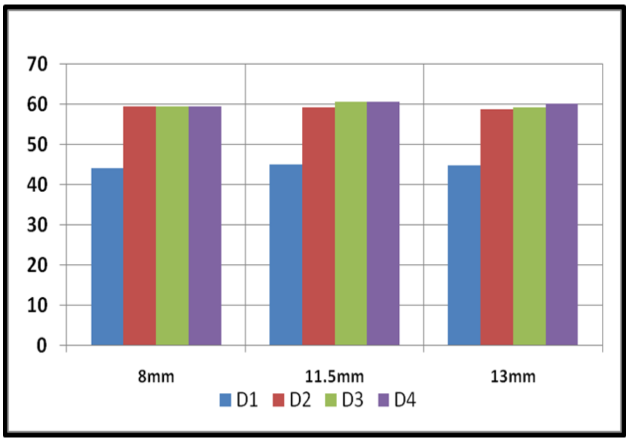

Fig. 5: Stress variation in crown under vertical loading

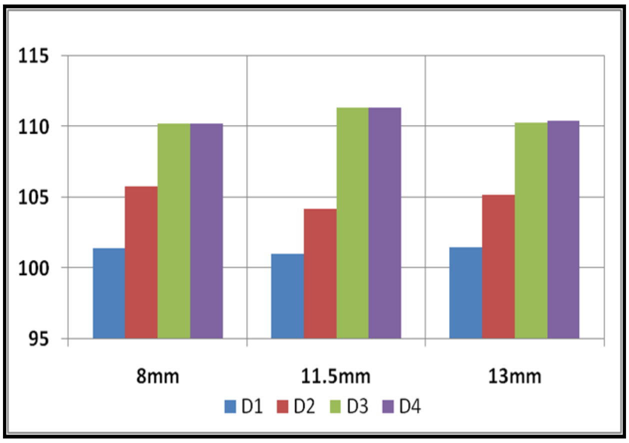

Fig. 6: Stress variation in crown under oblique loading

Maximum Von mises stresses were concentrated at the central fossa of the zirconia crowns on vertical loading and at the mesiobuccal and distobuccal cusps on oblique loading. When subjected to vertical loading, maximum stress of 60.5 MPa was seen in implant length of 11.5mm and in Type 3 and Type 4 bone. Minimum stress of 40 MPa was seen in implant length of 8mm in Type 1 bone. When subjected to oblique loading, maximum stress of 111.34 MPa was seen in implant length of 11.5mm and in Type 3 and Type 4 bone types. Minimum stress of 101.04 MPa was also seen in implant length of 11.5mm in Type 1 bone type.

Maximum Von mises stresses in cortical bone were concentrated at the coronal aspect of the implant bone interface. Stresses were concentrated on buccal side of implant bone interface for vertical loading and on lingual side of implant bone interface for oblique loading. When subjected to vertical loading, maximum stress of 14.2 MPa was seen in implant length of 13mm in Type 4 bone whereas minimum stress of 9.1MPa was seen in 8mm implant length and in Type 1 bone [Fig. 7]. When subjected to oblique loading, maximum stress of 32.3 MPa was seen in implant length of 13mm in Type 4 bone whereas minimum stress of 25.8MPa was seen in 8mm implant length and in Type 1 bone [Fig. 8].

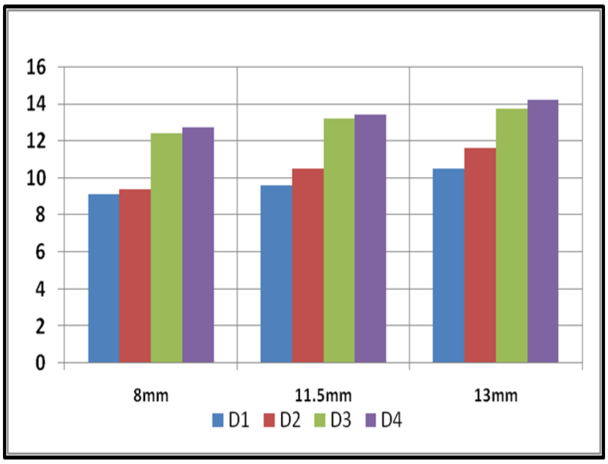

Fig. 7: Stress variation in cortical bone under vertical loading

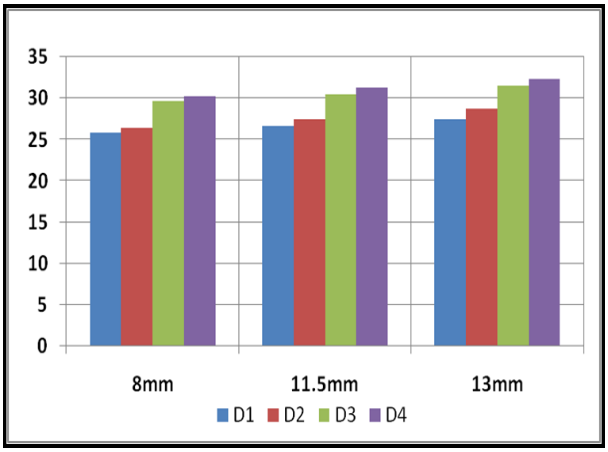

Fig. 8: Stress variation in cortical bone under oblique loading

Maximum Von mises stresses in trabecular bone were concentrated at the apical region of the Implant bone interface. When subjected to vertical loading, maximum stress of 5.4 MPa was seen in implant length of 13mm in Type 2 bone whereas minimum stress of 0.5 MPa was seen in 8mm implant length and in Type 1 bone [Fig. 9]. When subjected to oblique loading, maximum stress of 9.15 MPa was seen in implant length of 13mm in Type 4 bone whereas minimum stress of 0.9 MPa was seen in 8mm implant length and in Type 1 bone [Fig. 10]

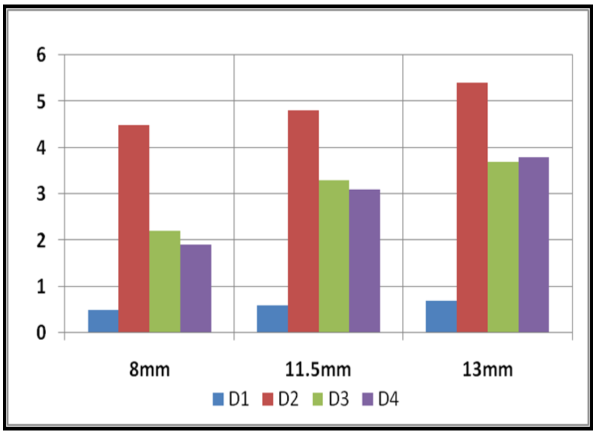

Fig. 9: Stress variation in Trabecular bone under vertical loading

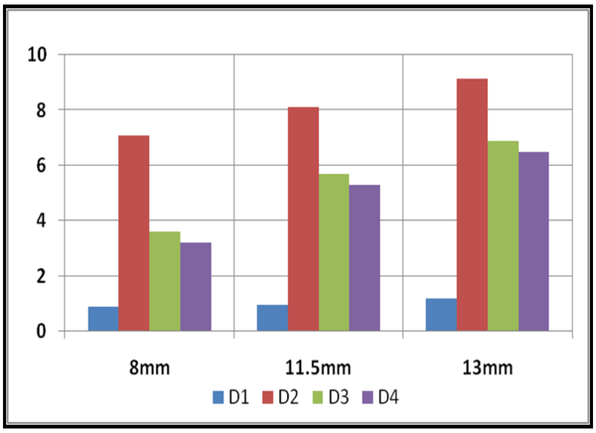

Fig. 10: Stress variation in Trabecular bone under oblique loading

Maximum Von mises stresses in implant abutment system comprising of the implant fixture, abutment and its screw were almost similar on vertical loading as contrast to that of oblique loading. When subjected to vertical loading, maximum stress of 135 MPa was seen in implant length of 11.5mm and 13mm in Type 2, Type 3 and Type 4 bone while minimum stress of 126.2 MPa was seen in implant length of 8mm in Type 1 bone [Fig. 11].

However, when subjected to oblique loading maximum stress of 64 MPa was seen in implant length of 13mm in Type 2 bone while minimum stress of 34.6 MPa was seen in implant length of 8mm in Type 1 bone [Fig. 12].

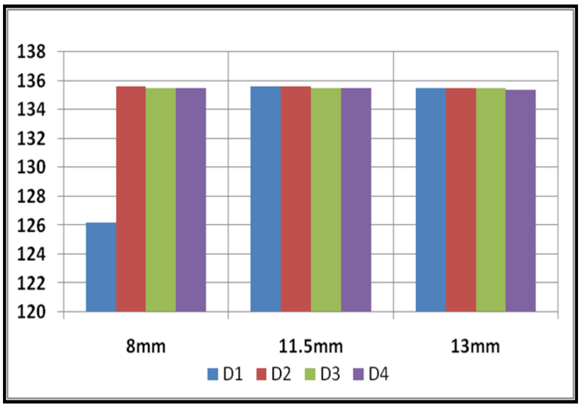

Fig. 11: Stress variation in Implant abutment system under vertical loading

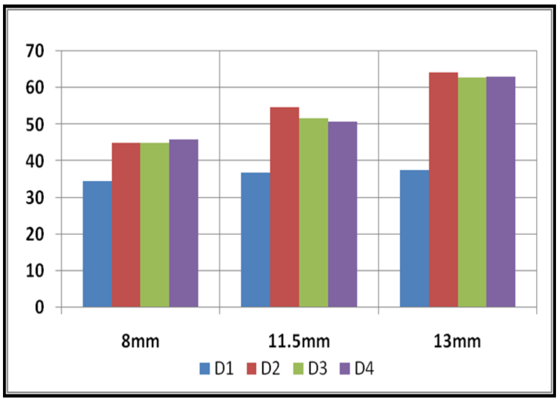

Fig. 12: Stress variation in Implant abutment system under oblique loading

Stress and strain distribution patterns around osseointegrated implants were greatly influenced by a number of biomechanical factors such as material properties of implants and the prosthesis, surface structure, implant geometry, quality and quantity of the surrounding bone and the type of loading conditions.

FEA can be used elaborately to predict the effect of clinical factors on the success of an implant and the biomechanical performance of implant designs.

The major etiology behind short term implant failure has been attributed to insufficient primary stability while long term implant failure was related to overloading on incomplete osseointegrated implants. In the present study, oblique force of 300 N was chosen to simulate the poor prognosis of clinical cases of patients exhibiting parafunctional habits namely, clenching, tooth grinding and bruxism [31].

In the present study, the peak stress values occurred in the implant due to the high modulus of elasticity of titanium and the tightness of the contacts in the implant system. The Von Mises stress values increased with an increase in the implant length. An increase in the implant length caused an increased stress of nearly 10N in the implant system, and this stress caused much more deformation on the implant system which was also seen in the study by Shih-Hao Chang [2].

Crestal bone loss is a phenomenon which is often seen clinically and radiologically exhibiting decrease in marginal bone level were observed in different stages of implant loading. In a number of radiological long-term studies and FEA analyses, loaded implants showed typical bone loss around the implant neck. The results of the present study indicated that maximum Von Mises stresses, in cortical bone, occurred mainly at the buccal aspect of alveolar crest at implant bone interface. Stresses were concentrated more on the buccal aspect of the alveolar crest for vertical loading and on the lingual aspect of the alveolar crest for oblique loading which were similar to results obtained by Papavasiliou et al [11]. Stresses were maximum in Type 4 bone and is minimum for Type 1 bone. In trabecular bone, stresses were concentrated at the apical region of implant-bone interface. Stresses were maximum in Type 2 bone and minimum for Type 1 bone.

M. Sevimay [30] in his study investigated the effect of 4 different bone qualities on stress distribution in an implant-supported mandibular crown, using 3-dimensional (3-D) finite element (FE) analysis. The authors observed that von Mises stresses in D3 and D4 bone quality were 163 MPa and 180 MPa, respectively, and reached the highest values at the neck of the implant and in D1 and D2 bone quality were 150 MPa and 152 MPa, respectively, at the neck of the implant. For the bone qualities investigated by the authors, they observed that stress concentrations in compact bone followed the same distributions as in the D3 bone model, due to the trabecular bone being weaker and less resistant to deformation than the other bone qualities modelled, the stress magnitudes were greatest for D3 and D4 bone which was in accordance with the results of the present study.

Shih-Hao Chang [2] in 2011 simulated the biomechanical behaviors and influences of SDI diameters under various conditions of bone quality by using a validated finite element (FE) model for simulation. The authors combined CAD and CT image system to construct the FE models with 6 mm length SDIs for 6, 7- and 8-mm diameters under three types of bone qualities, from normal to osteoporotic. The authors observed that the Von mises strains of bone under the vertical load was not affected by implant diameter. Similar results were seen in the present study where different implant lengths influence the von mises stress of bone under vertical and oblique loads as compared to different diameters.

Guan [33] in 2009 investigated the influence of bone and dental implant parameters on stress distribution in the mandible by finite element analysis (FEA) and in vitro experiments. The authors observed that the abutment displacement was greater under oblique loading than under axial loading and greater for the longer implants. Similar results were observed in the present study with longer implants when subjected to oblique loading showed greater stresses in the bone implant interface and implant abutment system.

IV. Conclusion

An increase in the implant length increased the stresses in the implant and the peri implant bone and increased the area in which the stresses were distributed. A change in the bone type did not affect the stresses on the implant. Tensile and compressive stresses formed mostly at the alveolar bone around the implant neck which was predominantly the cortical bone. Deformation due to the stresses had great importance for the Type 4 trabecular bone due to the increase in implant lengths. The present study concluded that lesser stresses are created in the peri implant bone by shorter implant lengths in all the four bone types as compared to longer implant lengths when loaded in vertical direction as opposed to on oblique direction. This would enable the clinicians to decide upon the best implant length to be considered in accordance with the quality of bone present. However, clinical studies are required to support the results obtained in the present study.

ACKNOWLEDGEMENT

The authors would like to acknowledge Mr. Rupesh Kumar of Ruthvi Tech Solutions Pvt Ltd. for his contribution in the mechanical tests of the study.

No potential conflict of interest was reported by the authors. The following study did not receive any external funding and was completely self-funded by the authors.

REFERENCES

[1] Bulaqi HA, Mousavi Mashhadi M, Safari H, Samandari MM, Geramipanah F. Effect of increased crown height on stress distribution in short dental implant components and their surrounding bone: A finite element analysis. *J Prosthet Dent.* 2015 Jun;113(6):548-57.

[2] Chang SH, Lin CL, Hsue SS, Lin YS, Huang SR. Biomechanical analysis of the effects of implant diameter and bone quality in short implants placed in the atrophic posterior maxilla. *Med Eng Phys*. 2012 Mar;34(2):153-60.

[3] Cinar D, Imirzalioglu P. The Effect of Three Different Crown Heights and Two Different Bone Types on Implants Placed in the Posterior Maxilla: Three-Dimensional Finite Element Analysis. *Int J Oral Maxillofac Implants*. 2016 Mar-Apr;31(2): e1-e10.

[4] Behnaz E, Ramin M, Abbasi S, Pouya MA, Mahmood F. The effect of implant angulation and splinting on stress distribution in implant body and supporting bone: A finite element analysis. *Eur J Dent.* 2015 Jul-Sep;9(3):311-8.

[5] Sugiura T, Yamamoto K, Horita S, Murakami K, Tsutsumi S, Kirita T. Effects of implant tilting and the loading direction on the displacement and micromotion of immediately loaded implants: an in vitro experiment and finite element analysis. *J Periodontal Implant Sci*. 2017 Aug;47(4):251-262.

[6] Koca OL, Eskitascioglu G, Usumez A. Three-dimensional finite-element analysis of functional stresses in different bone locations produced by implants placed in the maxillary posterior region of the sinus floor. *J Prosthet Dent.* 2005 Jan;93(1):38-44.

[7] Quaresma SE, Cury PR, Sendyk WR, Sendyk C. A finite element analysis of two different dental implants: stress distribution in the prosthesis, abutment, implant, and supporting bone. *J Oral Implantol*. 2008;34(1):1-6.

[8] Wheeler RC. Atlas of Tooth Form. Toronto: Harcourt Canada; 1969:68.

[9] Sevimay M, Turhan F, Kılıçarslan MA, Eskitaşcıoğlu 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; 93:227–234.

[10] Holmes DC, Loftus JT. Influence of bone quality on stress distribution for endosseous implants. *J Oral Implantol* 1997; 23:104–111.

[11] Papavasiliou G, Kamposiora P, Bayne SC, Felton DA. 3D-FEA of osseointegration percentages and patterns on implant bone inter- facial stresses. *J Dent* 1997; 25:485–491.

[12] Meyer U, Vollmer D, Runte C, Bourauel C, Joos U. Bone loading pattern around implants in average and atrophic edentulous maxillae: A finite-element analysis. *J Craniomaxillofac Surg* 2001; 29:100–105.

[13] Tada S, Stegaroiu R, Kitamura E, Miyakawa O, Kusakari H. Influence of implant design and bone quality on stress/strain distribution in bone around implants: A 3-dimensional finite element analysis. *Int J Oral Maxillofac Implants* 2003; 18:357–368.

[14] Papavasiliou G, Kamposiora P, Bayne SC, Felton DA. Three-dimensional finite element analysis of stress distribution around single tooth implants as a function of bony support, prosthesis type and loading during function. *J Prosthet Dent* 1996; 76:633–640.

[15] Iplikçioğlu H, Akça K. Comparative evaluation of the effect of diameter, length and number of implants supporting three-unit fixed partial prostheses on stress distribution in the bone. *J Dent* 2002; 30:41–46.

[16] Eskitascıoglu G, Uşumez A, Sevimay M, Soykan E, Unsal E. The influence of occlusal loading location on stresses transferred to implant- supported prostheses and supporting bone: A three-dimensional finite element study. *J Prosthet Dent* 2004; 91:144–150.

[17] Holmgren EP, Seckinger RJ, Kilgren LM, Mante F. Evaluating parameters of osseointegrated dental implants using finite element analysis—a two-dimensional

comparative study examining the effects of implant diameter, implant shape, and load direction. *J Oral Implantol* 1998; 24:80–88.

[18] Himmlová L, Dostálová T, Kácovský A, Konvicková S. Influence of implant length and diameter on stress distribution: A finite element analysis. *J Prosthet Dent* 2004; 91:20–25.

[19] Akça K, İplikçioğlu H. Finite element stress analysis of the effect of short implant usage in place of cantilever extensions in mandibular posterior edentulism. *J Oral Rehabil* 2002; 29:350–356.

[20] Koca OL, Eskitascioglu G, Usumez A. Three-dimensional finite element analysis of functional stresses in different bone locations produced by implants placed in the maxillary posterior region of the sinus floor. *J Prosthet Dent* 2005; 93:38–44.

[21] Barbier L, Vander Sloten J, Krzesinski G, Schepers E, Van Der Pere G. Finite element analysis of non-axial versus axial loading of oral implants in the mandible of the dog. *J Oral Rehabil* 1998; 25:847–858.

[22] Sertgöz A. Finite element analysis study of the effect of superstructure material on stress distribution in an implant-supported fixed prosthesis. *Int J Prosthodont* 1997; 10:19–27.

[23] Balshi TJ, Hernadez RE, Prsyzlak MC, Rangert B. A comparative study of one implant versus two replacing a single molar. *Int J Oral Maxillofac Implants* 1996; 11:372–378.

[24] Dejak B, Mlotkowski A, Romanowicz M. Finite element analysis of mechanism of cervical lesion formation in simulated molars during mastication and parafunction. *J Prosthet Dent* 2005; 94:520–529.

[25] Lee HE, Lin CL, Wang CH, Cheng CH, Chang CH. Stresses at the cervical lesion of maxillary premolar—a finite element investigation. *J Dent* 2002; 30:283–290.

[26] Aykul H, Toparlı M, Dalkız M. A calculation of stress distribution in metal-porcelain crowns by using three-dimensional finite element method. *J Oral Rehabil* 2002; 29:381–386.

[27] Martin RB, Burr DB, Sharkey NA. Skeletal Tissue Mechanics. New York: *Springer*.1998:127–178.

[28] 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; 100:422–431.

[29] Huang HL, Hsu JT, Fuh LJ, et al. Bone stress and interfacial sliding analysis of implant designs on an immediately loaded maxillary implant: A non-linear finite element study. *J Dent* 2008; 36:409–417.

[30] 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 Mar;93(3):227-34.

[31] Lee JS, Lim YJ. Three-dimensional numerical simulation of stress induced by different lengths of osseointegrated implants in the anterior maxilla. *Comput Methods Biomech Biomed Engin* 2013; 16:1143–1149.

[32] Tian K, Chen J, Han L, Yang J, Huang W, Wu D. Angled abutments result in increased or decreased stress on surrounding bone of single-unit dental implants: A finite element analysis. *Med Eng Phys2012*; 34:1526-31.

[33] Guan H, van Staden R, Loo YC, et al. Influence of bone and dental implant parameters on stress distribution in the mandible: A finite element study. *Int J Oral Maxillofac Implants* 2009; 24:866–876.