TITLE: Comparative evaluation of stress on the peri-implant bone of different densities in mandible restored by various diameter of implants -A Finite Element Analysis .

Dr. Amruta Gujar[*], Dr. Rajeev Singh[2], Dr. Gaurang Mistry[3], Dr. Simran Moorjani[4], Dr. Mishal D"souza[5] Dr.Kunal Mehta[6]

1. **,Post graduate student, Department of Prosthodontics, D. Y. Patil University School of Dentistry, Navi Mumbai, Maharashtra, India,**
2. **Professor, Department of Prosthodontics, D. Y. Patil University School of Dentistry, Navi Mumbai, Maharashtra, India,**
3. **Head of the Department, Department of Prosthodontics, D. Y. Patil University School of Dentistry, Navi Mumbai, Maharashtra, India**
4. **Post graduate student, Department of Prosthodontics, D. Y. Patil University School of Dentistry, Navi Mumbai, Maharashtra, India,**

1. **Associate professor, Department of Prosthodontics, D. Y. Patil University School of Dentistry, Navi Mumbai, Maharashtra, India**
2. **Associate professor, Department of Prosthodontics, D. Y. Patil University School of Dentistry, Navi Mumbai, Maharashtra, India**

ABSTRACT

Stress and strain fields around osseointegrated dental implants are affected by a number of biomechanical factors, including the type of loading, material properties of the implant and the prosthesis, implant geometry, surface structure, quality and quantity of the surrounding bone, and the nature of the bone-implant interface. This study was conducted to compare and evaluate stress on the peri-implant bone of different densities in mandible restored by various diameter of implants using finite element analysis. Materials and methods: An oblique load of 170N was applied on the implants of diameter 3mm, 4mm and 5mm with constant length of 10mm placed in the bone of densities D2 (GROUP I) and D4(GROUP II) using finite element analysis. Results: GROUP I showed less stress in the peri-implant region as compared to GROUP II. However, the mechanical responses obtained from simulated 3D model must be validated by clinical trials.

KEYWORDS: Finite Element Analysis, Solidworks, CAD model , Oblique loading, etc.

INTRODUCTION:

The goal of modern dentistry is to restore the patient's oral condition to normal function, esthetics, speech and comfort regardless of the atrophy, disease or injury to stomatognathic system. Bone loss at the peri-implant region is one of the most common reasons for implant failure compared to the mechanical failure of load bearing prosthetic structure. Furthermore, the failure rate is generally higher for implants in posterior region than in anterior region and in maxilla than in mandible. These evidences in edentulous patients are strictly related to the poor bone quality and quantity in molar regions, as well as to the different bone density between upper and lower jaws.

Stress and strain fields around osseointegrated dental implants are affected by a number of biomechanical factors, including the type of loading, material properties of the implant and the prosthesis, implant geometry, surface structure, quality and quantity of the surrounding bone, and the nature of the bone-implant interface. As far as implant shape is concerned, design parameters that primarily affect load transfer characteristics (the stress/strain distributions in the bone) include implant diameter and the length of the bone-implant interface, as well as, in the case of threaded implants, thread pitch, shape, and depth[1,2]. To increase the surface area for osseous integration, threaded implants are generally preferred to smooth cylindrical ones. Bone quality, surface treatment and thread geometry can significantly influence prognosis of implant. It also favors initial stability and the biomechanical nature of the bone-implant interface after the healing process.[1,2]

Huang S, Tsai C (2003)[3] conducted a 3-dimensional finite element study to assess the stress distribution in the bone, implant and abutment as a function of the implant's diameter and length. They concluded that using finite element analysis that implant length does not decrease the stress distribution of either the implant or the bone. Alternatively, when implant diameter is increased, the stress decreases. For the latter case, the contact area between implant and bone is increased thus the stress concentration effect is decreased. Also, with increased implant diameter the bone loss is decreased and as a consequence the success rate is improved.

Mechanical structures such as implants, abutments and restorations can be simulated in detail with FEA. Loading of these geometric models can provide details of calculated stress and strain values, similar to living structures. Considering the increased use of dental implants in restorative dentistry and the impact of different diameters of implant in densities of bone, this study will be done to evaluate and compare stress distribution on the peri implant bone of different densities in mandible restored by different diameter of implants using Finite Element Analysis.

Materials and Method:

The Finite Element Analysis was carried out using the following steps:

I. Pre-processor
II. Processor
III. Post processor

I. PRE-PROCESSOR

The modeling procedure is as follows:

A] Construction of the geometric model of edentulous mandible

B] Meshing of the CAD model

C] Assigning the material properties to CAD model

D] Loading of the FEA model

A] Construction of the geometric model of the edentulous mandible

This step involves:

1. Modeling of the bone
2. Placing the implant in the bone

1. Modeling of the bone ***:***

CAD model of bone was made using the dimensions 35cm×8cm×15cm in order to accommodate 3 implants of different diameters designed using suitable element type & size.

2. Placing the implant in the bone

The threaded part of all the three implants is embedded in the bone.

B] Meshing of the CAD Model

A three-dimensional Finite Element Mesh was created using the HYPERMESH software. The element size (0.2) was selected according to default setting. The solid 187 type of element suitable for the particular study was 10 nodes tetrahedron element. The elements were constructed to be as accurate as possible within the limitations of the software.

C] Assigning the material properties to the CAD model

All the structures depicted in the model (bone, implant and the prosthetic substructure) were assumed to be linearly elastic, homogenous and isotropic.[20]Although cortical bone contains anisotropic material characteristics and regional stiffness variation, sufficient data is unavailable to establish the principle axis of anisotropy, and so it is assumed to be isotropic.[20] The Young's Modulus, Poisson's ratio and densities for the different materials used were according to literature.[9,17,20] Stringent element quality parameters were followed for the better accuracy of the results.

MATERIAL PROPERTIES

Parts	Materials	Young's Modulus (MPa)	Poisson's ratio	Density (Kg/m^3)
Implant	Grade II Titanium	102000	0.3	4620
Mandibular bone	D2 (GROUP A)	5500	0.323	1720
	D4 (GROUP B)	690	0.3	1376

Depending on the densities of the bone, the models were divided into two groups

Group A: Three implants (Nobel active) of the dimension 3.0 × 10 mm, 4.0 × 10 mm and 5.0 × 10 mm placed vertical and parallel to each in D2 density bone.

Group B: : Three implants (Nobel active) of the dimension 3.0 × 10 mm, 4.0 × 10 mm and 5.0 × 10 mm placed vertical and parallel to each other in D4 density bone.

D] Loading of the FEA model

An oblique load of 170N was applied in the linguo-buccal direction on the titanium abutment. Fig. 1 shows loading of the model done to conduct the FEA.

II. PROCESSOR (SOLVER)

Once the geometry is converted to the finite element for matrices to be solved by the solver, which is a part of the software, the results were generated after all the equations were solved. The solver generates element matrices, computes nodal displacement values and derivatives and solves governing matrix equations.

III. POST PROCESSOR (RESULT)

This step involves going through the large amount of data generated during the solving stage and converted to a form that is easily understood by the operator. The results for the stress distribution are interpreted from colour-coded images seen in the 3D finite element models. Red color represents maximum stress values, this is followed in descending order of orange, yellow, light green, light blue and dark blue. Dark blue represents area of least stress values. The results involve the calculation of Von Mises stresses for each node. The stress flow in each component will be plotted. The results obtained were tabulated and analyzed.

RESULTS

GROUP I (D2) showed less stress in the peri-implant region as compared to GROUP II (D4).

DISCUSSION

The introduction and research in the field of implants have provided a significant improvement in the quality of life for a lot of edentulous patients by allowing for replacement of missing teeth and restoration of chewing function.

Justus and Luftl[25] found that altered strain in hydroxyapatite crystals caused the generation of additional calcium ions. Bone cells may react to this change by either bone apposition or resorption. The piezoelectric properties of bone have also been used to explain the stress dependency of bone remodeling. Bone produces electrical current and potentials proportional to the magnitude of applied stress with a polarity determined by the stress direction. Apposition is thought to occur in regions of negatively charged bone. The actual magnitudes of stress that cause bone remodeling are of particular importance to present study. It has been suggested that there is an optimal value of stress for which as much bone is removed by resorption as is deposited by apposition. With stresses above this value, hypertrophy takes place; below this value, atrophy occurs. There is also a maximum stress limit above which stresses will destroy bone by pathologic resorption. Chamay and Tschantz[26] concluded that compressive stresses promoted bone growth whereas tensile stresses caused resorption.

Meticulous quantifiable information on any place inside a mathematical model can be provided by Finite Element Analysis. As a result, Finite Element Analysis has turned out to be a valued analytical instrument in the estimation of stress and strain in implant systems. One of the salient characteristics of Finite Element Model rests in its near physical similarity among the real structure.[22]

It is said that to obtain more accurate stress predictions, advanced digital imaging techniques can be applied to model to achieve the bone geometry more realistically; the anisotropic and nonhomogeneous nature of the material must be considered; and boundary conditions must be carefully treated with the use of computational modeling techniques.[7]

Huang SC, Tsai CF[3] conducted a 3-dimensional finite element study to assess the stress distribution in the bone, implant and abutment as a function of the implant's diameter and length. They said that increasing implant diameter and length increases the stability of the implant system. They concluded that using finite element analysis that implant length does not decrease the stress distribution of either the implant or the bone. Alternatively, when implant diameter is increased, the stress decreases.

Himmlova et al[8] indicated that implant diameter was more important for improved stress distribution than implant length.

As many researchers have done study to evaluate relation between bone diameter and stress distribution in peri implant region, but only few studies are done to evaluate the effect of implant diameter on different bone densities. However, present study was done to evaluate the relation between implant diameters, bone densities, peri implant region and loading parameters using Finite Element Analysis.

During mastication, two types of loads are applied to an implant supported prosthesis, the first is created by sliding movement of mandible from lateral into centric occlusion when grinding a food bolus, the second arises from vertical approach of mandible into centric occlusion.[3]

Ritcher EJ[6] has done a study in which the maximum transverse and vertical forces applied eccentrically to the axis of dental implants during oral function. He concluded that the oblique load (linguo buccal) during chewing resulted in the highest bending moments (170N mean maximum) of 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.[5] Hence in the present study, linguo-buccal force of 170N was applied in oblique direction.

Stress distribution in FEA studies is generally interpreted as von Mises stress, which could be maximum and minimum principal stress or it could be principal strains. von Mises stress is estimated in three planes, that is, x-axis, y-axis, and z-axis using a formula. Validation is done by comparing the current

FEA results with that of the previous studies related to a particular topic. It provides knowledge on whether precise models were designed for the study or not. Further, it would corroborate the results of previous studies and it may either support or refute with the literature.[21]

As per the process of finite element analysis, a construction of CAD model of bone was done (Fig. 2) The loading forces on the models were static ones and the assumption was made that the left and right half of the mandible are equally loaded as in bilateral biting. Because of the symmetry of the models with respect to the midsagittal plane only the right half of the mandible was subdivided into elements. At the nodes in the symmetry plane symmetry boundary conditions were prescribed. All the materials used in the models were regarded as isotropic, homogeneous and linearly elastic. A XYZ-coordinate system is used to describe support and loading. The positive X-axis is directed towards the outside of the mandible, the positive Y-axis is directed towards the top of the mandible and the positive Z-axis (only present in the three- dimensional models) is directed backwards.[25] Followed by placement of the implants of different dimensions (3.0 × 10 mm, 4.0 × 10 mm and 5.0 × 10) in the bone using software AUTOCAD 2017 was done. Meshing of the model was done using HYPERMESH software. Then material properties like Young's modulus, density and Poisson's ratio were assigned to the model.

The best way to validate FEA results is to conduct in vitro and in vivo experimental studies simultaneously. If the results are good, then it could be recommended for future studies.[22]

According to finite element analysis done by Georgiopoulos B et al[10] specific correlation has been identified between implant diameter and strain distribution on an implant. Increased implant length results in stress reduction on the implant in both immediate and delayed loading. For a given implant length, the stresses are lower after the phase of osseointegration. To evaluate the relation between implant length, bone density and stress concentration in peri implant region, further research needs to be done.

CONCLUSION

It was found that distribution of stress is uniform in the middle and apical portion of the bone, in both groups. Stresses appeared to be more concentrated in crestal portion of the peri implant region in both groups. The stress concentration in D2 density bone (GROUP A) was found to be less than the stress concentration in D4 density bone (GROUP B).

The Finite Element Analysis gives us a good insight into the stress generated under different loading conditions in vitro. There are some limitations of study such as the loading conditions maybe an approximation of the complex function of mastication. The mechanical responses obtained from simulated 3D model must be validated by clinical trials. Hence, it is noted in clinical situations for rehabilitation of edentulous space in posterior mandible with D2 or D4 density of bone, wider diameter of the implant should be selected depending on the clinical and radiological diagnosis.

TABLE 1

AVERAGE STRESS VALUES IN GROUP A AND GROUP B IN CRESTAL REGION OF THE IMPLANT UNDER OBLIQUE LOADING OF 170N IN MPa

	3.0 × 10 mm	4.0 × 10 mm	5.0 × 10 mm
GROUP A	23.04	11.60	8.81
GROUP B	44.74	17.48	11.22

TABLE 2

MEAN STRESS VALUES IN GROUP A UNDER OBLIQUE LOADING OF 170N IN MPa

	3.0 × 10 mm	4.0 × 10 mm	5.0 × 10 mm
CRESTAL	16.32	11.58	9.33
MIDDLE	8.35	7.77	5.98
APICAL	7.87	6.37	5.55

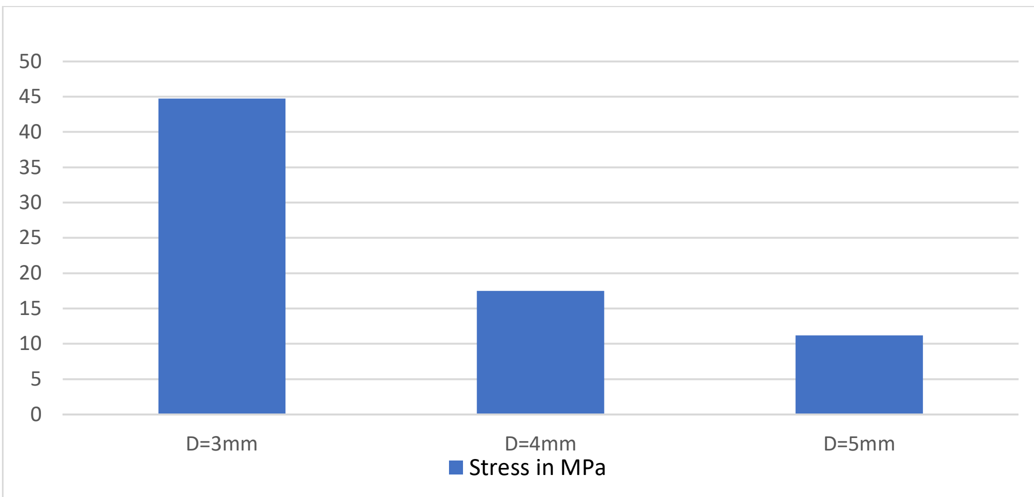
TABLE 3

MEAN STRESS VALUES IN GROUP B UNDER OBLIQUE LOADING OF 170N IN MPa

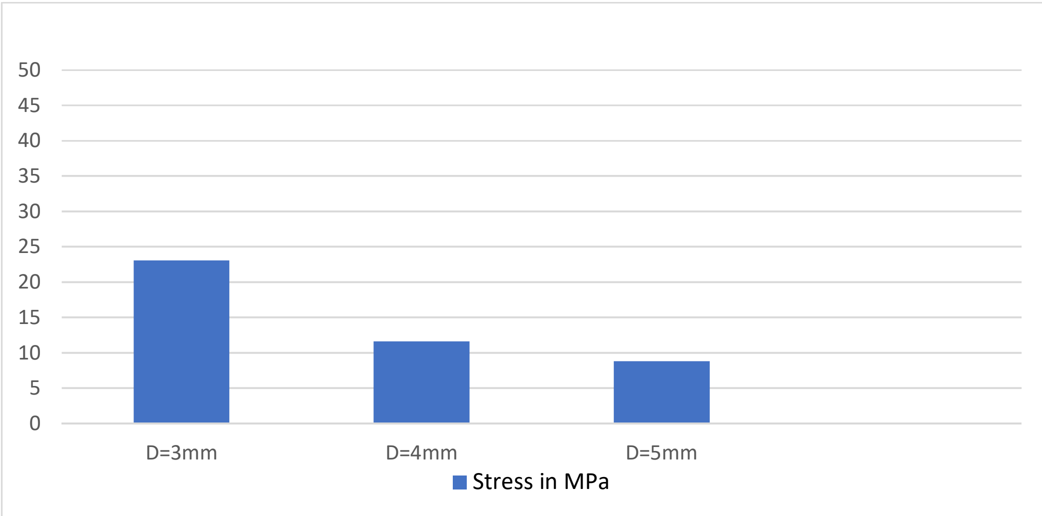
	3.0 × 10 mm	4.0 × 10 mm	5.0 × 10 mm

CRESTAL	**20.04**	**17.40**	**15.81**
MIDDLE	**15.36**	**11.1**	**6.27**
APICAL	**8.62**	**7.79**	**5.90**

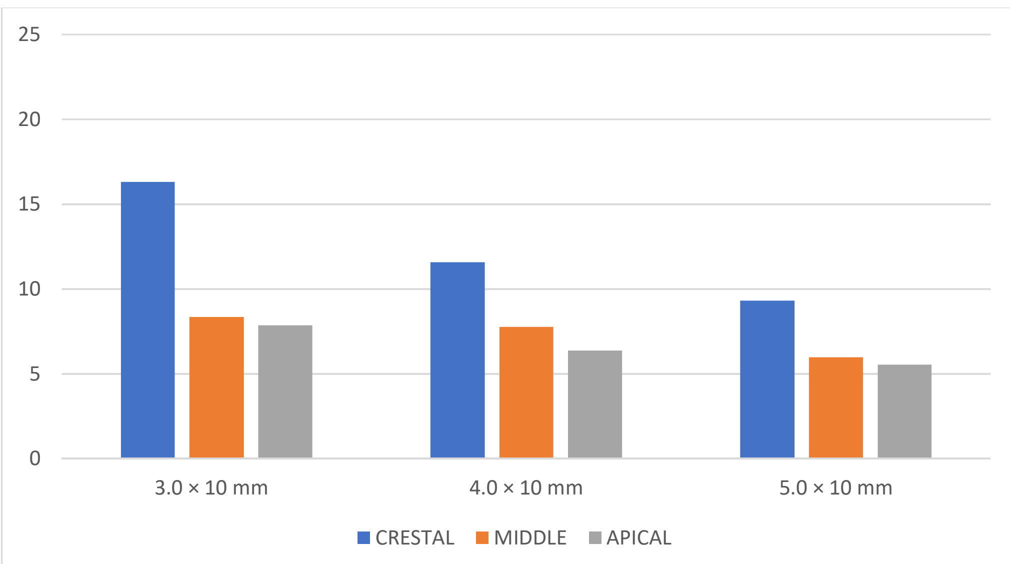
AVERAGE STRESS VALUES IN GROUP A IN CRESTAL REGION OF THE IMPLANT UNDER OBLIQUE LOADING OF 170N IN MPA.



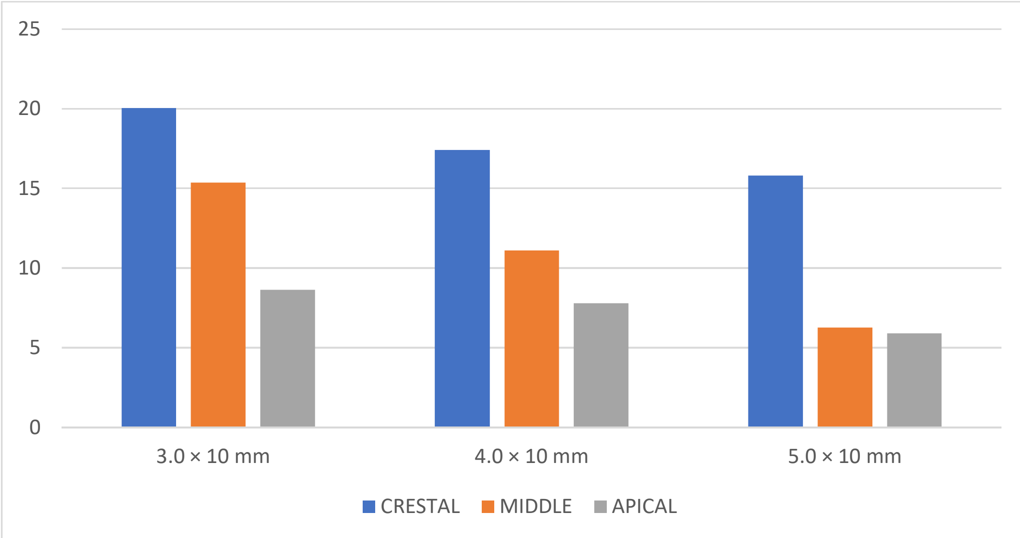
AVERAGE STRESS VALUES IN GROUP B IN CRESTAL REGION OF THE IMPLANT UNDER OBLIQUE LOADING OF 170N IN MPA



MEAN STRESS VALUES IN GROUP A UNDER OBLIQUE LOADING OF 170N IN MPa



MEAN STRESS VALUES IN GROUP B UNDER OBLIQUE LOADING OF 170N IN MPa



REFERENCES

1. Hsu ML and Chang CL. Application of finite element analysis in dentistry. Finite Element Analysis. 1st ed. Germany. 2010;123-7.
2. Ibrahim M, Thulasingam C, Nasser KSGA, Balaji V, Rajakumar M, Rupkumar P. Evaluation of Design Parameters of Dental Implant Shape, Diameter and Length on Stress Distribution: A Finite Element Analysis. J Indian Prosthodont Soc. 2011;11(3):165–171.
3. Huang SC, Tsai CF. Finite element analysis of a dental implant. Biomed Eng Appl Basis Comm, April. 2003;15(2):36-9.
4. Rieger MR, Fareed K, Adams WK, and Tanquist RA. Bone stress distribution for three endosseous implants. J Prosthetic Dentistry, February 1989;61(2):671-6.
5. Holmes DC, Loftus JT. Influence of bone quality on stress distribution for endosseous implants. J Oral Implantology. 1997;23(3):104-11.

6. Richter EJ, Med D, Dipl-Ing. In Vivo Horizontal Bending Moments on Implants. Int J Oral Maxillofac Implants.1998;13:232–44.
7. Geng JP, Keson, Tan BC, Liu GR. Application of finite element analysis in implant dentistry: A review of the literature. J Prosthet Dent. 2001;85:585-98.
8. Himmlova L, Dostalova T, Kacovsky A, Svatava K. Influence of implant length and diameter on stress distribution: A finite element analysis. J Prosthetic Dentistry. 2004;91(1):20-5.
9. Guan H, Staden RV, Loo YC, Johnson N, Ivanovski S, Meredith N. Evaluation of Multiple Implant-Bone Parameters on Stress Characteristics in the Mandible Under Traumatic Loading Conditions. Int J Oral Maxillofac Implants. 2006;25:461-72.
10. Georgiopoulos B, Kalioras K, Provatidis C, Manda M, Koidis P. The effects of implant length and diameter prior to and after osseointegration: A2-D finite element analysis. J Oral Implantology. 2007;33(5):243-56.M
11. Measuring Implant Stability: A REVIEW OF DIFFERENT METHODS JOURNAL OF DENTAL IMPLANTS(OFFICIAL PUBLICATION OF INDIAN SOCIETY OF ORAL IMPLANTOLOGIST VOL 4 ISSUE 2 NO 165-169 JULY 2014)