Revue des Composites et des Matériaux Avancés-Journal of Composite and Advanced Materials
Vol. 34, No. 5, October, 2024, pp. 565-571
Journal homepage: http://iieta.org/journals/rcma

Finite Element Analysis of Stress Distribution in 30% CFR PEEK Implant with Varying Thread Designs

Jackson Irudhayam S*, Hariram Venkatesan

Department of Mechanical Engineering, Hindustan Institute of Technology and Science, Chennai 603103, India

Corresponding Author Email: jackson.irudhayam@gmail.com

https://doi.org/10.18280/rcma.340504

Received: 11 June 2024
Revised: 1 August 2024
Accepted: 23 August 2024
Available online: 31 October 2024

Keywords:
biomaterials, CFR PEEK, FEA, implant, thread design

ABSTRACT

The surgical effectiveness of dental implants is entirely reliant on the biomechanical behaviour of the implant material, the pitch of the implant, and the kind of implant thread design. An enhancement in the implant's efficacy may be achieved by the selection of the appropriate settings. This research aimed to examine the stress distribution in cortical and cancelous bone by using various kinds of implant thread designs with a pitch value of 0.8mm and a 30% carbon fiber reinforced plastic (30% CFR-PEEK). 30% CFR-PEEK, which stands for carbon fiber reinforced (30%) poly ether ether ketone, has shown its efficacy as a viable alternative to titanium in orthopaedic applications. This research investigated four implants with distinct thread designs, namely V-thread, square thread, buttress and reverse buttress. The mandibular molar area, consisting of both cortical and cancellous bone, is often regarded as isotropic and homogenous. The implant was created using SolidWorks software, and a vertical force of 200 N was exerted. The stress values were determined by the use of the Finite Element Analysis (FEA) technique using ANSYS software. When the bone implant interface was taken into consideration, the von Mises stresses were found to be at their lowest at the cancellous bone for the V thread design, whereas the square thread design showed the highest stresses at the cancellous bone under 200N axial load. When cortical bone was taken into consideration, the square thread design seemed to have the lowest von Mises stresses, whereas the V thread design had the highest stress value. Therefore, the findings that were obtained may be employed therapeutically for the purpose of making a suitable selection of implant thread design in order to achieve a predicted level of success with implant treatment, within the constraints of this research.

1. INTRODUCTION

Dental implants have been more popular as a means of restoring lost teeth in recent years. An "implant" is the process of inserting inanimate components into live tissue. However, notwithstanding the findings of certain research that reported an implant survival rate ranging from 78% to 100% [1]. Further investigation has shown that several aspects, including the features of the implant surface, the configuration of the threads, the integrity of the bone, and the material properties of the implant, might potentially lead to failures in certain dental restorations. There is a risk of bone disintegration in the regions around dental implants, even after the implants have been firmly integrated with the surrounding bone tissue. This danger may be produced by bacterial infection or excessive masticatory pressure [2]. The recognition of bone quality and implant thread design as crucial factors in ensuring long-term viability has been accepted. Extensive studies in this domain have shown that bone loss occurs at a higher rate when the bone quality deteriorates due to thinner cortical and low-density cancellous bones, as well as increased thread pitch. These factors also contribute to the reduced ability of the bones to withstand the pressures caused by dental loads [3]. Given this assumption, it is crucial to assess the impact of implant thread patterns on stress distribution in the bone when subjected to a load, in order to make a suitable therapeutic decision. Therefore, this work aims to ascertain the impact of four distinct thread patterns on the arrangement of stress in the bone by using a three-dimensional (3D) Finite Element Analysis (FEA).

The Finite Element Method (FEM) stands out as a powerful computational technique that is widely employed to analyze and tackle complex biomechanical challenges, especially within the fields of dentistry and orthopedics. Its versatility and precision make it an invaluable tool for researchers and practitioners alike, enabling them to simulate and predict the behaviour of biological structures under various conditions. By breaking down intricate systems into smaller, manageable elements, FEM allows for detailed examination and understanding of stress distributions, deformations, and other critical factors that influence the performance of dental and orthopedic applications. This method not only enhances the accuracy of biomechanical assessments but also contributes to the development of innovative solutions tailored to meet the specific needs of patients. As the demand for advanced medical technologies continues to grow, the significance of FEM in driving progress in these disciplines cannot be overstated. Its ability to provide insights into the mechanical properties of biological tissues and materials paves the way for improved treatment strategies and outcomes in both dentistry and orthopedics. In summary, the Finite Element Method serves as a cornerstone in the realm of biomechanical analysis, offering a sophisticated framework for understanding and addressing the complexities inherent in the human body. The Finite Element Method (FEM) has proven to be a powerful tool for modelling intricate structures and simulating how they respond to various forces. Its effectiveness is particularly notable when exploring processes that are difficult or impossible to study in living organisms or controlled laboratory settings, often due to ethical considerations, practical limitations, or biological complexities. This method allows researchers to gain insights into scenarios that would otherwise be challenging to analyze, providing a valuable alternative for understanding complex interactions and behaviors within materials and systems [4]. The Finite Element Method (FEM) functions by breaking down a complex structure into smaller, more manageable components. Each of these components undergoes its own specific calculations, which allows for precise assessments of stress, strain, and deformation within the overall structure. This method has gained significant traction in improving both the design and functionality of dental implants.

The accurate forecasting of stress distribution and deformation when subjected to various loads is essential for ensuring the stability and overall success of the implants. In the realm of scientific literature, the Finite Element Method (FEM) serves as a crucial tool for evaluating the stress and deformation properties of dental implants. This method plays a significant role in enhancing the performance and reliability of these implants, ensuring they meet the necessary standards for effectiveness. By employing FEM, researchers can simulate various conditions and analyze how dental implants respond under different stresses, leading to better design and material choices. This analytical approach not only aids in understanding the mechanical behaviour of implants but also contributes to advancements in dental technology, ultimately benefiting patient outcomes. The application of FEM in this field underscores its importance in the continuous pursuit of innovation and improvement in dental implantology. Recent studies have employed the Finite Element Method (FEM) to explore and simulate a variety of thread configurations utilized in dental implants. This innovative approach allows for a detailed analysis of how different thread designs can impact the performance and stability of dental implants. By examining four distinct thread configurations, researchers aim to gain insights into the mechanical behaviour and potential advantages of each design. The findings from these simulations could lead to more effective implant designs that better meet the needs of patients, ensuring greater longevity and success rates in dental treatments. As the field of dental implantology continues to evolve, the chosen thread shapes were established based on their influence on the transfer of force from the implant to the bone. This factor plays a crucial role in both the stability of the implant and the process of osseointegration, which refers to the adherence of bone tissue to the surface of the implant [5]. The Finite Element Method (FEM) analysis successfully mirrored the performance of these thread designs when subjected to static axial stresses.

At the same time, it incorporated different levels of osseointegration to simulate a range of clinical scenarios. The findings revealed that the initial thread of the implant, along with the cervical cortical bone—this is the dense bone situated adjacent to the gumline—experienced the highest levels of stress. The concentration of stress in this context can result in bone resorption and the loosening of implants as time progresses. This highlights the critical importance of carefully designed threads to alleviate excessive stress and ensure the longevity and stability of the implant. However, FEM models come with inherent constraints, particularly the assumption of static loads. This assumption may not accurately represent the dynamic loading conditions that dental implants face in real-life scenarios, such as those caused by chewing or the forces of occlusion. Consequently, although finite element modeling provides valuable information regarding stress distributions and the effectiveness of implants, it is crucial to incorporate dynamic loading conditions in future studies to achieve results that are relevant to clinical practice [6]. Previous investigations mostly focused on examining the effects of stationary forces exerted on a particular area of the implant's interface. Nevertheless, in a Finite Element Method (FEM) simulation, it is crucial that the force applied to the surface of the head during chewing be periodic, consistent, and dynamic over time to properly depict the movement of the lower jaw. The implant thread significantly affects the flow of stress throughout the implant and the healing process of the surrounding bone. To provide a more even distribution of load and minimize the intensity of stress at the interface between the bone and implant, it is necessary to create a meticulously engineered thread pattern [5, 6]. Dental implants provide a range of thread designs, including square, triangular, buttress, reverse buttress, and trapezoidal options. The face angle, thread width, thread pitch, and other design factors of the thread significantly impact the kind of force produced at the implant's surface and the effectiveness of load transmission to the bone. The characteristics of the bone material have a significant impact on the flow of stress inside dental implant systems.

However, when the reduction in complexity is contrasted to the actual stress levels, it leads to greatly overestimated projections of stress for the bone around the implant [7]. Consequently, the outcomes of the simulation have limited usefulness in practical clinical settings. The elastic modulus of polyetheretherketone (PEEK), which exhibits robust mechanical performance, is around 3-5GPa, much lower than the elastic modulus of bone (13.7GPa). Carbon fiber-reinforced polyetheretherketone, a variant of PEEK, has an elastic modulus of 18GPa, similar to that of bone. Researchers have postulated that the bone pressures induced by titanium and CFR-PEEK implants are comparable. Therefore, it may be used as a preferred substance for implants and their suprastructures in order to minimize the distribution of stress in the bone [8].The aim of this research is to get a deeper comprehension of the biomechanical characteristics shown by four 30% CFR PEEK dental implants with different thread patterns under a loading rate of 200 N. Additionally, the research seeks to examine the influence of these patterns on the distribution of stress throughout the implant and the surrounding bone tissue. Since this research is a study in vitro, it may not accurately replicate the clinical setting. The current research did not seek to replicate specific clinical scenarios; instead, it merely examined a section of the jaw with an implant, providing a basic understanding of the typical patterns of stress fluctuations under ordinary settings.

2. MATERIALS AND METHODS

2.1 CBCT scan and selection of thread type

Cone beam computed tomography (CBCT) was performed on March 20, 2022, to scan the maxillary and mandibular regions of the jawbone (Figure 1). Future research will take into account factors such as bone type, CBCT imaging data, required implant size, and observed bone state. The following data is derived from the CBCT scan. Specimens of D2 bone type, with a cortical bone thickness of 2mm, a length of 13mm, and a diameter of 4.2mm, were collected. Details are shown in Figure 1. Four distinct threads were simulated inside the implant to evaluate the level of tension.

The thread type includes square, V thread, buttress, and reverse buttress. SolidWorks is used for modelling the implant, cortical bone, and cancellous bones, which are further examined via Ansys software. An assessment is conducted on the particular characteristics of the four different thread-form combinations to optimize stress distribution for the implant.

2.2 Material selection

Potential uses of PEEK, the most prevalent PAEK polymer, in dental implants include its ability to withstand high temperatures, its stiffness, and its resistance to corrosion. While it has been used in implant bodies, abutments, and superstructures, its use in the mandible is still in the early stages [8]. PEEK is an excellent choice for dental implant components, including abutments and frames, since it has a lower modulus of elasticity compared to cortical bone, titanium, and ceramic materials. PEEK composites, including CFR-PEEK and GFR-PEEK, have been developed with impressive elastic moduli of up to 18GPa for 30% CFR-PEEK and 12GPa for GFR-PEEK. The medical implant industry is attracted to CFR-PEEK because of its versatility, compatibility with existing imaging equipment, exceptional mechanical properties, and biocompatibility [9].

2.3 Model generation

The research used a mandibular second molar model created from specifications derived from Cone Beam Computed Tomography (CBCT) scan data, guaranteeing great anatomical precision consistent with actual morphology. A three-dimensional finite element analysis model of an intact mandibular second molar and its supporting structures was developed using SolidWorks, a computer-aided design program. This model included the fundamental geometry required to replicate the intricate interplay among the implant, bone, and adjacent anatomical components.

Upon completion, the CAD models were exported in the Standard for the Exchange of Product Data (STEP) file format, a prevalent standard for exchanging 3D data across software, facilitating seamless input into the pre-processing program, ANSYS Workbench. ANSYS Workbench is a robust simulation tool often used in Finite Element Analysis (FEA) for configuring and examining intricate engineering and biomechanical models. Four 3D finite element analysis models were created, each including different thread geometries, to evaluate the biomechanical performance of the implant under simulated settings.

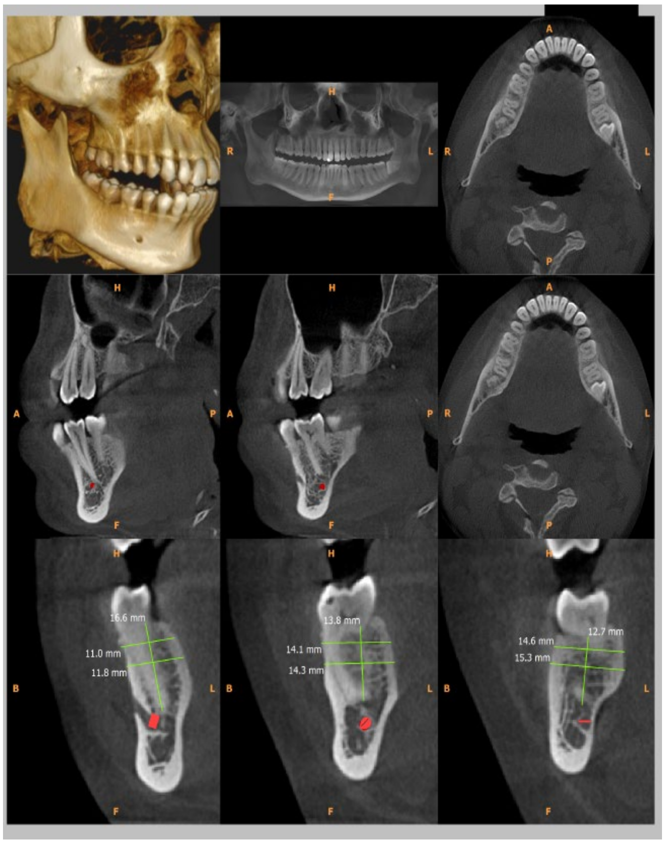

Figure 1. CBCT scan data

The study's implant model was developed according to literature guidelines, using carbon fiber-reinforced polyetheretherketone (CFR-PEEK) as the principal biomaterial. CFR-PEEK was chosen for its compatibility with bone characteristics, since its elasticity more closely resembles that of bone than standard materials such as titanium. The implant has a length of 13mm, a diameter of 4.2mm, and four unique thread configurations with a pitch of 0.8mm. These threads were meticulously designed to examine the impact of various designs on stress distribution and stability in the adjacent bone tissue under loading circumstances. Figure 2 illustrates the implant-bone model used in this study, highlighting the intricate structural configuration.

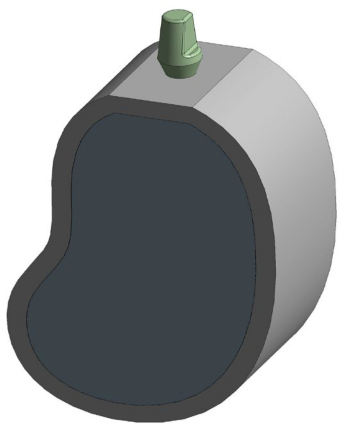

Figure 2. Implant- Bone model

2.4 Material properties and load condition

The implant's top was subjected to a 200 N axial load to simulate parafunctional stress. Load values are taken from literature for reference. To monitor the distribution of stresses in implant, cortical, and cancellous bone, Von Mises stress analysis was performed. The Young's modulus and Poisson ratio of the 30% CFR PEEK, cortical bone, and cancellous bone materials are obtained from the literature.

Table 1. Young's modulus of different types of materials

Materials	Young's Modulus (GPa)	Poisson Ratio
Cortical bone	13.7	0.3
Dense cancellous bone Type II, Type III	1.37	0.3
Low-dense cancellation bone	0.231	0.3
30% CFR PEEK	18	0.39

Table 2. Details of node and element

Material	Node	Element
V thread	669006	468103
Square thread	665514	465490
Buttress	664862	465250
Reverse buttress	664610	465137

The components of each model have been assumed to be homogeneous and isotropic. Material properties are given in Table 1. Node and element details given in Table 2.

3. RESULTS

3.1 Analysis of stress behavior in bone

The outcomes of this research were rigorously evaluated among four distinct implant thread types, specifically focusing on the performance of a 30% carbon fiber-reinforced polyetheretherketone (CFR-PEEK) material. Specifically, the purpose of the research was to determine the extent to which various thread patterns influence the distribution of stress throughout the bone structure, especially under settings that simulate the quality of type II bone. Detailed information on the measured stress values that correlate to each implant thread type and material composition can be found in Tables 3 and 4, respectively. The information included in these tables is very useful in gaining an understanding of how stress dynamics may be affected by thread shape and material selection, which eventually contributes to the development of dental implants that are more efficient.

3.2 Vertical load

When 0.8mm pitch square threads were used in the implant, the maximum Von-Mises stresses were seen in the cancellous bone. The figure depicts the distribution of stress in cancellous bone induced by different thread kinds. When compared to various thread types, it has been observed that a V thread type implant with a 0.8mm pitch and 30% CFR PEEK material achieves the lowest von-Mises stress. Table 3 and Table 4 show Von-Mises stress in Cancellous and cortical bone.

Table 3. Cancellous bone-Von-Mises stress

Type II Bone Quality, Single Thread 0.8mm, Vertical Load			
Cancellous Bone	**MIN (MPa)**	**MAX (MPa)**	**AVE (MPa)**
V thread	2.4818e-002	8.5696	0.248111
Square thread	1.2805e-002	27.109	0.21387
Buttress	3.8559e-003	11.733	0.12332
Rev buttress	2.4612e-002	11.148	0.2291

Table 4. Cortical bone-Von-Mises stress

Type II Bone Quality, Single Thread 0.8mm, Vertical Load			
Cortical Bone	**MIN (MPa)**	**MAX (MPa)**	**AVE (MPa)**
V thread	1.9174e-002	23.321	0.19981
Square thread	1.2459e-005	10.01	0.13856
Buttress	3.0294e-003	12.623	0.20441
Rev buttress	1.8993e-002	18.606	0.20784

When the bone implant interface and maximum stress values were taken into consideration, the von Mises stresses were found to be at their lowest at the cancellous bone for the V thread design, whereas the square thread design showed the highest stresses at the cancellous bone under 200 N axial load. When cortical bone was taken into consideration, the square thread design seemed to have the lowest von Mises stresses, whereas the V thread design had the highest stress value. Figure 3 and Figure 4 show the Results of Von-Mises Stress in Cancellous Bone and Cortical Bone.

4. DISCUSSION

Finite Element Analysis (FEA) in dental implant research provides critical insights into the distribution of stress across various implant thread designs inside the bone structure. Finite Element Analysis (FEA) is a computer instrument especially adept in assessing and illustrating stress distribution patterns, which is essential in dental implantology. This research examines four specific implant thread configurations—square, V thread, buttress, and reverse buttress—evaluating their impact on stress distribution in cortical bone, particularly using 30% carbon fiber-reinforced polyetheretherketone (CFR-PEEK) for the implants. A key discovery was the resemblance in von Mises stress distribution throughout the four thread designs, with only negligible variances seen among them. Von Mises stress, a prevalent measure in engineering and biomechanics, aids in forecasting possible breakdown locations in a material subjected to complicated loading circumstances. The study's results indicated that thread geometry had no significant effect on stress distribution across the bone structure, irrespective of the material used. This corresponds with prior research comparing square and V thread configurations, which also identified low impact of both designs on stress distributions in cortical bone [10-13]. The incorporation of buttress and reverse buttress designs in this study offered a more comprehensive foundation for comparison, revealing that both thread types also exhibited no significant changes in stress distribution. This result suggests that thread morphology may not significantly influence stress transmission to bone, a crucial insight for researchers developing implants [12].

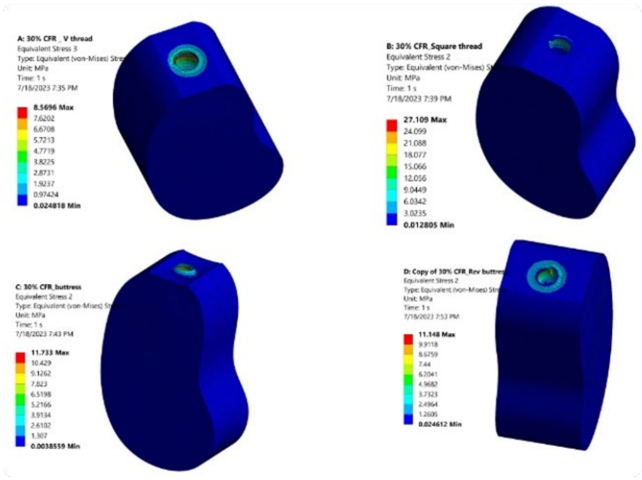

Figure 3. Results of cancellous bone-Von-Mises stress

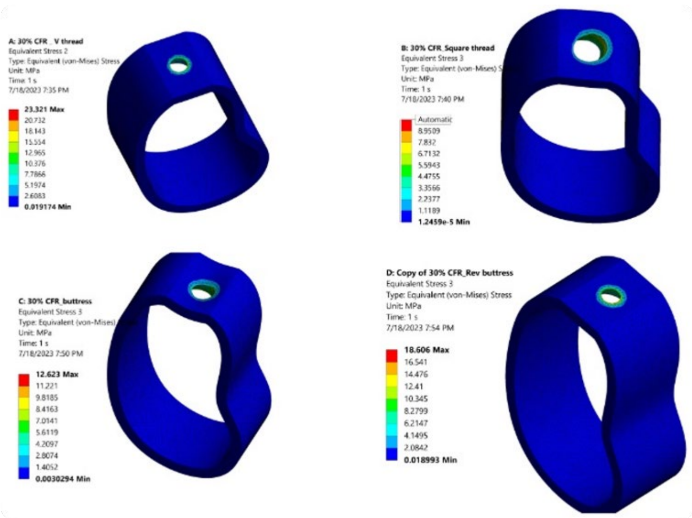

Figure 4. Results of cortical bone-Von-Mises stress

The research noted that square thread designs often provide a more equal stress distribution than V-thread varieties, due to their structural characteristics that allow a greater contact area with the bone. An equitable stress distribution is essential in implantology, as it mitigates the possibility of elevated stress concentrations that may result in localized bone resorption or implant failure [12]. The incorporation of 30% CFR-PEEK implants enabled an assessment of the material's influence on stress distribution patterns. 30% CFR-PEEK is recognized for its compatibility with bone owing to its flexibility, which closely resembles that of genuine bone tissue [7]. In contrast to titanium, a conventional implant material, 30% CFR-PEEK reduces the stress-shielding effect, which may result in bone loss over time due to under-stimulation of the surrounding bone. CFR-PEEK presents a compelling alternative; yet, the investigation revealed that analogous von Mises stress patterns manifested across various thread configurations, suggesting that while the material influences stress transmission to bone, thread form is a minor consideration.

The study indicated that stress concentrations are most pronounced in the cortical bone, especially around the implant neck and next to the first thread. Cortical bone is often more resilient to compressive stresses than cancellous bone, the porous inner layer of bone. Previous research has shown that the neck area, next to the original implant threads, endures the highest stress levels owing to the mechanical load transfer mechanism, where pressures exerted on the implant are dispersed throughout the bone, peaking at the first contact site. Excessive compressive stress in this area might result in bone resorption, undermining implant durability. The results of this FEA analysis support the concept that designs producing greater compressive stresses, such as square and buttress threads, may facilitate bone health by stimulating osteoblastic activity, which is crucial for bone regeneration [11-14].

In addition to thread form, thread pitch and depth also affect stress distribution. The thread pitch, defined as the spacing between consecutive threads, influences the density of thread packing and alters the manner in which stress is transmitted to the bone [8]. A reduced pitch may increase the number of threads in contact with the bone, thereby enhancing load distribution but simultaneously posing a higher risk of stress concentration. Thread depth, which denotes the extent of thread incision, also affects force transfer to the bone [9, 11]. This study mainly focused on thread form; however, further research might investigate these characteristics to enhance comprehension of their collective influence on stress distribution. Literature indicates that deeper threads may enhance implant stability by improving mechanical interaction with bone, while they also provide a higher risk of stress concentration. The examination of pitch and depth may enhance implant design by facilitating the optimization of form and structural characteristics to accommodate unique patient requirements [15-18].

The efficacy of FEA is considerable; yet, its applicability in actual clinical circumstances is limited by model simplifications. This study's models presume that bone and implant materials are homogenous, isotropic, and linearly elastic [16, 19]. The research used a static 200 N axial load to imitate the force applied to the implant, offering insights into stress distribution; however, it does not entirely mimic clinical situations, where dynamic stresses are generated in various directions owing to mastication and phonation. These forces include axial, lateral, and rotational components, potentially affecting stress distribution. Integrating dynamic loading circumstances and fatigue analysis may provide a more precise comprehension of implant performance in authentic oral environments [16, 20]. This research emphasizes CFR-PEEK above titanium, indicating a transition towards materials exhibiting mechanical capabilities more akin to those of bone. Titanium, while robust and biocompatible, has much more stiffness than bone, resulting in stress shielding, whereby the implant bears a larger portion of the load than the adjacent bone, possibly resulting in bone resorption. In contrast, CFR-PEEK has an elastic modulus that approximates that of bone, facilitating natural load distribution and mitigating bone loss resulting from under-stimulation. Nonetheless, CFR-PEEK has some restrictions [14, 16, 19, 20]. Despite providing superior flexibility, it is a relatively novel material in dental implantology, with few long-term performance studies available. Subsequent investigations may explore the longevity and osseointegration of CFR-PEEK implants, especially in circumstances that more accurately reflect actual oral environments.

This study may influence implant design and material choice in clinical dentistry. Research indicates that thread form may have a lesser impact on stress distribution compared to material selection and thread diameters. Although CFR-PEEK has potential, more clinical investigations are required to substantiate its efficacy relative to titanium in practical applications. Subsequent study may build upon these results by investigating dynamic loads, fatigue, sophisticated material modelling, and individualized patient analysis. Dynamic pressures would replicate the real conditions encountered by implants in the oral cavity, providing more relevant data for clinical application, while fatigue research would provide insights into long-term durability. Custom implants designed to match the specific anatomical features of individual patients may improve implant success rates, as may in vivo studies evaluating thread designs and materials. This study elucidates the impact of implant design on stress distribution, emphasizing the advantages of CFR-PEEK and the possible benefits of thread shapes that generate compressive stress. These results highlight the need for further assessment of implant design elements and stress the significance of continuous research to link computational outcomes to clinical applications.

5. CONCLUSION

Using the FEM approach, this study examined the effects of thread designs on the stress distribution of dental implants placed in jaw bones. Overall, the simulation findings are consistent with the key deductions that follow:

(1) V thread and square-threaded implants have approximately similar von Mises stress distributions at the bone- implant interface.

(2) Different implant thread types result in varying levels of stress intensity on the bone structure.

(3) The cortical bone and bone structure next to the first thread experience higher levels of von Mises stresses compared to the spongy bone.

(4) Implant materials also directly influence the stress distribution in the supporting bone structure.

(5) Based on the information provided above, it can be determined that selecting threads based on certain situations might provide more favourable results.

(6) This investigation provides a clearer knowledge of how different prosthetic thread designs affect the overall distribution of stress. In this work, implant geometries, material selection and loading scenarios taken into account. It was discovered that there was a strong correlation between most stress parameters and the earlier research. Certain observations were novel and will aid in the proper choice of implant thread design.

REFERENCES

[1] Li, R., Wu, Z., Chen, S., Li, X., Wan, Q., Xie, G., Pei, X. (2023). Biomechanical behavior analysis of four types of short implants with different placement depths using the finite element method. The Journal of Prosthetic Dentistry, 129(3): 447.e1-447.e10. https://doi.org/10.1016/j.prosdent.2023.01.005

[2] Ueda, N., Takayama, Y., Yokoyama, A. (2017). Minimization of dental implant diameter and length according to bone quality determined by finite element analysis and optimized calculation. Journal of Prosthodontic Research, 61(3): 324-332. https://doi.org/10.1016/J.JPOR.2016.12.004

[3] Narendrakumar, U., Mathew, A.T., Iyer, N., Rahman, F., Manjubala, I. (2018). A 3D finite element analysis of dental implants with varying thread angles. Materials Today: Proceedings, 5(5): 11900-11905. https://doi.org/10.1016/J.MATPR.2018.02.163

[4] Leblebicioğlu Kurtuluş, I., Kilic, K., Bal, B., Kilavuz, A. (2022). Finite element analysis of the stress distribution associated with different implant designs for different bone densities. Journal of Prosthodontics, 31(7): 614-622. https://doi.org/10.1111/jopr.13539

[5] Pirmoradian, M., Naeeni, H.A., Firouzbakht, M., Toghraie, D., Darabi, R. (2020). Finite element analysis and experimental evaluation on stress distribution and sensitivity of dental implants to assess optimum length and thread pitch. Computer Methods and Programs in Biomedicine, 187: 105258. https://doi.org/10.1016/j.cmpb.2019.105258

[6] Bordin, D., Bergamo, E.T., Bonfante, E.A., Fardin, V.P., Coelho, P.G. (2018). Influence of platform diameter in the reliability and failure mode of extra-short dental implants. Journal of the Mechanical Behavior of Biomedical Materials, 77: 470-474. https://doi.org/10.1016/J.JMBBM.2017.09.020

[7] Marcián, P., Wolff, J., Horáčková, L., Kaiser, J., Zikmund, T., Borák, L. (2018). Micro finite element analysis of dental implants under different loading conditions. Computers in Biology and Medicine, 96: 157-165. https://doi.org/10.1016/J.COMPBIOMED.2018.03.012

[8] Ouldyerou, A., Merdji, A., Aminallah, L., Roy, S., Mehboob, H., Özcan, M. (2022). Biomechanical performance of Ti-PEEK dental implants in bone: An in-silico analysis. Journal of the Mechanical Behavior of Biomedical Materials, 134: 105422. https://doi.org/10.1016/J.JMBBM.2022.105422

[9] Blanch-Martínez, N., Arias-Herrera, S., Martínez-González, A. (2021). Behavior of polyether-ether-ketone (PEEK) in prostheses on dental implants. A review. Journal of Clinical and Experimental Dentistry, 13(5): e520-e526. https://doi.org/10.4317%2Fjced.58102

[10] Lee, H., Jo, M., Noh, G. (2021). Biomechanical effects of dental implant diameter, connection type, and bone density on microgap formation and fatigue failure: A finite element analysis. Computer Methods and Programs in Biomedicine, 200: 105863. https://doi.org/10.1016/j.cmpb.2020.105863

[11] Zaparolli, D., Peixoto, R.F., Pupim, D., Macedo, A.P., Toniollo, M.B., de Mattos, M.D.G.C. (2017). Photoelastic analysis of mandibular full-arch implant-supported fixed dentures made with different bar materials and manufacturing techniques. Materials Science and Engineering: C, 81: 144-147. https://doi.org/10.1016/j.msec.2017.07.052

[12] Aguir, H., Mabrouk, Y., Chamekh, R., Saadellaoui, I. (2023). Influence of implants number on the biomechanical behavior of implant-supported complete prosthesis: A 3D finite element analysis. Heliyon, 9(7): e17956. https://doi.org/10.1016/j.heliyon.2023.e17956

[13] Tonin, B.S., Fu, J., He, Y., Ye, N., Chew, H.P., Fok, A. (2023). The effect of abutment material stiffness on the mechanical behavior of dental implant assemblies: A 3D finite element study. Journal of the Mechanical Behavior of Biomedical Materials, 142: 105847. https://doi.org/10.1016/J.JMBBM.2023.105847

[14] Liao, X., Cao, R., Zhong, J., Chen, C., Pan, S. (2024). Influence of implant distribution on the biomechanical behaviors of mandibular implant-retained overdentures:

A three-dimensional finite element analysis. BMC Oral Health, 24(1): 405. https://doi.org/10.1186/s12903-024-04146-4
[15] Shash, M., Nazha, H., Abbas, W. (2019). Influence of different abutment designs on the biomechanical behavior of one-Piece zirconia dental implants and their surrounding bone: A 3D-FEA. IRBM, 40(6): 313-319. https://doi.org/10.1016/J.IRBM.2019.07.001
[16] Thomková, B., Marcián, P., Borák, L., Joukal, M., Wolff, J. (2024). Biomechanical performance of dental implants inserted in different mandible locations and at different angles: A finite element study. The Journal of Prosthetic Dentistry, 131(1): 128.e1-128.e10. https://doi.org/10.1016/J.PROSDENT.2023.10.012
[17] Yang, B., Landa, A.I., Heuberger, P., Ploeg, H.L. (2024). Effects of dental implant diameter and tapered body design on stress distribution in rigid polyurethane foam during insertion. Medical Engineering & Physics, 129: 104181. https://doi.org/10.1016/j.medengphy.2024.104181
[18] Qiu, P., Cao, R., Li, Z., Fan, Z. (2024). A comprehensive biomechanical evaluation of length and diameter of dental implants using finite element analyses: A systematic review. Heliyon, 10(5): e26876. https://doi.org/10.1016/j.heliyon.2024.e26876
[19] Rajaeirad, M., Fakharifar, A., Posti, M.H.Z., Khorsandi, M., Watts, D.C., Elraggal, A., Ouldyerou, A., Merdji, A., Roy, S. (2024). Evaluating the effect of functionally graded materials on bone remodeling around dental implants. Dental Materials, 40(5): 858-868. https://doi.org/10.1016/j.dental.2024.04.002
[20] Karash, E.T., Slewa, M.Y., AL-Maula, B.H. (2023). State stress analysis of dental restoration materials using the ANSYS program. Revue des Composites et des Matériaux Avancés-Journal of Composite and Advanced Materials, 33(3): 183-192. https://doi.org/10.18280/rcma.330306