ISSN: 2581-5989
PubMed - National Library of Medicine - ID: 101738774

International Journal of Dental Science and Innovative Research (IJDSIR)
IJDSIR : Dental Publication Service
Available Online at:www.ijdsir.com
Volume – 7, Issue – 4, July– 2024, Page No. : 45 - 51

Evaluation of Stress and Strain Generated Around Different Dental Implants, Prosthetic Designs and Loading Conditions- A Finite Element Analysis Study

[1]Dr.Nilesh Patel,Government Dental College and Hospital, Ahmedabad

[2]Dr. Sanjay Lagdive, Government Dental College and Hospital, Ahmedabad

[3]Dr. Rupal J Shah, Government Dental College and Hospital, Ahmedabad

[4]Dr. Nilesh Gadiya, Government Dental College and Hospital, Ahmedabad

[5]Dr. Pallavi Naik, Government Dental College and Hospital, Ahmedabad

Corresponding Author: Dr. Nilesh Patel, Government Dental College and Hospital, Ahmedabad

Citation of this Article: Dr. Nilesh Patel, Dr. Sanjay Lagdive, Dr. Rupal J Shah, Dr. Nilesh Gadiya, Dr. Pallavi Naik, “Evaluation of Stress and Strain Generated Around Different Dental Implants, Prosthetic Designs and Loading Conditions- A Finite Element Analysis Study”, IJDSIR- July– 2024, Volume –7, Issue - 4, P. No.45 – 51.

Type of Publication: Original Research Article

Conflicts of Interest: Nil

Abstract

Introduction: The macro design of dental implants significantly impacts their stability, load distribution, and stress generation. Although various studies have explored these factors, clinical detection of stress distribution remains challenging.

Objective: This FEA study seeks to assess the influence of different implant designs on stress and strain distribution, aiming to provide valuable insights for optimized treatment planning and implant success.

Material and method: Using SOLIDWORKS-2014 and ANSYS Workbench, this finite element study analyzed stress and strain distributions in three mandibular implant designs. Assumptions about material properties and osseointegration were made, with axial and non-axial loads applied. The study visualized stress using color graphics and presented strain distribution in contour plots, focusing on implant components and bone. Findings provide insights into stress and strain variations around different components, aiding the comparison of three implant designs.

Results: This study assessed stress and strain in six finite element models under axial (Group 1) and non-axial (Group 2) loads. In Group 1, Model A had the highest strain in cortical bone and gingiva, while Model C showed the lowest. For prosthetic components, Model A had the highest strain, and Model C the lowest. In Group 2, Model B had the highest strain in cortical and trabecular bone, while Model A exhibited the highest gingival strain. Group 1 showed maximum stress in Model A's cortical bone and gingiva, while Group 2 had less stress but maximum in the gingiva. Model A consistently had higher von Mises stress in all bone types, concentrating at the most coronal portion of the cortical bone.

Conclusion: Implants with double lead threads showed lower stress than those with greater taper and microthread designs. Cortical bone patterns reduced stress, while ceramic crowns had higher stress than abutments with screws and implant fixtures. Opting for a double lead thread and cortical bone may enhance implant success, but further clinical research is needed for validation as a reliable treatment modality.

Keyword: Finite element analysis, Dental implant, Stress, Strain, Alveolar Bone, Prosthesis, Axial and Non-axial Loading.

Introduction

The macro design of a dental implant is crucial for its primary stability, load distribution, and stress generation. Factors such as bone quality, location, and clinical situation influence the choice of design elements. A proper assessment and customized treatment plan are essential for optimal results.(1)

Numerous studies have been conducted to evaluate the effect of implant macrodesign on primary stability and its correlation with the success of osseointegration. Various factors affecting implant stability during the healing process have been studied, including stress distribution, particularly at the crestal region.(2,3)

However, it is difficultto detect stress distribution on the implant and bone clinically. Therefore, using numerical methods, such as finite element analysis (FEA) makes it possible to evaluate the stress on dental implants and surrounding bone.(4)

According to Chunet al., the maximum effective stress induced by an oblique load could be twice as high as the maximum effective stress caused by an equal amount of vertical load.(5)

This FEA study aimed to analyze the influence of different implant designs on the stress and strain distribution to the implants, prosthetic crown, and surrounding bone.

The null hypothesis H0, predicts thatthere is no significant difference in the stress and strain distribution to the implants, prosthetic crown, and surrounding bone among different implant designs in the FEA study.

Material and method

A finite element assessment requires a definition of the parameters that characterize the model in which the study is carried out.

In this study, the software used for modeling was SOLIDWORKS-2014, which is a computer program used for solid modeling in computer-aided design (CAD) and computer-aided engineering (CAE). Once a structure is created and assigned material properties, it can be analyzed for stress distributions during force application using finite element software. The ANSYS Workbench was used as the finite element software in this study. The stresses were expressed as either compressive (negative) or tensile (positive) values. The combination of the absolute values squared off all stresses in the global (x, y, z directional axes) is known as Von Mises stresses.

System configuration

A computer with the following system configuration was used

- Windows edition- Windows 7 Ultimate, service pack 2
- Processor- Intel® Core ™ i5 CPU M 430@ 2.27GHz 2.26GHz
- RAM: 4.00GB
- 64-bit operating system

The geometric model of the mandibular body was constructed based on the measurements of a dried human dentulous mandible. Three different implant thread designs were designed to be placed in the body of the mandible at the right and left of the first molar region.

Geometric model of Mandible

The edentulous section of the mandible was modeled based on the measurements of a dried human edentulous mandible.

- The dimensions of the mandibular section are,Height - 15mm Width - 9mm
- Thickness of cortical bone, Crestal - 2mm, Buccal and lingual - 2mm

Cylindrical implant

Three different implant thread designs according to the design

Model A: Microthread double thread and tapered

Length of the implant: 12 mm, Diameter of the implant: 4mm

Model B: Morse taper and internal hex connection

Length of the implant: 11.5mm, Diameter of the implant:4.5mm

Model C: Double lead threads

Length of the implant:11.5mm, Diameter of the implant:4.5mm

The program used implied several assumptions about the mechanical properties of the simulated structures.

1. **Homogeneity**

   The mechanical properties of a material are thought to be the same in the entire structure.
2. **Isotropy**

   The material properties are the same in all directions.
3. **Linear elasticity**

   The deformation or strain of the structure is proportional to the applied force and independent of the strain rate.

Bone Implant Interface

A continuous bond between bone and implant along the entire interface was assumed, which under loading resulted in no relative motion between the bone and implant. This was accepted as the clinical situation assuming the implant was completely osseointegrated.

Loads applied

Two clinical situations were considered for load application:

- Axial loads of magnitudes 150N are applied during uniform bilateral biting which is directed downwards parallel to the long axis of the implant.
- Non-axial loads of magnitudes 150N are given at an angle of 45°from the long axis of the implant as during lateral movements.

Analysis

A total of 3 models were formed and grouped into two for ease of analysis.

- Group 1 consisted of 3 models on which axial loads were applied.
- Group II consisted of 3 models on which non-axial loads were applied.

Model Aconsists of a mandibular section with microthread, double thread, and tapered implant thread design with a 4.5mm diameter and 11.5mm length placed in the mandibular molar region.

Model B consists of a mandibular section with Morse taper and internal hex connection implant design with a 4.5mm diameter and 11.5mm length placed in the mandibular molar region.

Model C consists of a mandibular section with a double lead thread implant design with a 4.3mm diameter and 11.5mm length placed in the mandibular molar region.

The different models as discussed above were analyzed using the linear static module of the finite element software to obtain the deformation pattern and the stress and strain distribution in the structure.

The output form, the finite element analysis of a 3D model of this mesh size, will be very voluminous. Hence to better visualize the stress state in the structure, using colour graphics the calculated stresses are presented in the form of color bands. Each color band represents a particular range of stress values.

Stresses

The stress distribution in the structure is presented in the form of contour plots for different cases of the model studied. To get a clear picture of the stress status the contour plots have been made separately for areas of special interest i.e. implant fixture and abutment with screw and cortical and trabecular bone around the implant. For comparison of the magnitude of stress in each model, the peak Von Mises stresses in the areas of special concern was tabulated.

Strain

The strain distribution in the structure is presented in the form of a contour plot for different cases of the model studied. To get a clear picture of the strain the contour plot has been made separately for an area of special interest i.e. implant fixture and abutment with screw and cortical and trabecular bone.

This study includes stress and strain generated around the cortical, trabecular bone, abutment with screw and ceramic crown, and gingiva all these variables comparing with three implant designs.

Results

The present study evaluated the stress and strain distribution of six finite element models which were grouped into two, Group 1 consisted of a model on which axial loads of magnitude 150N were applied and Group 2 consisted of models on which non-axial loads of magnitudes 150N.

For all the models peak Von Mises stresses and Principal strain were studied.

Group 1 model A showed maximum strain in cortical types of bone and gingiva and model c showed minimum strain in both cortical bone and gingiva. Group 1 prosthetic component (ceramic crown, abutment screw) model A was maximum strain generated and model c was minimum and implant fixture of all three models are almost the same strain generated.

Group 2 model B showed maximum strain in cortical bone as well as trabecular bone but in the gingival portion model A showed maximum strain. Group 2 prosthetic component (ceramic crown, abutment screw) model A was maximum strain.

Group 1 maximum stress generated in model A cortical type of bone and gingiva and less generated in model C.Group 1 maximum stress generated in model A ceramic crown and fixture and model B and model C showing comparative less stress. Group 2 maximum stress generated in model A cortical type of bone and gingiva and less in model C.Group 2 maximum stress generated in model A ceramic crown and fixture and less in model C.

Group 1 (axial loading)had maximum stress generated around the implant in cortical bone and Group 2 (non-axial loading)had comparatively less stress but Group 2 had maximum stress in the gingiva. All the values of stress generated in bone and implant in the following table

The von Mises stress distribution in bone was higher for model A in all types of bone. The maximum values in stress distribution in bone decreased with an increase in medullar bone density for all implant designs. The distribution of stress to the bone was concentrated, in all the examples, at the level of the most coronal portion of the cortical bone.

Statistical Analysis

Data was analyzed using the statistical package SPSS 26.0 (SPSS Inc., Chicago, IL) and the level of significance was set at $p<0.05$. Descriptive statistics was performed to assess the mean and standard deviation of the respective groups. The normality of the data was assessed using the Shapiro-Wilkinson test. A non-parametric test was used as the Data was not following normality. Inferential statistics to find out the difference between the groups was done using the Whitney U test. Within-group analysis was done using the Kruskal-Wallis test. The Bonferroni test was used as a post hoc test.

Discussion

The present study aimed to evaluate the stress and strain distribution around the implant thread design placed in the molar region of the mandible. The study consists of two types of bone (trabecular and cortical bone). The study also consistsof stress and strain generated around the ceramic crown, abutment with screw and implant fixture, and gingiva.The study consists of two groups each group has a set of three implant designs.

Strain distribution around implant and bone in both groups showed minimum difference. Model A showed maximum strain around the implant crest model and gingiva. Stress generated around the bone in different thread design cortical types of bone and model A showed maximum stress around cortical bone and gingiva and in nonaxial loading maximum stress around in gingiva and crest module of the implant fixture.

Sugiura et al.(6)showed the same pattern in their study. These authors also observed that the strain distribution was higher in low-density medullar bone and decreased with an increase in cortical thickness, in conditions of immediate and delayed loading.

Works from Yalçin et al.(7)and Sevimay et al.(8)reported maximum von Mises stress values in D4 bone quality (1 mm thick cortical bone and low-density medullar bone), compared to three other bone qualities (with higher medullar densities and thicker cortical bone). Baggy et al.observed higher stress distributions in the maxillary bone (less dense) than in the mandibular bone, with five different types of implants.

In Group I and Group II the Von Mises stress values and maximum principal stress values were obtained for microthread double thread and tapered implant and Morse taper implant thread design and less von Mises stress and principal stress and strain generated around double lead threaded implant design.

The present finite element study suggests that implants with microthread and tapered and Morse tapered implant designs have more favorable stress and strain distribution compared to double lead thread implant designs. Ceramic crowns have a more favorable stress distribution compared to the abutments with screws.

From the value attained, it was also shown the von Mises stress and strain in different bone patterns and gingiva results have more stress around cortical bone, and gingiva and trabecular bone have less stress compared to cortical bone and trabecular bone.

The present study suggests that double lead threaded implant and cortical bone patternsare most favorable and enhance the primary stability and survival rate of the implant. In Finite element analysis, since the variables may be manipulated with computer precision, chance variation from sampling error is eliminated. The same Finite element analysis repeated any number of times will yield identical results 100% of the time.

Thus, it is certain that the results are always caused by the manipulation of the variables and not by chance. Hence conventional interferential statistical analysis is not normally included in a Finite element analysis study. However, there are different sources of potential error. If key features such as material properties, geometry, interface status, boundary conditions, or loading of the real system to be modeledare inaccurately represented, the model may be deficient or incorrect Finite element analysis has proved to be an extremely accurate and precise method for analyzing structures.

However, living structures are more than mere objects. Finite Element analysis is based on mathematical calculations based on simulation of the structure in its environment. But living tissues are beyond the confines of set parameters and values i.e. biology is not a computable entity.

Therefore, although Finite Element analysis provides a very sound theoretical basis for understanding the behavior of a structure in a given environment, it should not be considered alone. Actual experimental techniques and clinical trials should follow the finite element analysis to establish the true nature of the biological system.

The alternative hypothesis (H1) asserts a statistically significant disparity in stress and strain distribution among diverse implant designs within the FEA study, elucidating their impact on implants, prosthetic crowns, and surrounding bone.

Limitations of this study should be acknowledged, particularly the reliance on a mathematical model that inherently falls short of fully capturing the intricate complexities within the biological context. The obtained results serve as preliminary guidelines, highlighting the need for subsequent in-vitro stress analyses and eventual validation through clinical trials.

Conclusion

The study revealed that implants featuring a double lead thread exhibited lower stress levels compared to those with a greater taper and microthread design. Additionally, cortical bone patterns demonstrated reduced stress, while ceramic crowns exhibited higher stress levels in comparison to abutments with screws and implant fixtures. These findings imply that opting for a double lead thread and cortical bone type may enhance implant success. Nevertheless, it is essential to note that further clinical research is warranted to establish these observations as a reliable and successful treatment modality.

References

1. Buser D, Janner SFM, Wittneben J, Brägger U, Ramseier CA, Salvi GE. 10-Year Survival and Success Rates of 511 Titanium Implants with a Sandblasted and Acid-Etched Surface: A Retrospective Study in 303 Partially Edentulous Patients. Clin Implant Dent Relat Res. 2012 Dec 30;14(6):839–51.
2. Stoilov M, Shafaghi R, Stark H, Marder M, Kraus D, Enkling N. Influence of Implant Macro-Design, -Length, and -Diameter on Primary Implant Stability Depending on Different Bone Qualities Using Standard Drilling Protocols—An In Vitro Analysis. J FunctBiomater. 2023 Sep 12;14 (9):469.
3. Lozano-Carrascal N, Salomó-Coll O, Gilabert-Cerdà M, Farré-Pagés N, Gargallo-Albiol J, Hernández-Alfaro F. Effect of implant macro-design on primary stability: A prospective clinical study. Med Oral Patol Oral Cir Bucal. 2016 Mar 1;21 (2):e214-21.
4. LeblebicioğluKurtuluş I, Kilic K, Bal B, Kilavuz A. Finite Element Analysis of the Stress Distribution Associated With Different Implant Designs for Different Bone Densities. Journal of Prosthodontics. 2022 Aug 6;31(7):614–22.
5. Chun HJ, Cheong SY, Han JH, Heo SJ, Chung JP, Rhyu IC, et al. Evaluation of design parameters of

osseointegrated dental implants using finite element analysis. J Oral Rehabil. 2002 Jun;29(6):565–74.

6. Sugiura T, Yamamoto K, Horita S, Murakami K, Kirita T. Micromotion analysis of different implant configuration, bone density, and crestal cortical bone thickness in immediately loaded mandibular full-arch implant restorations: A nonlinear finite element study. Clin Implant Dent Relat Res. 2018 Feb 6;20(1):43–9.
7. Yalçın M, Kaya B, Laçin N, Arı E. Three-Dimensional Finite Element Analysis of the Effect of Endosteal Implants with Different Macro Designs on Stress Distribution in Different Bone Qualities. Int J Oral Maxillofac Implants. 2019;34(3):e43–e50.
8. 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.

Legend of Tables

Table 1: Strain generated due to axial loading in bone and implant in group 1.

GROUP 1 AXIAL LOADING			
STRAIN DISTRIBUTION IN BONE			
IMPLANT MODEL	CORTICAL BONE	TRABECULAR BONE	GINGIVA
MODEL A	0.0012	0.00091	0.0021
MODEL B	0.00061	0.00061	0.0007
MODEL C	0.00025	0.00059	0.00029
GROUP 1 AXIAL LOADING			
STRAIN DISTRIBUTION IN IMPLANT			
IMPLANT MODEL	CERAMIC CROWN	ABUTMENT WITH SCREW	IMPLANT FIXTURE
MODEL A	0.0088	0.0087	0.00098
MODEL B	0.0044	0.0035	0.00091
MODEL C	0.0058	0.002	0.00096

Table 2: Strain generated due to non-axialloading on implant

GROUP 2 NON-AXIAL LOADING			
STRAIN DISTRIBUTION IN IMPLANT			
IMPLANT MODEL	CERAMIC CROWN	ABUTMENT WITH SCREW	IMPLANT FIXTURE
MODEL A	0.009	0.0086	0.00093
MODEL B	0.0044	0.0035	0.00096
MODEL C	0.0059	0.002	0.00096
STRAIN DISTRIBUTION IN BONE			
IMPLANT MODEL	CORTICAL BONE	TRABACULAR BONE	GINGIVA
MODEL A	0.00095	0.00065	0.0026
MODEL B	0.00107	0.0014	0.00085
MODEL C	0.00036	0.00059	0.00066

Table 3: Von Mises Stress generated due to axialloading

GROUP 1 AXIAL LOADING			
VON MISES STRESS DISTRIBUTION IN BONE			
IMPLANT MODEL	CORTICAL BONE(Mpa)	TRABACULAR BONE (Mpa)	GINGIVA (Mpa)
MODEL A	18.01	0.89	66.54
MODEL B	8.13	0.57	24.801
MODEL C	3.6	0.54	11.01
VON MISES STRESS DISTRIBUTION IN IMPLANT			
IMPLANT MODEL	CERAMIC CROWN (Mpa)	ABUTMENT WITH SCREW(Mpa)	IMPLANT FIXTURE (Mpa)
MODEL A	575.99	109.53	224.96
MODEL B	259.74	96.15	226.7
MODEL C	305.29	92.5	211.5

Table 4: Von Mises Stress generated due to non-axialloading

GROUP 2 NON-AXIAL LOADING			
VON MISES STRESS DISTRIBUTION IN BONE			
IMPLANT MODEL	CORTICAL BONE(Mpa)	TRABACULAR BONE (Mpa)	GINGIVA (Mpa)
MODEL A	14.12	0.64	85.71
MODEL B	14.65	1.34	31.23
MODEL C	5.05	0.57	24.66
VON MISES STRESS DISTRIBUTION IN IMPLANT			
IMPLANT MODEL	CERAMIC CROWN (Mpa)	ABUTMENT WITH SCREW(Mpa)	IMPLANT FIXTURE (Mpa)
MODEL A	544.6	104.18	239.74
MODEL B	259.4	101.26	224.69
MODEL C	310.44	98.34	210.5