IOSR Journal of Dental and Medical Sciences (IOSR-JDMS)
e-ISSN: 2279-0853, p-ISSN: 2279-0861.Volume 24, Issue 1 Ser. 5 (January. 2025), PP 63-68
www.iosrjournals.org

Analyzing the Stress Distribution Effects of Different Implant Thread Designs through Three-Dimensional Finite Element Analysis – Original Research

Dr. Sarankumar M
Dr. Sreedevi V
Dr. Haritha V
Dr. Swetha D
Dr. Alagammai S
Dr. Kalyani Ramkumar Sadhana

PG Student, Department of Prosthodontics and Crown & Bridge, Sree Balaji Dental College & Hospital, Chennai.
Professor & Research Scholar, Department Of Prosthodontics and Crown & Bridge, Sree Balaji Dental College & Hospital, Bharath Institute of Higher Education and Research, Chennai.
Pg Student, Department Of Prosthodontics and Crown & Bridge, College Of Dental Sciences, Davangere.
Pg Student, Department Of Prosthodontics and Crown & Bridge, Sri Hasanamba Dental College and Hospital, Hassan.
Pg Student, Department of Prosthodontics and Crown & Bridge, Sree Balaji Dental College & Hospital, Chennai.
Senior Lecturer, Department of Prosthodontics and Crown and Bridge, Sree Balaji Dental College & Hospital, Chennai.

Abstract
This study investigates the impact of various implant thread designs on stress distribution at the bone-implant interface using three-dimensional finite element analysis (FEA). Implants with V-shaped, square, buttress, reverse buttress, and trapezoidal threads were modelled and subjected to a 250 N vertical load in ANSYS software. The results indicated that trapezoidal threads showed superior stress distribution, reducing stress levels at the implant-bone interface, while V-shaped threads resulted in the highest stress concentrations. These findings emphasize the importance of thread geometry in achieving optimal stress distribution, which may enhance implant longevity and improve clinical outcomes.
Keywords: *Implant thread designs, Finite Element Analysis, Stress distribution, Dental implants, Osseointegration*

I. Introduction

Dental implants are integral to modern prosthodontic treatment, offering a stable and functional solution for patients with missing teeth. The geometry of implant threads plays a vital role in how stress is transferred to the surrounding bone, directly influencing the implant's stability and long-term success. Implant designs that promote balanced stress distribution reduce the risk of bone loss, thereby improving the chances of successful osseointegration.

Despite ongoing innovations in implant technology, the comparative influence of different thread designs on stress management is not fully understood. Although previous studies have explored various aspects of implant biomechanics, comprehensive evaluations of thread geometry's role in stress distribution remain limited. This study aims to address that gap by examining five distinct thread designs using three-dimensional finite element analysis. Through this approach, we aim to provide insights that can assist clinicians in selecting optimal implant designs for improved performance under physiological loading.

II. Materials And Methods

This research employed FEA to analyze stress patterns across different implant thread geometries. Five thread designs -V-shaped, square, buttress, reverse buttress, and trapezoidal were modelled using Creo Parametric 7.0 (figure 1,2,3,4). Cortical and cancellous bone were also constructed (figure 5,6) as part of the model to simulate realistic bone conditions. The implant models, including the abutments, were standardized to 13 mm in length and 5 mm in diameter, with thread width 0.6mm and pitch 1.2mm consistent across designs to isolate the impact of thread geometry (figure 7).

Material properties were assigned based on previous literature (table 1), with bone tissues modelled as isotropic and linearly elastic for simplification. Once the virtual models were prepared, they were imported into ANSYS Workbench for simulation (figure 8). A vertical force of 250 N was applied along the axis of the implant to simulate typical masticatory loads (figure 9). Fine meshing was used to improve result accuracy, and appropriate boundary conditions were implemented to replicate clinical constraints. The von Mises stress criterion was employed to evaluate stress distribution within the implant and surrounding bone.

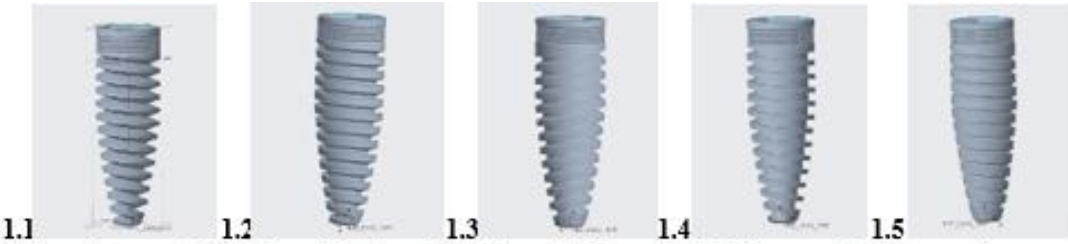


Figure 1 various modelled thread designs 1.1 v-shaped,1.2 squared,1.3 butress, 1.4 reverse buttress, 1.5, 1.6 trapezoidal

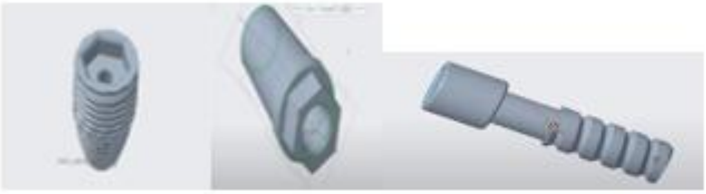


Figure 2 Internal hex configuration

Figure 3 Implant abutment

Figure 4 Abutment Screw

Material	Young's modulus(ε) (MPa)	Poisson's ratio (δ)
Cortical bone	13000	0.30
Cancellous bone	690	0.30
Implant (Titanium)	102000	0.35

Table 1: Mechanical properties of different materials used in the model

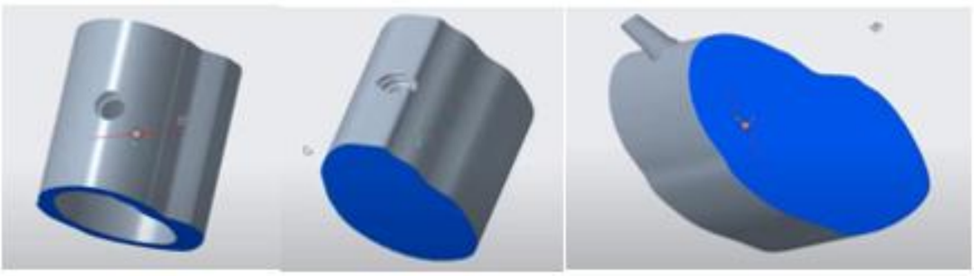


Figure 5

Mould space in cortical bone

Figure 6

Mould space in cancellous bone

Figure 7

Assembled bone and implant 3d model

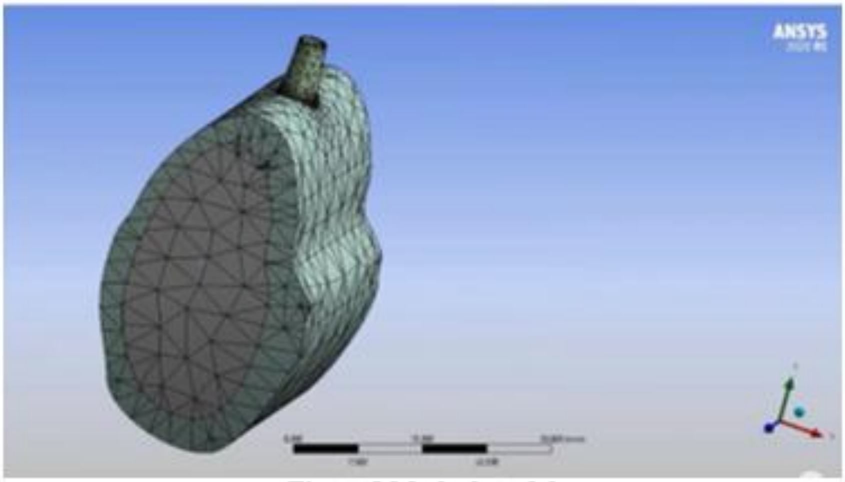


Figure 8 Meshed model

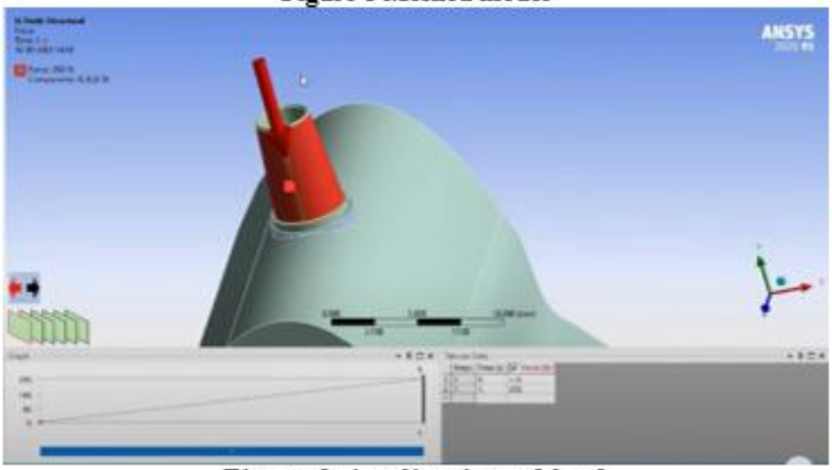
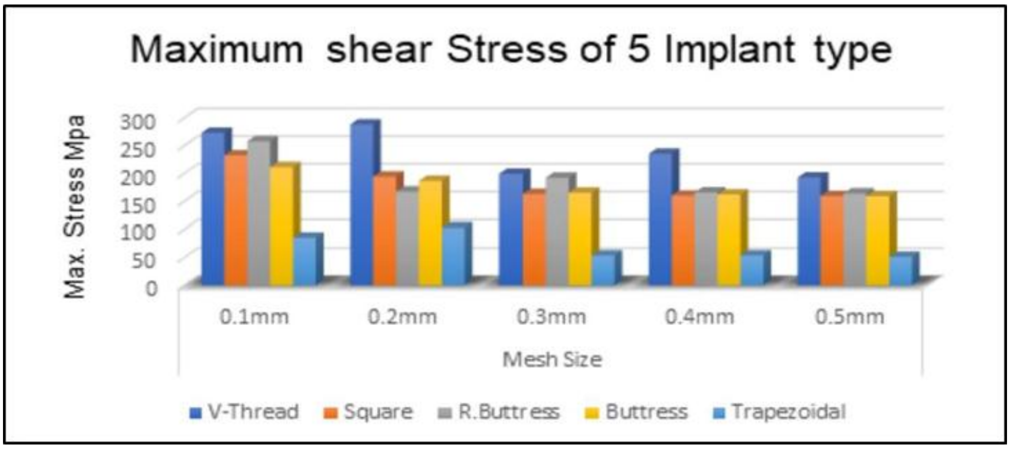


Figure 9 Application of load

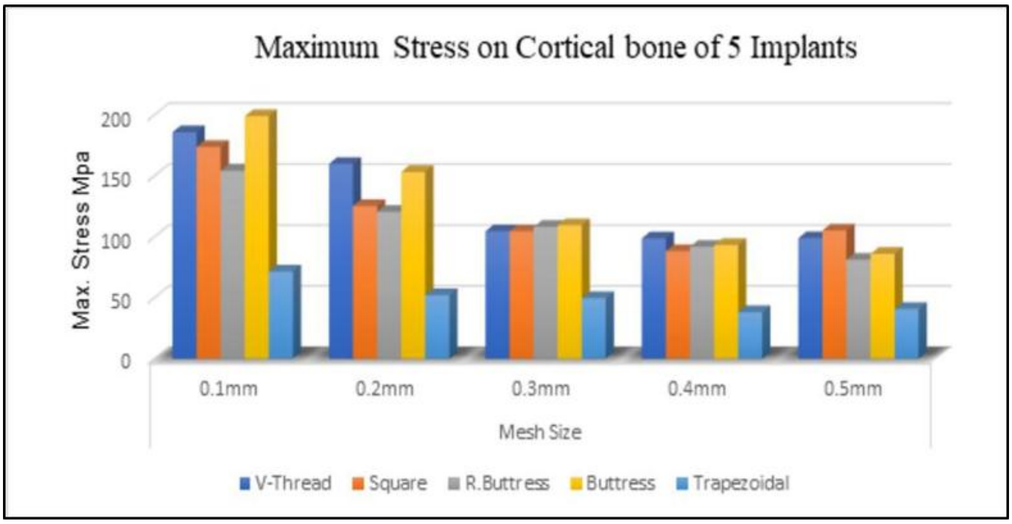
III. Results

The results indicated that thread geometry significantly affects stress distribution within the implant system. The V-shaped thread demonstrated the highest von Mises stress value of 274.3 MPa, suggesting increased stress concentration, particularly at the implant neck. In contrast, the trapezoidal thread recorded the lowest stress at 66.08 MPa, indicating superior stress distribution. These findings suggest that trapezoidal threads may reduce the risk of localized stress concentrations, thereby enhancing bone preservation.



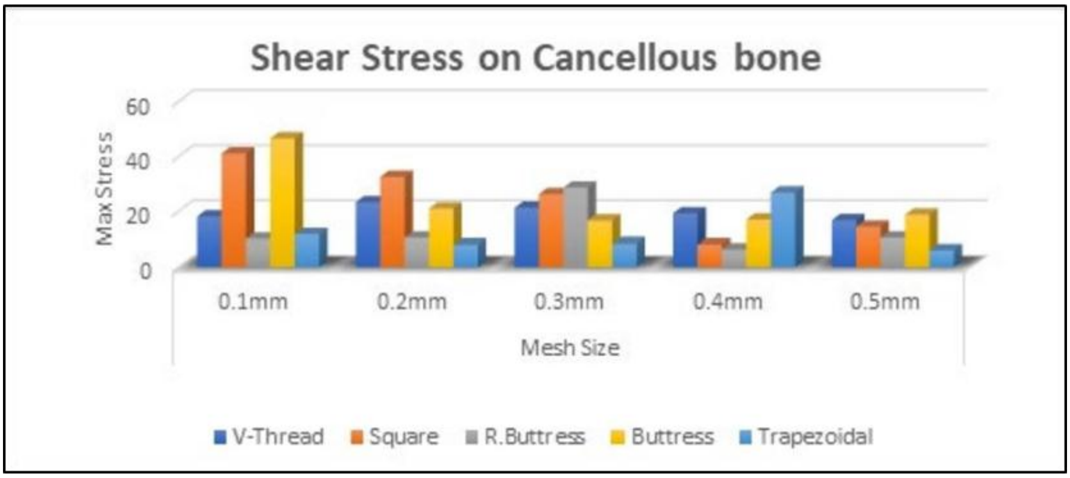
Graph 1: Shear stress of 5 Implant types at different mesh sizes

In cortical bone, buttress threads showed the highest stress levels, measuring 138.1 MPa, whereas trapezoidal threads exhibited the lowest stress at 52.74 MPa.



Graph 2: Max. Shear stress on Cortical bone of 5 Implant types at different mesh sizes

Similarly, in cancellous bone, buttress threads generated the most stress at 41.37 MPa, while trapezoidal threads recorded the least at 10.27 MPa.



Graph 3: Maximum Shear Stress On Cancellous Bone Of 5 Implant Types At Different Mesh Sizes

The results align with the understanding that cortical bone, due to its higher modulus of elasticity, tends to absorb more stress than cancellous bone. Thus, implant designs that distribute stress efficiently in both cortical and cancellous bone are essential for long-term success.

This study supports previous findings that stress concentration near the implant neck can compromise implant stability and increase the likelihood of bone resorption. The uniform stress distribution observed with trapezoidal threads highlights their potential to enhance osseointegration and reduce the risk of implant failure. While FEA provides valuable insights, it is essential to recognize that the results are based on idealized models and may require clinical validation for broader applicability. Future studies involving patient-specific models and dynamic loading conditions are recommended to further explore these findings.

IV. Discussion

Clinical study reports that the predictable success rate of endosseous implants in many systems was above 90% [1]. Success or failure of implant and prosthesis is due to various biomechanical factors such as implant geometry which includes diameter, length, taper, surface topography like thread pitch, type and number, magnitude and direction of masticatory force to implant through abutment and prosthesis. Para functional force also plays a vital role in prognosis of implant treatment. Rangert et al [2] also reported that patients with fractured implants were diagnosed to have parafunctional activities. Petrie and Williams [3] and Meijer et al[4] observed that the length of implant had less influence on the amount of stress levels than diameter did. Apart from the geometrical factors of implant, factors such as surface coating like hydroxyapatite, plasma spray also induce the healing period and osseointegration of bone-implant interface. Finite element method is used to analyze the complicated geometries under static and dynamic load conditions with certain limitations under various simulated environment types. Clinically measurement of stress and strain in bone and implant by using strain gauge is impossible because of ethical reasons. Vertical force with certain magnitude from mastication induces axial forces and bending movements that result in stress gradients in the implant as well as in bone[5]. Finite element method is used to predict and measure the amount of stress in contact area between bone and implant and also in apical part of implant. In this study three dimensional finite element analysis was done rather than two dimensional analysis to visualize the stress distribution in all axes.

Three dimensional model considerations:

In finite element modelling, the structures modelled are simplified, simulated to reflect the reality. In this study the segment of mandible and implant are modelled in three dimensional way. The segments of modelled cortical and cancellous bone are around 26-16mm.

Material properties:

In this study the cortical bone, cancellous bone and implant with abutment were assumed to be linearly elastic, homogenous and isotropic. O'Mahony and Williams [6] reported about the anisotropic properties of cancellous bone. However the cortical and cancellous bone have anisotropic characteristics and regional stiffness variation, although they are modelled isotropically because of non-availability of sufficient scientific data to perform the analysis and difficulty in establishing principle axis geometry.

Loading and constraints:

For this analysis, the constraints at the end of the bone segment and force application on top of the abutment is within the physiological limit. These simplifications result from limitations of the modelling procedure and thus give only a general insight into the tendencies of stress variations under average conditions, without attempting to simulate individual clinical situations. Although this simplification could be expected to bring about quantitative changes in the results, it was not expected to influence them qualitatively. Therefore, it is advisable to focus on qualitative comparison rather than quantitative data from these analyses. Richter et al[10] quantifies the vertical forces applied to dental implants during oral functions. Implants in the molar position that were fixed to a premolar with a prosthesis withstood maximum vertical forces of 60 to 120 N during chewing. Single molars and premolars carried maximum vertical forces of 120 to 150 N. Clenching in centric occlusion caused a load level of approximately 50 N for both natural and artificial abutments [7]. In this study, 250N force was applied as axial load.

Distribution of stress and strain:

This analysis shows that the stress concentration is more at the coronal part of the implant and on the cortical bone, these results coincide with the previous studies and also in invivo and invitro studies. The stress concentration is more in cortical bone because of higher modulus of elasticity $\varepsilon=13000$ Pa which provides more rigidity and thus more capability to withstand higher stress.

Distribution of stress and strain in cortical bone:

In this analysis, the highest stress observed in cortical bone was recorded on the Buttress thread, measuring 138.1 MPa, while the lowest stress was found on the Trapezoidal thread, measuring 52.74 MPa. Therefore, to safeguard cortical bone integrity, minimizing stress on the bone is crucial, making the Trapezoidal thread a preferable option. When determining the ideal stress level for bone health, it's imperative to prioritize minimizing stress magnitude.

According to Mosavar's [8] research findings, square thread configuration demonstrates superiority by exhibiting the lowest stress levels in the cortical bone region around the implant. However, this study overlooks the square thread design and instead evaluates V-shaped, buttress, and reverse thread models. Misch [9] argues that three thread characteristics enhance surface operation, facilitate initial contact, and promote stress distribution at the interface. Analysis of stress patterns indicates that threaded structures mitigate stress near the thread region. Furthermore, threaded implant types offer additional clinical advantages, including improved stabilization and promotion of stress-responsive tissue formation.

Distribution of stress and strain in cancellous bone:

In this analysis, the highest stress observed in cancellous bone was recorded on the Buttress Thread, measuring 41.37 MPa, while the lowest stress is found on the Trapezoidal

Thread, measuring 10.27 MPa. Consequently, following the same principle as for cortical bone, the Trapezoidal thread is preferred due to its ability to minimize stress magnitude. Therefore, the Trapezoidal thread is once again chosen to optimize stress levels in trabecular bone.

Distribution of stress in implant with abutment:

Irrespective of the direction and magnitude of loading, implant with abutment withstands maximum amount of stress compared to any other component of the model. The probable reason could be its high elastic modulus ε=120000 MPa which is nearly nine times the elastic modulus of cortical bone ε=13000 MPa and nearly 173 times the elastic modulus of cancellous bone ε=690 MPa. During axial loading, stress generated within the implant was least as compared to the stress generated during buccolingual and mesiodistal loading.

V. Conclusion

This study demonstrates that implant thread geometry plays a crucial role in stress distribution at the bone-implant interface. Among the designs tested, trapezoidal threads showed the most favorable stress distribution, suggesting they may contribute to improved implant stability and bone health. V-shaped threads, on the other hand, exhibited the highest stress concentrations, potentially increasing the risk of bone resorption. The findings emphasize the need to strengthen the implant neck region to withstand stress concentrations effectively. Ultimately, this research highlights the importance of thread design optimization in improving clinical outcomes and maximizing implant longevity.

References

[1] Meijer Hj, Kuiper Jh, Starmans Fj, Stress Distribution Around Dental Implants: Influence Of Superstructure, Length Of Implants, And Height Of Mandible. J Prosthet Dent 1992;68:96-102.
[2] Rangert B, Krogh Ph, Langer B, Et Al: Bending Overload And Implant Fracture: A Retrospective Clinical Analysis. Int J Oral Maxillofac Implants 1996;11:575.
[3] Dincer Bozkaya, Sinan Muftu, Ali Muftu, Evaluation Of Load Transfer Characteristics Of Five Different Implants In Compact Bone At Different Load Levels By Finite Elements Analysis, J Prosthet Dent 2004;92:523 -30.
[4] Meijer Hj, Kuiper Jh, Starmans Fj, Stress Distribution Around Dental Implants: Influence Of Superstructure, Length Of Implants, And Height Of Mandible. J Prosthet Dent 1992;68:96-102.
[5] Ernst-Jürgen Richter, In Vivo Vertical Forces On Implants, Int J Oral Maxillofac Implants 1995;10:99 – 108.
[6] O'mahony Am, Williams Jl, Katz Jo, Spencer P., Anisotropic Elastic Properties Of Cancellous Bone From A Human Edentulous Mandible, Clin Oral Impl Res 2000: 11: 415– 421.
[7] Ernst-Jürgen Richter, In Vivo Vertical Forces On Implants, Int J Oral Maxillofac Implants 1995;10:99–108.
[8] Rismanchian M, Birang R, Shahmoradi M, Talebi H, Zare Rj. Developing A New Dental Implant Design And Comparing Its Biomechanical Features With Four Designs. Dent Res J (Isfahan). 2010 Summer;7(2):70-5.
[9] Lan Th, Du Jk, Pan Cy, Lee He, Chung Wh. Biomechanical Analysis Of Alveolar Bone Stress Around Implants With Different Thread Designs And Pitches In The Mandibular Molar Area. Clin Oral Investig. 2012 Apr;16(2):363-9.
[10] Richter Ej. In Vivo Vertical Forces On Implants. Int J Oral Maxillofac Implants. 1995 Jan-Feb;10(1):99-10