Scientific Research Journal,
Vol 22, special issue (2025) 55-72
https://doi.org/10.24191/srj.v22is.11950

In-Depth Analysis of the Biomechanical Performance of Dental Implants: Square vs. Spiral Thread Designs Under Chewing Conditions

Marsya Athirah Ahmad Kamal[1], Muhammad Faris Abd Manap[1,*], Nur Faiqa Ismail[2], Anas Hakimee Ahmad Ubaidillah[3], Nor Shamimi Shaari[1], Mahfuzah Zainudin[1], Solehuddin Shuib[4], Hazimi Ismail[1]

[1]Mechanical Engineering Studies, College of Engineering, Universiti Teknologi MARA, Pulau Pinang Branch, Malaysia
[2]Faculty of Information Sciences and Engineering, Management and Science University, Shah Alam, Malaysia
[3]Faculty of Dentistry, Universiti Malaya, Kuala Lumpur, Malaysia
[4]School of Mechanical Engineering, College of Engineering, University Teknologi MARA, Shah Alam, Malaysia

[*]Corresponding Author's E-mail: muhammadfaris@uitm.edu.my

Received: 30 August 2024
Accepted: 10 September 2024
Online First: 25 January 2025

ABSTRACT

Dental implants are known as a procedure to replace a missing tooth. The success of dental implants is affected by the amount of force, implant design, and the tension of implant growth surrounding the bone. The purpose of this project is to study the von-Mises stress, total deformation, and contact pressure of biomechanical stress at tooth implants in the lower jaw during chewing simulation by using the finite element method. The models, including the crown, a few screw implants, abutment, and jawbone, were designed in Solidwork with refinement by using Altair Inspire Studio, while the implant simulation has been conducted in ANSYS Workbench. The simulation began by applying three different loadings of 1000N, 1500N and 2000N to the two assembled models of jawbone with different materials of crown and implant. The two constructed models of the jawbone consist of one featuring a square implant and another incorporating a spiral implant. The simulation results indicate that the jawbone model with two different implants experienced deformation and changes in the von-Mises stresses of the implants. It was observed that crowns made of metal or zirconia *experienced the lowest stresses, with a value of 4407.5 MPa on the crown during chewing conditions. The mechanical analysis of a square implant under a 1000N load showed a reduction of 896.66 MPa, 1230.1 MPa, and 6.992 mm in von-Mises stress, contact pressure, and total deformation, respectively. The study concludes that square implants under 1000N load demonstrate significantly lower von-Mises stress, contact pressure, and total deformation compared to other implant designs, highlighting their potential effectiveness in reducing biomechanical stress during chewing.*

Keywords: Dental; Implant; Abutment; Finite Element Analysis; Mechanical Properties

INTRODUCTION

The demand for dental implants has risen over the past three decades. Dental implants are surgical fixtures that replace missing tooth roots by integrating with the jawbone over several months. Often considered the most effective treatment for tooth loss due to injury, trauma, or decay, some research suggests other treatments may better retain natural teeth [1,2].

There are two categories of dental implants: subperiosteal implants and endosteal implants. Endosteal implants are surgically inserted, wherever tooth origins are situated within the mandible, On the contrary, subperiosteal implants do not require drilling into the mandible; rather, they are submerged underneath the molars. Research suggests that subperiosteal implants offer advantages with less time consuming procedures [3]. This would suggest their positioning on or above the bone. The implant, crown, and abutment were subsequently used during the procedure.

Factors of Successful and Failure of Dental Implant

In recent years, dental implants have become increasingly popular in oral rehabilitation and orthopedics as replacements for lost or damaged teeth, helping to restore chewing functions. This growing use shows that dental implants are reliable [4]. With proper care, dental implants can last a lifetime and improve the quality of life for many people [5]. Studies have shown that dental implants can have a high success rate, with a retention rate of over 95% if they are well-designed, correctly manufactured, and properly placed [6–9]. For long-term success, it is crucial that the implant can handle biting forces and transfer them safely to the surrounding tissues.

However, dental implant failures can occur due to infection, heavy loading, and poor primary stability [10]. Poor dental hygiene can lead to peri-implantitis, an inflammatory condition that damages bone and tissue. Overloading happens when the forces on the implant exceed its capacity. Improper positioning or excessive biting pressure can hinder osseointegration, the process by which the implant bonds with the bone [11]. Implants can also fail due to poor bone quality or quantity, bad surgical techniques, or inadequate implant design. These factors highlight the importance of careful surgical planning, proper patient selection, and good post-operative care for the success of dental implants.

Despite these advancements and understanding, there remains a specific gap in the comprehensive evaluation of mechanical properties, particularly in the context of thread designs and their impact on stress distribution and deformation under chewing conditions. Current studies have not fully addressed how different thread geometries influence the biomechanical performance of dental implants. This gap is critical because understanding the interaction between thread design and mechanical stress can lead to improved implant designs that minimize failure rates and enhance patient outcomes.

Types of Screw Thread and Component of Tooth Implant

The thread's geometrical features affect the transmission of stresses from the implant to the bone [12]. Lowering the initial contact with the surrounding bone is crucial in improving the primary stability of the implant and optimising stress distribution [13,14]. The screw implant can have a lot of type of design whether it is different on it its thread, it shapes which tapper or not or its design itself. There are many types of screw thread implant which standard V-thread, square thread, buttress thread, spiral thread, and many others [11]. The among top usage of implant thread are shown in Figure 1. The thread lead and pitch are important thread characteristics that affect bone-implant contact, stress distribution and primary stability.

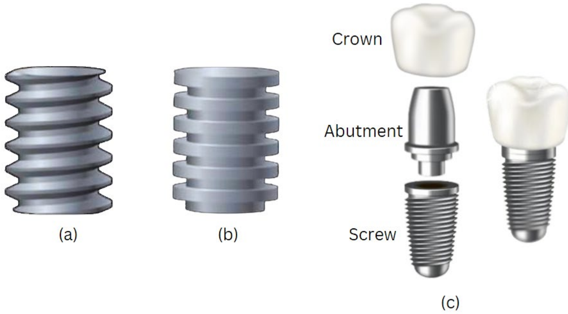

Figure 1: View of dental implant. a) spiral thread, b) square thread, and c) component of dental implant

The main parts of a dental implant are the crown, abutment, and screw. The crown, which mimics the look and function of a natural tooth, can be screwed into the implant or cemented onto the abutment [15]. Crowns come in various shapes, sizes, and strengths based on the material used. The abutment, a small connector that extends through the gum line, attaches the crown to the implant and can be made from metal or tooth-colored materials like titanium or zirconia [16]. The screw, or root of the implant, is drilled into the jawbone and integrates with the bone tissue, typically made from different grades of titanium [17]. Thus, the main objective of this research is to analyze the biomechanical stress, deformation, and contact pressure on dental implants (as whole) during chewing simulation, considering different design, and materials using finite element analysis (FEA).

METHODOLOGY

Modelling Design

The jawbone and implant geometry were designed in Solidworks, with the crown refined using Altair Software. Solidworks was also used to assemble the implant parts into the jawbone model. The assembled models were then imported into Ansys Workbench for simulation. There are two distinct types of screw implants: the spiral thread implant and the square thread implant. Initially, the model acquired the parameter of two distinct types of screw implant abutment and jawbone, which was based on the prior research [18–20].

Both types of implants in this experiment have identical dimensions, with a diameter of 6.2 mm and a length of 10 mm. The spiral thread has a length of 0.45 mm, a depth of 0.30 mm, and a width of 0.10 mm, while the square thread has a length of 0.20 mm and a depth of 0.30 mm. The abutment used has a top diameter of 4.00 mm, a bottom diameter of 6.00 mm, a screw length of 3.00 mm with an M12×0.25 thread, and a total length of 9.20 mm. The assembly process began with the crown, followed by the abutment, and then the screw implant, integrated into two rectangular jawbone sections to create a 3D model for finite element simulation. The jawbone model included inner cortical bone and outer cancellous bone. In the contact modeling, the crown is bonded to the abutment, the abutment connects to the implant through frictional contact, and the implant interacts with both cortical and cancellous bones through friction. The cortical bone is bonded to the cancellous bone. Figure 2 shows the full 3D model and sectional views for both square and spiral implants.

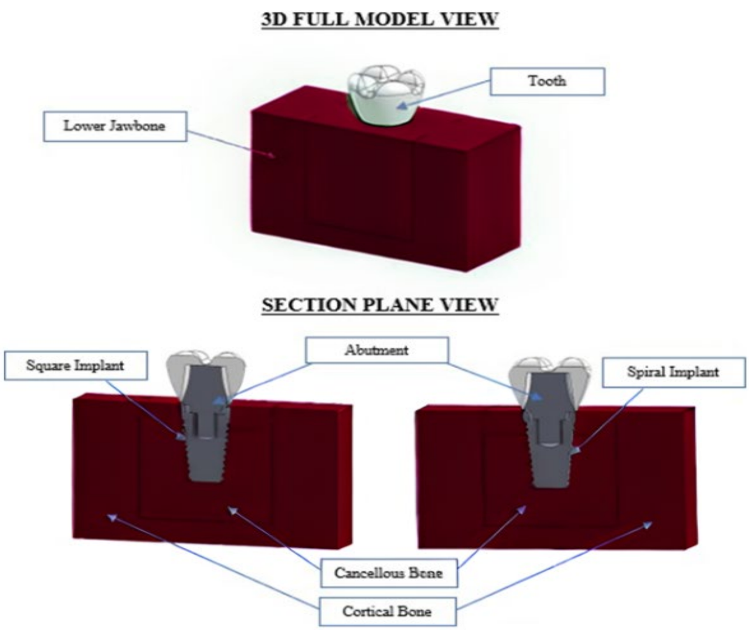

Figure 2: Full view and section plane view of the dental implant in jaw area

For square implants, titanium is used for the implant material, while ceramic, metal, and zirconia are options for the crown material under forces of 1000, 1500, and 2000 N. Similarly, for spiral implants, titanium is used for the implant material, with ceramic, metal, and zirconia as choices for the crown material under the same force conditions. The material properties used in this analysis simulation which the Young's Modulus, Poisson's ratio and density are obtained from previous research studies shown in Table 1 and boundary condition shown in Figure 3.

Table 1: Loading condition in FE analysis

Young's Modulus, MPa		Poison ratio	Density, g/cm^3	Reference
Ceramic	68 900	0.28	-	Zhang et al. [9]
Zirconia	200 000	0.30	-	Cheng et al. [21]
Metal (common)	208 000	0.31	-	Cantó-navés et al. [22]
Titanium	96 000	0.30	4.50	Cheng et al. [21]
Cortical bone	147000	0.30	1.85	Paracchini et al. [23]
Cancellous bone	1470	0.30	0.90	Paracchini et al. [23]

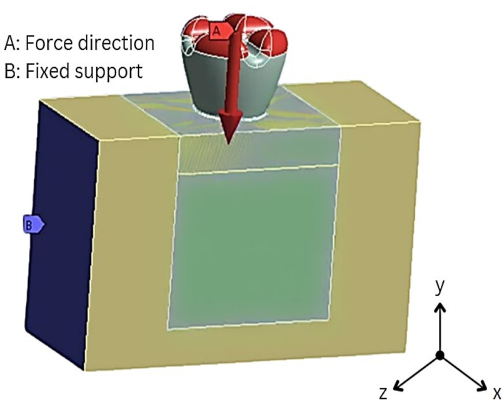

Figure 3: Loading condition and boundary conditions at jawbone. (A) for fixed support and (B) loading condition

After completing the design of the model and the observation of simulation in Ansys Workbench, the comparison material crown and screw implant were continued to validate the mechanical properties which is design had lower stress and deformation. Comparative research was conducted to determine the optimal material for the components of the implant.

RESULTS AND DISCUSSION

FEA of Tooth Implant Component

As accordance the methodology outlined in a previous research paper, simulation experiments were conducted using ANSYS Workbench. This analysis provides a simulation of the maximum von-Mises stress and total deformation for certain components of the models which vital in dental implant system [24]. The first simulation was the abutment part, in which the 1000N, 1500N, 2000N loads are applied to the crown's apex while three fixed support faces were applied to the cortical bone's based on Figure 4. The outcomes of each applied load for which the maximum von-Mises and total deformation were determined by analysis. The maximum von-Mises stress gained for the load 2000N was greater than load 1000N and 1500N. The changes load from 1500N to 1000N will give 40% reduction and from the 2000N to 1500N will give 28.57% of reduction. This demonstrates that during the chewing condition, the initial load supplied to the abutment will have a greater effect than the subsequent load. The total deformation was determined for loads of 1000N is 8.7958×10^{-3}mm, 1500N is 1.3194×10^{-2} mm and 2000N is 1.7592×10^{-2} mm. The overall deformation of the 2000N load is greater compared to other loads. This demonstrated maximum deformation value increases as the biting force increases [21].

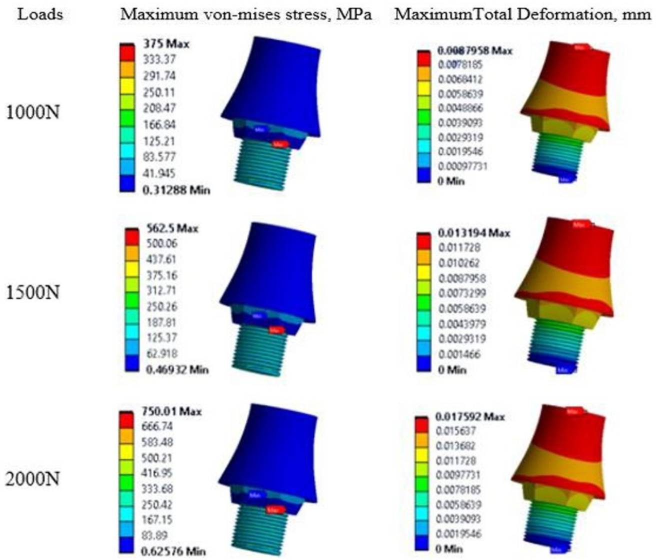

Figure 4: Abutment mechanical properties and maximum von-Mises stress after loading conditions applied

According to Figure 5, the square design had lowered von-Mises stress than the spiral design. The values maximum von-Mises stress of square thread for 1000N is 282.74 MPa, 1500N is 424.12 MPa and 2000N is 565.49 MPa. While spiral thread values for 1000N is 410.39 MPa, 1500N is 615.59 MPa and 2000N is 820.79 MPa. This suggests that the square design may be more effective at distributing stress and preventing deformation, compared to the spiral design. This is consistent with the findings of Oliveira *et al.* [10], who found that different implant designs can have a significant impact on the stresses and deformations experienced by the implant and surrounding bone. In this analysis, the results were compared with those from the study by Alemayehu *et al.* [20]. The maximum von-Mises stress values computed in this study showed some differences from those in the referenced paper. Specifically, the spiral thread exhibited a 14.22% error, while the square thread showed a 23.62% error compared to the reference values. These differences are due to the different loading conditions used to replicate chewing circumstances in this study. Additionally, slight variations in the implant design dimensions between this study and the reference paper also influenced the results. Despite these differences, the findings are reasonable and useful for understanding thread patterns' biomechanical performance.

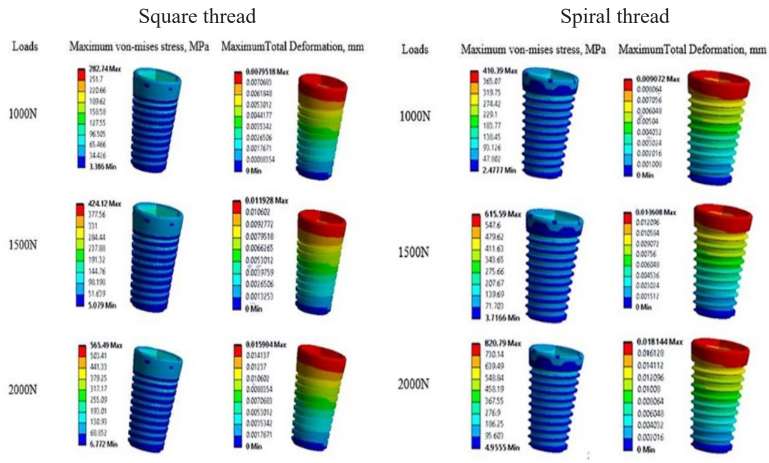

Figure 5: Square thread vs spiral thread mechanical properties

Under different loading conditions, the spiral thread design had a total deformation of 9.072×10^{-3} mm for 1000N, 1.3608×10^{-2} mm for 1500N, and 1.8144×10^{-2} mm for 2000N. In contrast, the square thread design had a lower total deformation of 7.9518×10^{-3} mm, 1.1928×10^{-2} mm, and 2000N is 1.5904×10^{-2} mm. This suggests that the square thread design may be more effective at resisting deformation and maintaining its shape, compared to the spiral thread design. The square thread design also appears to have a greater influence on the magnitude and distribution of total deformation in the cortical bone, which may make it more suitable for dental implant applications [20]. Overall, it seems that the square thread design performed better than the spiral thread design in terms of both von-Mises stress and total deformation.

FEA Mechanical Properties for Assemble Models of Jawbone

This study analyzed jawbone models with square and spiral implants under a 1000N load using a ceramic crown, as illustrated in Figure 6. The primary objective was to evaluate and compare the mechanical properties, including von Mises stress, total deformation, and bone contact pressure, for each implant design. These parameters are crucial for assessing the performance and potential clinical outcomes of dental implants.

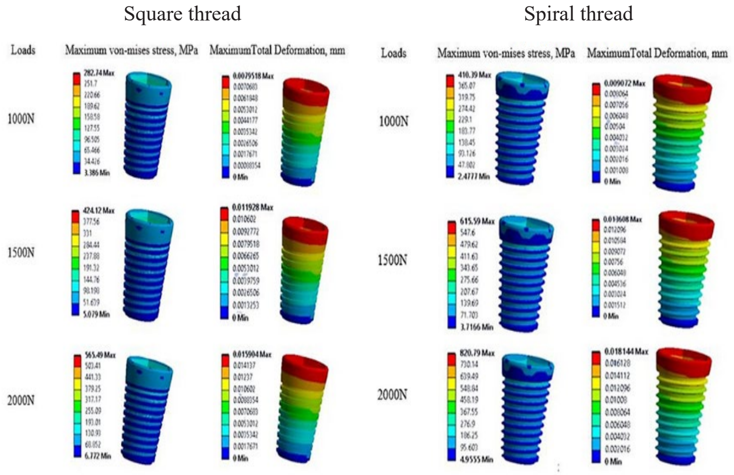

Figure 6: Total deformation at three different loading conditions at assembled models

The finite element analysis (FEA) results revealed distinct differences between the two implant designs. The square implant model exhibited a lower von Mises stress of 896.66 MPa. This indicates that the square implant design is more effective in distributing the applied load across the implant and the surrounding bone, thereby reducing the peak stress concentration. Lower von Mises stress is desirable as it suggests a reduced likelihood of implant material fatigue and bone resorption, which are critical factors for the long-term success of dental implants.

In terms of total deformation, the square implant showed a higher deformation of 6.992 mm compared to the spiral implant model, which had a deformation of 6.9542 mm. Although the difference in deformation is relatively small, it suggests that the square implant, despite experiencing lower stress, undergoes slightly more displacement under load. This could be attributed to the design geometry and the way the load is transferred through the implant to the bone. Higher deformation in the square implant may imply a greater flexibility, which could be beneficial for accommodating masticatory forces and reducing stress shielding effects.

Conversely, the spiral implant model exhibited a significantly higher von Mises stress of 1518 MPa. This higher stress concentration indicates that the spiral design may be less effective in evenly distributing the load, potentially leading to increased risk of implant fatigue and bone damage over time. The design characteristics of the spiral threads, which might concentrate stress at specific points, could explain this higher stress value.

However, the spiral implant demonstrated a remarkable 45.02% increase in bone contact pressure compared to the square implant. Bone contact pressure is a critical factor as it influences the primary stability of the implant and the subsequent osseointegration process. Higher bone contact pressure in the spiral implant suggests a more intimate and forceful contact with the bone, which could enhance the initial stability and promote faster and more robust integration with the surrounding bone tissue.

These findings highlight the complex interplay between stress distribution, deformation, and bone contact pressure when evaluating and comparing different implant designs. The lower stress and higher deformation observed in the square implant indicate a design that may reduce the risk of material fatigue and bone resorption, potentially leading to a more durable and longer-lasting implant. On the other hand, the higher bone contact pressure in the spiral implant suggests superior primary stability, which is crucial during the early stages of implantation and healing.

The graph in Figure 7 shows comparison between the jawbone model spiral implant with jawbone model square implant. The result of this histogram indicated that model jawbone with square implant gave the least maximum von-Mises stress after applied three different loading which 1000N is 896.66 MPa, 1500N is 1345 MPa and 2000N is 1793.3 MPa to simulate chewing conditions. Compared with the model jawbone with spiral implant which affect the most for maximum von-Mises stress where for 1000N, 1500N and 2000N recorded 1518.8 MPa, 2278.2 MPa and 3037.6 MPa. This analysis shows the square implant has thread design where provide less stresses and torque occurred in surrounding bone.

Three loading conditions were given to spiral and square implant model jawbones to imitate chewing. The square implants distort jawbone more than spiral implants. Maximum total deformation of 1000N, 1500N, and 2000N loadings was 6.9542 mm, 10.431 mm, and 13.908 mm for model jawbone with spiral implant and 6.992 mm, 10.488 mm, 13.984 mm for model jawbone with square implant. Square implants cortical bone deformation suggests they can tolerate chewing pressures and strains. The square implant model deforms more than spiral, providing bone stability and improving bone remodelling.

.

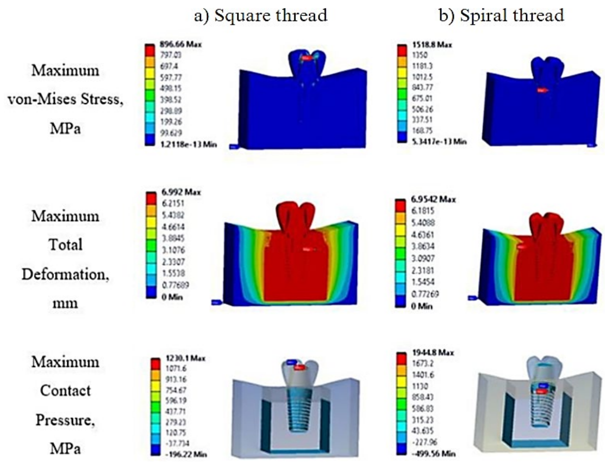

Figure 7: Comparison maximum von-Mises stress, total deformation for square and spiral thread implant

FEA Crown Materials

In this study the performance of different crown materials (metal, zirconia, and ceramic) was examined under a load of 1000N. Metal and zirconia crowns exhibited the lowest stress levels at 4407.5 MPa, while ceramic crowns experienced slightly higher stress at 4468.2 MPa. The applied force during chewing influenced the biomechanical stresses and jawbone deformation. Ceramic crowns deformed the most (0.0719 mm), compared to metal (0.0239 mm) and zirconia (0.0248 mm). Zirconia deforms more than metal. Wazeh *et al.* [25] noted that switching from zirconia to ceramic increases crown deformation, and El-Anwar *et al.* [26] observed that crown material affects jawbone stress. This indicates that the material of the crown impacts both jawbone stress and deformation.

Figure 8 compares the maximum von-Mises stress on different crown materials under various loads. Ceramic crowns showed the highest stress levels, with values of 4468.2 MPa at 1000N, 6702.3 MPa at 1500N, and 8936.3 MPa at 2000N. Metal and zirconia crowns had the same maximum von-Mises stress values of 4407.5 MPa at 1000N, 6611.2 MPa at 1500N, and 8815 MPa at 2000N. These crowns are made from strong and durable materials, making them resistant to fractures. During chewing, the crown material plays a crucial role as the first point of contact for applied force. The graph shows that metal crowns have the lowest total deformation under different loads, deforming 0.0239 mm at 1000N, 0.0358 mm at 1500N, and 0.0478 mm at 2000N, compared to ceramic and zirconia crowns. The highest deformation values for ceramic and zirconia crowns are 0.1438 mm and 0.0497 mm, respectively. This proves that metal crowns are flexible and can withstand chewing forces, protecting teeth from further damage due to their strength and resistance to chipping or cracking.

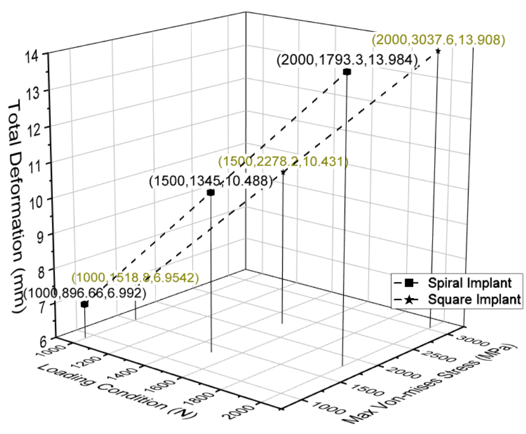

Figure 8: Comparison using various crown materials

CONCLUSION

In contrast to existing studies, this research leverages advanced simulation techniques to provide clear insight into the mechanical properties of dental implants under varying chewing conditions with a refined crown's design using Altair Software. This study examined how different thread designs in dental implants perform under simulated chewing. Fully whole component including simplified jawbone model and crown added more precise data comparison. The results showed that square thread designs have lower stress and deformation compared to spiral threads. This will ensure making them better for handling chewing forces. Metal and zirconia crowns also performed better than ceramic crowns, reducing stress on the implants. Choosing the right thread design and crown material is crucial for the success and durability of dental implants. Future research should continue to optimize these factors with a fully modelled jawbone to improve understanding of dental implants.

ACKNOWLEDGEMENT

The authors extend deepest gratitude to the final year project students for their dedication and invaluable contributions to this study. Appreciation is also extended to Universiti Teknologi MARA Cawangan Pulau Pinang for providing the necessary equipment, and to the advisors, colleagues, and friends for their unwavering guidance and support.

REFERENCES

[1] K. Xuan, B. Li, H. Guo, W. Sun, X. Kou, X. He, Y. Zhang, J. Sun, A. Liu, L. Liao, S. Liu, W. Liu, C. Hu, S. Shi and Y. Jin, 2018. Deciduous autologous tooth stem cells regenerate dental pulp after implantation into injured teeth, *Science Translational Medicine*, 10(455), 1–13.

[2] D. Clark and L. Levin, 2019. In the Dental Implant Era - Why We Still Bother Saving Teeth?, *Dental Traumatology : Official Publication of International Association for Dental Traumatology,* 35(6), 368–375.

[3] D. Ângelo and J.V. Ferreira, 2020. The Role of Custom-made Subperiosteal Implants for Rehabilitation of Atrophic Jaws - *A Case Report, Annals of Maxillofacial Surgery*, 10, 507–511.

[4] S. Ivanov, A.A. Muraev, M. Fomin and A. Dymnikov, 2013. Dental Implants in Complex Oral Rehabilitation in Patients with Class III Occlusion and Partial Teeth Loss, *Stomatologiia*, 92(3), 38–42.

[5] M. V. Peshkov, L. Tishkina and I. Uspenskaya, 2023. Orthopedic Rehabilitation by Dental Implantation in Public Healthcare System, *Nauka Molodykh (Eruditio Juvenium),* 11(2), 281–288.

[6] J. Griggs, 2017. Dental Implants, *Dental Clinics of North America,* 61(4), 857–871.

[7] J. Steigenga, K.F. Al-Shammari, F. Nociti, C. Misch and H. Wang, 2003. Dental Implant Design and Its Relationship to Long-Term Implant Success, *Implant Dentistry,* 12, 306–317.

[8] M.K. Noorani and S. Ekram, 2023. Dental Implant Treatment Planning on "Implant Design" – Review of Literature, *Dental Journal of Indira Gandhi Institute of Medical Sciences,* 2(2), 116–122.

[9] G. Zhang, H. Yuan, X. Chen, W. Wang, J. Chen, J. Liang and P. Zhang, 2016. A Three-Dimensional Finite Element Study on the Biomechanical Simulation of Various Structured Dental Implants and Their Surrounding Bone Tissues, *International Journal of Dentistry,* 20(1), 486.

[10] H. Oliveira, A. Brizuela Velasco, J.-V. Ríos-Santos, F. Sánchez Lasheras, B.F. Lemos, F.J. Gil, A. Carvalho and M. Herrero-Climent, 2020. Effect of Different Implant Designs on Strain and Stress Distribution under Non-Axial Loading: A Three-Dimensional Finite Element Analysis, *International Journal of Environmental Research and Public Health*, 17(13), 1–17.

[11] J. Li, J.A. Jansen, X.F. Walboomers and J.J.J.P. van den Beucken, 2020. Mechanical aspects of dental implants and osseointegration: A narrative review, *Journal of the Mechanical Behavior of Biomedical Materials,* 103(1), 103574.

[12] H. Abuhussein, G. Pagni, A. Rebaudi and H. Wang, 2010. The effect of thread pattern upon implant osseointegration., *Clinical Oral Implants Research,* 21(2), 129–136.

[13] A. Nandi, N. Bulbule, S.S. Mondal, A. Bhandari, P.N. Kulkarni and A.K. Jagtap, 2021. The Effect of Different Dental Implant Thread Designs on Stress Distribution on Bone – *A Systematic Review, International Journal of Current Research and Review*, 13(1), 47–51.

[14] P. Grishin, R. Saleev, S. Ksembaev, O. Torgashova and E. Kalinnikova, 2022. Finite Element Analysis of The Humana Dental Implants With An Innovative Surface And Thread Design To Reveal The Stress Distribution In The Implant, Bone Tissue And At The Abutment-Implant-Bone Interface, *Actual Problems in Dentistry,* 18(1), 99–107.

[15] A. Warreth, N. Ibieyou, R.B. O'Leary, M. Cremonese and M. Abdulrahim, 2017. Dental implants: an overview, *Dental Update,* 44(7), 596–620.

[16] M.E. Ou, Y. Ding, W.F. Tang and Y.S. Zhou, 2023. Three-dimensional finite element analysis of cement flow in abutment margin-crown platform switching, *Journal of Peking University. Health sciences,* 55(3), 548–552.

[17] M.M. McCracken, 1999. Dental implant materials: commercially pure titanium and titanium alloys., Journal of Prosthodontics : Official *Journal of the American College of Prosthodontists, 8(1),* 40–43.

[18] H. Topkaya and M.O. Kaman, 2018. Effect of Dental Implant Dimensions on Fatigue Behaviour: A Numerical Approach, *Uludağ Üniversitesi Mühendislik Fakültesi Dergisi*, 23(3), 249–260.

[19] D. Darvish, S. Khorramymehr and M. Nikkhoo, 2021. Finite Element Analysis of the Effect of Dental Implants on Jaw Bone under Mechanical and Thermal Loading Conditions, *Mathematical Problems in Engineering,* 1(1), 1–17.

[20] D.B. Alemayehu and Y.R. Jeng, 2021. Three-Dimensional Finite Element Investigation into Effects of Implant Thread Design and Loading Rate on Stress Distribution in Dental Implants and Anisotropic Bone, *Materials*, 14(22), 6974.

[21] H.C. Cheng, B.Y. Peng, M.S. Chen, C.F. Huang, Y. Lin and Y.K. Shen, 2017. Influence of Deformation and Stress between Bone and Implant from Various Bite Forces by Numerical Simulation Analysis, B*ioMed Research International,* 2017(1), 1–7.

[22] O. Cantó-navés, R. Medina-galvez, X. Marimon, M. Ferrer, Ó. Figueras-álvarez and J. Cabratosa-termes, 2021. A 3D finite element analysis model of single implant-supported prosthesis under dynamic impact loading for evaluation of stress in the crown, abutment and cortical bone using different rehabilitation materials, *Materials,* 14(13), 3519.

[23] L. Paracchini, C. Barbieri, M. Redaelli, D. Di Croce, C. Vincenzi and R. Guarnieri, 2020. Finite Element Analysis of a New Dental Implant Design Optimized for the Desirable Stress Distribution in the Surrounding Bone Region, *Prosthesis*, 2(3), 225–236.

[24] R. Liang, W. Guo, X. Qiao, H. Wen, M. Yu, W. Tang, L. Liu, Y. Wei and W. Tian, 2015. Biomechanical analysis and comparison of 12 dental implant systems using 3D finite element study, *Computer Methods in Biomechanics and Biomedical Engineering,* 18(12), 1340–1348.

[25] A.M. Wazeh, M.I. El-Anwar, R.M.G. Atia, R.M. Mahjari, S.A. Linga, L.M.A. Al-Pakistani and S.A. Yousief, 2018. 3D FEA Study On Implant Threading Role on Selection of Implant and Crown Materials., *Open Access Macedonian Journal of Medical Sciences,* 6(9), 1702–1706.

[26] M.I. El-Anwar, M.S. EL-Mofty, A.H. Awad, S.A. El-Sheikh and M.M. El-Zawahry, 2014. The effect of using different crown and implant materials on bone stress distribution: a finite element study, *Egyptian Journal of Oral & Maxillofacial Surgery*, 5(2), 58–64.