**How to Cite**:
Farhan-ul-Haq, F.- ul-H., Sahoo, S., Gupta, A. D., Sharma, Y., Sharma, A., & Iftekhar-ul-Haq, I.- ul-H. (2022). To evaluate and compare the stress distribution pattern in the crestal bone around implants with various thread designs : A three dimensional finite element study. *International Journal of Health Sciences*, *6*(S2), 11793–11805. https://doi.org/10.53730/ijhs.v6nS2.8168

# To evaluate and compare the stress distribution pattern in the crestal bone around implants with various thread designs - A three dimensional finite element study

**Dr. Farhan-ul-Haq**
Post Graduate Student final year, department of prosthodontics crown and bridge Shree Bankey Bihari dental college, Ghaziabad (U.P.)

**Dr. Sukant Sahoo**
Head of the Department, Department of prosthodontics crown and bridge Shree Bankey Bihari dental college, Ghaziabad (U.P.)

**Dr. Aakarshan Dayal Gupta**
Reader, Department of prosthodontics crown and bridge Shree Bankey Bihari dental college, Ghaziabad (U.P.)

**Dr. Yukti Sharma**
Senior Lecturer, Department of prosthodontics crown and bridge Shree Bankey Bihari dental college, Ghaziabad (U.P.)

**Dr. Aparana Sharma**
Senior Lecturer, Department of prosthodontics crown and bridge Shree Bankey Bihari dental college, Ghaziabad (U.P.)

**Dr. Iftekhar-ul-Haq**
BDS, Shree Bankey Bihari dental college, Ghaziabad (U.P.)

***Abstract*---**Aim:- To evaluate and compare the effect of thread geometry i.e. V-shape, Square and Buttress under 100 N force on peri-implant bone via Finite Element. Materials & Method:- Three virtual models was prepared having implants with different thread designs:- V– thread design Model I; Square thread design Model II and Buttress thread design Model III. Loading points were marked on the superstructure (crowns virtually fabricated for each of the abutments) and 100N of vertical and oblique forces at an angle of 45° were applied. ANSYS software was used for qualitative and quantitative stress analysis. Maximum von Mises stresses and strain values at the

International Journal of Health Sciences ISSN 2550-6978 E-ISSN 2550-696X © 2022.
*Manuscript submitted: 09 March 2022, Manuscript revised: 18 April 2022, Accepted for publication: 27 May 2022*

implant surface and abutment were recorded. Results:- Highest displacement was seen in square shaped thread designed implants (0.46) followed by buttress shaped and then V-shaped thread design of implants (0.056). The mean stress for V-shaped design implants, square shaped design implants and buttress design implants were 6.12, 6.18 and 6.11 respectively. Highest difference in stress on abutment was 33.84, 32.59 and 20.53 in V shaped, square shaped and buttress shaped designs respectively. Conclusion:- Square shape thread design has biomechanical advantages in terms of reducing stress concentration and microstrain in bone.

***Keywords*---**dental implants, finite element analysis, rehabilitation, stress.

## Introduction

Dental implants have been adopted and implemented in oral rehabilitation in order to replace the lost tooth that occurs due to trauma, damage due to oral infections or birth deformities etc wherein, the most common material used is titanium. It has been documented through various researches that the stability of an implant is dependent on the joint between the bone and the implant.[1] A well-organized and accomplished prosthesis is indispensable to evade extreme and pointless stresses on dental implant components and the bone.[2] Numerous dental implants and materials have been applied to produce restorations like abutments and crowns and hence, the dentists can avail a variety of options to decide on the best implant treatment according to the patient's requirements. There are factors that play a major role in achieving a good primary stability and secondary stability of dental implant.[3]

The success or failure of implant-based rehabilitation is dependent on the stresses that are transported via the implant to the adjacent bone and tissues which is further affected by the macroscopic criteria like implant geometry, structural metallurgy of implant, and variations in the thread designs and few microscopic criteria such as the surface chemistry and microtopography.[4] It is observed that unwarranted stress on the bone may lead to bone resorption and influence the good long-term stability of the implant.[3] The threaded implants display geometric disparities with respect to thread pitch, shape, and depth. It is mandated that threads increases the surface area of the implant, as their geometry helps with regards to the type of force transmitted, hence the use of different thread conformations for different bone qualities have been proposed.[5] The total load transfer at the junction of bone and implant depends on loading type, properties of implant materials, implant geometry (length, diameter and shape), surface structure, bone–implant boundary, and quality and quantity of surrounding bone.[6] It is perceived that dental implants may be subjected to occlusal overloading leading to peri-implant bone loss and failure of the implant. Certain overloading factors causing negative influence on implant durability comprise large cantilevers, parafunctions, improper occlusal designs and premature contacts.[7] Hence, it is imperative to analyze the force transfer at the bone–implant interface for the overall analysis of loading thus determining the success or failure of an implant. Therefore, the implant thread design helps in the stress distribution thus, facilitating suitable clinical decision.[8]

The thread form and configuration are utilized to maximize the preliminary contact, enhance stability, expand the surface area, and provide favourable dissipation of interfacial stresses.[2] Owing to the advancements in technology, analyzing these interfacial stresses applied on the implant and surrounding bone is easier. The procedure known as Finite Element Analysis (FEA) includes 3-D modelling [Finite Element Model (FEM)] of an implant with the surrounding bone and application of loads on this 3-D model discloses the stress dispersal pattern. FEA can deliver a non-damaging structure for measuring the stress produced at innumerable interfaces of analogous or different materials. It helps to replicate complex geometrical shapes, properties and several surface situations.[8]

The finite element (FE) method has been applicable to dental implants in order to forecast stress distribution patterns in the implant bone interface by assessment of several root form implant designs, modeling many clinical situations and prosthesis plans.[5] So it is significant for the practitioner to know the stress factor coordinated with vivid thread designs so as to facilitate in the treatment planning. Hence, this study was undertaken to evaluate and compare the effect of thread geometry i.e. V-shape, Square and Buttress under 100 N force on peri-implant bone via FiniteElement.

## Materials and Method

The present study was conducted by Anamac Design private limited, Gurugram. A modeling software 'DASSAULT SOLIDWORKS-2021' was used which is a solid modeling computer aided design (CAD) and computer aided engineering (CAE) computer program. Once a structure was numerically created and material properties was assigned which was analyzed for stress distributions during force application using finite elements software. The finite element used in this study was 'ANSYS R16.2 Workbench. The stresses were expressed as compressive (which are negative) or tensile (which are positive). The global (x,y,z directional axes) combination of the absolute values squared of all stresses has evolved. A Windows edition-Windows 10 Pro, Processor-AMD Ryzen 3 2200G, RAM: 4.00GB, 64-Bit operating system, x64- based processor, Graphic Card: Radeon Vega Graphics 3.50 GHz computer was used.

## Finite Element Modeling

Construction of geometric model was done by:- (a) Modeling of the bone, (b) Modeling of the implant with Abutment and (c) Modeling of the interface. Three virtual models was prepared (a b c) having implants with different thread designs:- V– thread design Model I (Figure 1); Square thread design Model II (Figure 2) and Buttress thread design Model III (Figure 3). The crown that was taken was of crown Porcelain-fused-to-metal (PFM) with dimensions 12 mm mesiodistal & 9 mm buccolingual. The same size of the crowns was virtually fabricated for each of the abutments. Loading points were marked on the superstructure and 100N of vertical and oblique forces at an angle of 45° were applied.

Figure 1: V-Shaped Thread of Implant

Figure 2: Square shaped Thread of Implant

Figure 3: Buttress shaped Thread of Implant

## Finite element Analysis

Pre-processing included- Mesh Generation, Specifying material properties, Applying boundary conditions, Application of different loads

## Post processing:- Von Mises stress analysis

Finite Element Model (FEM):- Computed tomographic (CT) images of the edentulous mandible at the first molar region of an adult male was used to develop 3 model by means of a modeling software SOLIDWORKS 2021 and rendered by using the keyshot software. Three separate 3D models of a dental implant of 12 mm length 4mm Diameter, was stimulated in solid work 2021. The implants was placed in bone with 2 mm thick cortical bone on both the sides over the cancellous core which was cortical in nature. Three virtual models was prepared (a b c) having implants with different thread designs. (a) V– thread design Model I, (b) Square thread design Model II, (c) Buttress thread design Model III. The crown that was taken was of crown Porcelain-fused-to-metal (PFM) with dimensions 12mm mesiodistal & 9mm buccolingual. The same size of the crowns was virtually fabricated for each of the abutments. Loading points were marked on the superstructure and 100N of vertical and oblique forces at an angle of 45° were applied. The stress and strain were evaluated on the virtual osseointegrated implants and the surrounding bone. The Von-Mises Stress was used to assess bone stress and determine whether any bone damage occurred under a complex loading condition. The stimulated bite force used in this study was 100N, applied to the center of the crown's occlusal surfaces and obliquely at a $45^{o}$ angle relative to the long axis.

FEA Data collection:- ANSYS software was used for qualitative and quantitative stress analysis, considering material properties, meshing and loading. The maximum von Mises stresses (Maximum Equivalent Stress) at the implant surface and abutment and also strain values were reported as qualitative analysis. The stress distribution pattern was also evaluated by color coded diagrams as the quantitative analysis, in which areas with the highest and the lowest stresses were depicted as red and blue, respectively.

## Results

The collected data was analysed with STATA 16 software. The highest displacement was seen in square shaped thread designed implants (0.46) followed by buttress shaped and least displacement was observed in V-shaped thread design of implants (0.056) as was evident from Graph 1. For overall stress, it was observed that the highest stress was observed in square shaped designs of implant (193.4), followed by buttress thread design of implant (166.39) and V-shaped thread design of implant (152.16).

Graph1: Maximum von Miss Stress values (MPa) in Model I, II and III under axial and non- axial loading conditions

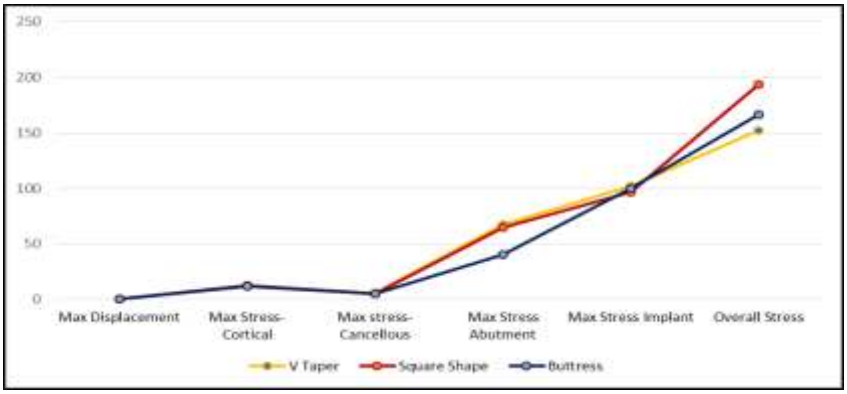

Table 1 shows that there was a significant difference between all the parameters and various types of implants. The highest stress to bone and implant was observed in V-Shaped thread tapered implants and the highest displacement and overall stress was seen in square shaped thread implants.

Table 1: Chi-square test (comparison of different thread designs)

| Groups | V-Shape | Square shape | Butters shape | Row Total |
|---|---|---|---|---|
| Displacement | 2270 | 2300 | 2300 | 6870 |
| Stress bone | 44 | 43 | 44 | 131 |
| Stress on Abutment | 338 | 326 | 205 | 869 |
| Stress on Implant | 510 | 488 | 499 | 1497 |
| Overall Stress | 765 | 975 | 834 | 2574 |
| Column Totals | 3927 | 4132 | 3882 | 11941 |

*Note: The chi-square statistics is 56.1892 with p-value of < 0.00001. The result is significant at $p < .05$.

The displacement group and thread design implants were shown in Table 2 which depicted that V shaped design implants, square shaped design implants and buttress shaped design implants are almost equal. The mean stress for V-shaped design implants, square shape design implants and buttress design implants were 6.12, 6.18 and 6.11 respectively. Highest difference was found in stress on abutment in V shaped design implants (33.84), square shaped design implants (32.59) and buttress shaped design implants (20.53).

Table 2: Mean and Standard deviation comparison across the groups

| | V-SHAPED | SQUARE | BUTTRESS |
|---|---|---|---|
| Displacement | | | |
| Mean | 0.23 | 0.23 | 0.23 |
| SD | 0.15 | 0.15 | 0.15 |
| Stress on Cortical Bone | | | |
| Mean | 6.12 | 6.18 | 6.11 |

| | | | |
|---|---|---|---|
| SD | 12.19 | 12.35 | 12.19 |
| Stress on Cancellous bone | | | |
| Mean | 2.73 | 2.73 | 2.51 |
| SD | 5.50 | 5.08 | 5.45 |
| Stress on Abutment | | | |
| Mean | 33.84 | 32.59 | 20.53 |
| SD | 67.67 | 64.34 | 40.72 |
| Stress on implant | | | |
| Mean | 51.01 | 48.77 | 49.93 |
| SD | 102.55 | 96.73 | 31.85 |
| Overall Stress | | | |
| Mean | 76.49 | 97.54 | 83.39 |
| SD | 48.31 | 61.19 | 52.79 |

Table 3 illustrated T statistics and showed the association between Max Displacement, Max stress-Cortical, Max Stress-Cancellous, Max Stress Abutment, Overall Stress and Max Stress Implant and different thread design which non-significant across the groups.

Table 3: T Student statistics association between the different thread designs implants

| Groups | V- Shaped thread vs. Square Shaped thread | Square shaped thread v s. buttress shaped thread | Buttress Shaped thread vs. V-Shaped thread |
|---|---|---|---|
| T student test | | | |
| Max Displacement | 0.96 | 1.00 | 0.96 |
| Max Stress-Cortical | 0.99 | 0.99 | 1.00 |
| Max stress-Cancellous | 1.00 | 0.93 | 0.99 |
| Max Stress Abutment | 0.60 | 0.62 | 0.60 |
| Overall Stress | 0.40 | 0.59 | 0.76 |
| Max Stress Implant | 0.96 | 0.93 | 0.94 |

For overall stress it was observed that the highest stress was observed in square shape thread design of implant followed by Buttress thread design of implant and V-shaped thread design of implant. For both cortical and cancellous bone the highest stress was found to be in V-shaped thread design of implants followed by buttress shaped thread design of implant and square shaped thread design of implant. This finding suggests that V-shaped thread design of implant transfer stressed to bone the highest. For Abutment, highest stresses were observed in V-shaped thread design of implant

followed by square shaped thread design of implant and least by buttress shaped thread design of implant. These findings suggested that abutments are least susceptible to breakage in buttress shaped thread design of implant and most in V-shaped thread design of implant. For implants highest stress transference was observed in V-shaped thread design of implant followed by buttress shaped thread design of implant and least was observed in square shaped thread design of implant. This finding suggests that square shaped implants are the least likely to break and V-shaped thread design of implant are the most likely to break.

## Axial loading of different thread designs

- → Distribution of stress on the Crestal Cortical Bone:- Under axial and non-axial loading, the maximum stress concentration in Model II was 12.3 MPa compared to 12.19 MPa in Model I and 12.15 Mpa in Model III.
- → Distribution of stress on the Cancellous Bone:- Under loading conditions, the maximum stress on cancellous bone was found to be concentrated in Model I, with a value of 5.46 MPa, as opposed to Model III, which had a stress of 5.4 MPa, and Model II, which had a stress of 5.02 MPa.
- → Distribution of Stress on the implant:- When axial and non-axial loads were applied, the maximum stress on the implant was found to be concentrated in Model I, i.e. 101.82 MPa, as opposed to Model III, which was 99.83 MPa, and Model II, which was 96.69 MPa.
- → Distribution of stress on the Crown:- Under load, the maximum stress on the crown was found to be concentrated in Model I, which was 67.36 MPa, as opposed to Model II, which was 64.46 MPa, and Model III, which was 40.7 MPa.

## Discussion

It has been established though researches that primary implant stability is indispensable for accomplishing expectable osseointegration and the long-standing clinical success of dental implants.[9] The masticatory movements and forces applied includes repetitive cyclic effects that creates pressure on the dental implant components and dispenses the forces to the surrounding bone structure.[10] The notion of load transmission and subsequent stress disseminations at the bone–implant junction have been determined through FEA studies previously, that has been used progressively to envisage the outcome of dental implants.[11,12] Finite element analysis (FEA) is a mathematical process that has acquired immense popularity in biomechanical disciplines including orthopedic, cardiac, and dental mechanics. This is based on the principal of separating the structure into finite numbers of small element being inter-connected with each other at the nodes with 3 degrees of freedom (translated in x, y, & z directions).[13,14] Modern dentistry intends to rehabilitate the form, function and aesthetics of the patients with lost dentition, either with fixed or removable prostheses.[13]

The success of failure of an implant-based rehabilitation is primarily and most importantly reliant on the method and style by which stresses are being transported via the implant to the adjacent bony structures. This further is encouraged by macroscopic criteria such as the geometry and structural metallurgy of implant, and alterations in thread designs and also on the microscopic criteria including the surface chemistry and microtopography. Threads are designed and fabricated in order to make best use of initial contact, augment the surface area, and enable distribution of loads at the bone-implant interface. It becomes necessary to modify the functional surface area per unit length of the implant by changing three geometric thread parameters namely:

thread pitch, thread shape, and thread depth.[4,15] Henceforth, the present study was conducted to evaluate and compare the effect of thread geometry (V-shape, Square and Buttress) on peri-implant bone via Finite Element analysis which showed that the highest stress to bone and implant was exerted through V-shaped thread tapered implants and the highest displacement and overall stress was seen in square shaped thread implants and less in buttress shaped implants. This was found to be in accordance to the study done by Oswal MM et al. (2016)[4].

There are several implant designs consisting of various of implant-abutment connections that improves the manner of stress distribution at the bone-implant interface. Thread design helps in the structural optimization of dental implants and initiates the interaction between the tissue and the implant thus enhancing stability and increasing the surface area of the interaction by decreasing the stresses.[16,17] The various thread shapes that have been used extensively in commercial dental implants are square, triangular or V-shaped, buttress and reverse buttress. The distribution of load is also attributed to the thread angle wherein it is observed that standard V-shaped thread has 10 times greater shear load on bone as compared to the square shaped ones.[18]

Isidor F *et al.* (2006)[19] have shown in their study that implant supported prostheses are subjected to various types of axial and non-axial stresses which includes vertical and horizontal loads and bending movements that result in stress gradients in the implant as well as bone. Tonillo MB *et al.* (2013)[20] have stated from their study that such forces result in resorption of the surrounding alveolar bone and more importantly concentration of stresses in the different parts of the implants which may lead to implant failure. This led to innovations in implant design, which attempts to minimize detrimental effect of the normal masticatory loads, and hence increasing the longevity of implant supported prostheses.[20]

In a study by Zhang G *et al.* (2016)[21], 12 various types of dental implants along with their adjacent bone tissues with numerous structured abutments were analyzed for their stress distribution with respect to characteristics, implant threads, and healing methods with diverse amount of rigorous loading. The authors observed that the influence of stress on the implant threads was not substantial; nevertheless, it was obvious on the cancellous bone. They concluded that a dental implant system typified by a straight abutment, rectangular tooth, and non-submerged curing process is the best design that can be accepted for appropriate treatment. Narendra Kumar U *et al.* (2018)[18] analysed the effect of varying thread angles in a V-shaped thread on the stress distribution and total deformation of the bone around the implant and showed that 60-degree thread angle has a distinctly varying von-Mises stress dispersal as compared to the angles 20, 30 and 45 degrees. They suggested that stress dissipation is impacted the thread shape, structural design and the type of bone-implant interface.

According to Udomsawat C *et al.* (2018)[22], various designs of implant could affect different methods of stress generation during implant insertion procedure. It is important to note the implant surface contact area that creates an impact on the features of stress during implantation. A specific magnitude of stress, produced in the adjacent bone during implant insertion, may impact the bone healing around dental implants and also alter the final implant stability. In a similar study as the present one, Paracchini L *et al.* (2020)[23] used Finite Element Analysis (FEA) to assess the stress distribution in cortical and cancellous bone surrounding by using standard cylindric implant, cylindrical neck and V-shaped threads, and a new conical implant, reverse conical neck and with a "nest shape" thread design. They suggested that the novel design of the dental implant could reduce the transmission of stress to the peri-implant cortical bone. Alemayehu DB *et al.* (2021)[24] conducted a study and showed that the thread design and occlusion loading rate have and imperative influence on the stress distribution and deformation of the dental implant and surrounding bone structure. They showed that square thread implants had more favourable stress and strain distribution thus improving bone remodeling. The present study showed analogous results to previous research conducted by Sharma C *et al.* (2020)[25] who showed that least amount of von mises Stress concentration was obtained by tapered implant body with reverse buttress thread design under axial load 100N and tapered implant body with V-thread under buccolingual load of 50N at cortical bone thus denoting bone conservation.

The human mandible has a very unique and complicated geometrical structure that makes it very tough to attain a systematics and logical resolution. Thus, the application of numerical method like FEA is obligatory. Finite element analysis is applied to evaluate the stress along implant-bone interface, since *in vivo* strain gauge measurements cannot be done inside the bone, for ethical reasons.[26] This methodology permit us to safely forecast the distribution of stress at bone-implant interface. 3-D Finite Element Method is suggested to be used precisely in asymmetric bones like the mandible, to evaluate the stress/strain in bone-implant interface.[27] Clinical Consideration: The present Finite Element study demonstrated that axial and non-axial loading shows the least stress in square shape thread design which is 12.3 Mpa on cortical bone, 5.02 Mpa on cancellous bone, 64.46 Mpa on abutment and 96.69 Mpa on implant.

Limitation of Finite Element Modelling: The vital anisotropic tissues were considered isotropic, subsequent to which static loads that varied from dynamic loading especially during function, were applied. This methodology is an extremely correct and exact technique of analyzing the structure. FEA is reliant on scientific arithmetic calculations that simulate the structure that surrounds the living tissues, although, living tissues are not forced by predefined parameters and values and since biology and physics are incompatible. As a result, while FEA provides a solid theoretical foundation for understanding the behavior of a structure in a given environment, it should not be used in isolation. FEA should be used in actual experimental techniques and clinical trials to determine the true nature of the biologic system.

## Conclusion

The stress concentration in the crestal bone in the square shape thread design is optimal when compared to the V-shape thread design and the Buttress shape thread design, based on a comparative evolution of the stress distribution pattern around the implants supporting restoration of different thread designs. The stress value of the V-shaped thread design is greatest on the abutment collar. Furthermore, the stress value for buttress shape thread design and square shape thread design is above the abutment's neck. According to the findings of this study, square thread designs and buttress thread designs have a higher risk of abutment fracture. The V-shaped thread design of the implant exhibits the least displacement under load when compared to the buttress thread design and the square thread design. As a result, square shaped thread and buttress thread designs have a higher risk of cortical bone fracture. All of the models showed increased stress under axial and non-axial loading, indicating that a prosthesis should be designed with the minimum amount of load possible. Hence, it can be concluded that square shape thread design has biomechanical advantages in terms of reducing stress concentration and microstrain in bone.

## References

1. Jadhav RS, Dhatrak PN, Kulloli VK. Mechanical design and stress analysis threaded dental implants along the implant bone interface IOSR J Mech Civi Eng. 2016:9-14.
2. Meriç G, Erkmen E, Kurt A, Eser A, Çelik G. Biomechanical evaluation of a fiber-reinforced composite prosthesis supported by implants with and without a microthread collar design. Journal of Dental Sciences.2010;5(4):201–08.
3. Calì M, Zanetti EM, Oliveri SM, Asero R, Ciaramella S, Martorelli M, Bignardi C. Influence of thread shape and inclination on the biomechanical behaviour of plateau implant systems. Dental materials: official publication of the Academy of Dental Materials.2018;34(3*)*:460-69.
4. Oswal MM, Amasi UN, Oswal MS, Bhagat AS. Influence of three different implant thread designs on stress distribution: A three-dimensional finite element analysis. J Indian Prosthodont Soc. 2016 Oct-Dec;16(4):359-365.
5. Atwood DA. Some clinical factors related to rate of resorption of residual ridges. 1962. J Prosthet Dent. 2001 Aug;86(2):119-25.
6. El-Anwar MI, Yousief SA, Soliman TA, Saleh MM, Omar WS. A finite element study on stress distribution of two different attachment designs under implant supported overdenture. Saudi Dent J. 2015 Oct;27(4):201-7.
7. Ali B, Chikh EBO, Meddah HM, Merdji A, Bouiadjra BB. Effects overloading in mastication on the mechanical behaviour dental implants. Materials Design. 2013;47:210-17.
8. Achour T, Merdji A, Bouiadjra B B, Serier B, Djebbar N. Stress distribution in dental implant with elastomeric stress barrier. Materials & Design. 2011;32:282-90.
9. Yang J, Xiang HJ. A three-dimensional finite element study on the biomechanical behavior of an FGBM dental implant in surrounding bone. J Biomech. 2007;40(11):2377-85.
10. Assunçoa WG, Gomes EA, Barao VA, Delben JA, Tabata LF, de Sousa EA. Effect of superstructure materials and misfit on stress distribution in a single implant-supported prosthesis: a finite element analysis. J Craniofac Surg. 2010 May;21(3):689-95.

11. 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 Apr;29(2):100-5.
12. Herekar MG, Patil VN, Mulani SS, Sethi M, Padhye O. The influence of thread geometry on biomechanical load transfer to bone: A finite element analysis comparing two implant thread designs. Dent Res J (Isfahan). 2014 Jul;11(4):489-94.
13. Carvalho MA, Sotto-Maior BS, Del Bel Cury AA, Pessanha Henriques GE. Effect of platform connection and abutment material on stress distribution in single anterior implant-supported restorations: a nonlinear 3-dimensional finite element analysis. J Prosthet Dent. 2014 Nov;112(5):1096-102.
14. Alvarez-Arenal A, Brizuela-Velasco A, DeLlanos-Lanchares H, Gonzalez-Gonzalez I. Should oral implants be splinted in a mandibular implant-supported fixed complete denture? A 3-dimensional-model finite element analysis. J Prosthet Dent. 2014 Sep;112(3):508-14.
15. Misch CE. A scientific rationale for dental implant design. In: Dental Implant Prosthetics. 1st ed. St. Louis, MO: Mosby; 2004. p. 322-43.
16. Ivanoff CJ, Gröndahl K, Sennerby L, Bergström C, Lekholm U. Influence of variations in implant diameters: a 3- to 5-year retrospective clinical report. Int J Oral Maxillofac Implants. 1999 Mar-Apr;14(2):173-80.
17. Brunski JB. Biomechanical considerations in dental implant design. Int J Oral Implantol. 1988;5(1):31-4.
18. NarendraKumar U, Mathew AT, Iyer N, Rahman F, Manjubala I. A 3D Finite Element Analysis Of Dental Implants With Varying Thread Angles. Materials Today: Proceedings.2018;5(5):11900–05.
19. Isidor F. Influence of forces on peri-implant bone. Clin Oral Implants Res. 2006 Oct;17 Suppl 2:8-18.
20. Toniollo MB, Macedo AP, Rodrigues RC, Ribeiro RF, Mattos Mda G. Three-dimensional finite element analysis of the stress distribution on morse taper implants surface. J Prosthodont Res. 2013 Jul;57(3):206-12.
21. Zhang G, Yuan H, Chen X, Wang W, Chen J, Liang J, Zhang P. A Three-Dimensional Finite Element Study on the Biomechanical Simulation of Various Structured Dental Implants and Their Surrounding Bone Tissues. Int J Dent. 2016;2016:4867402.
22. Udomsawat C, Rungsiyakull P, Rungsiyakull C, Khongkhunthian P. Comparative study of stress characteristics in surrounding bone during insertion of dental implants of three different thread designs: A three-dimensional dynamic finite element study. Clin Exp Dent Res. 2018 Dec 26;5(1):26-37.
23. Paracchini L, Barbieri C, Redaelli M, Di Croce D, Vincenzi C, Guarnieri R. Finite Element Analysis of a New Dental Implant Design Optimized for the Desirable Stress Distribution in the Surrounding Bone Region. Prosthesis. 2020; 2(3):225-236.
24. Alemayehu DB, Jeng YR. Three-Dimensional Finite Element Investigation into Effects of Implant Thread Design and Loading Rate on Stress Distribution in Dental Implants and Anisotropic Bone. Materials (Basel). 2021 Nov 18;14(22):6974.
25. Sharma, C., Kalra, T., Kumar, M., Bansal, A., & Chawla, A. K. (2020). To Evaluate the Influence of Different Implant Thread Designs on Stress Distribution of Osseointegrated Implant: A Three-Dimensional Finite-Element

Analysis Study–An In Vitro Study. Dent J Adv Stud. 2020;8:9-16.

26. Maló P, Nobre Mde A, Petersson U, Wigren S. A pilot study of complete edentulous rehabilitation with immediate function using a new implant design: case series. Clin Implant Dent Relat Res. 2006;8(4):223-32.
27. Jivraj S, Chee W, Corrado P. Treatment planning of the edentulous maxilla. Br Dent J. 2006 Sep 9;201(5):261-79.

Table 1: Chi-square test (comparison of different thread designs)
Table 2: Mean and Standard deviation comparison across the groups
Table 3: T Student statistics association between the different thread designs implants

Graph1: Maximum von Miss Stress values (MPa) in Model I, II and III under axial and non- axial loading conditions