

ORIGINAL ARTICLE

CONTENTS

Development of a methodology and study of the stress-strain state of the biomechanical system "jaw-implant"

Olha S. Musiienko[1], Vitalii V. Parii[2], Oleksii I. Morhun[1] Ihor G. Diadik[2]
[1]NATIONAL TECHNICAL UNIVERSITY OF UKRAINE "IGOR SIKORSKY KYIV POLYTECHNIC INSTITUTE", KYIV, UKRAINE
[2]BOGOMOLETS NATIONAL MEDICAL UNIVERSITY, KYIV, UKRAINE

ABSTRACT
Aim: To investigate the stress-strain state of the "jaw-implant" system under static loading using finite element analysis (FEA) and experimental photoelastic methods, with a focus on different implant placement angles.
Materials and Methods: The study combined numerical and experimental approaches. Samples made of epoxy resin were tested using a polariscope to observe stress distributions, while FEA was performed in SolidWorks 2021 and Ansys 2019 R2. Implant placement angles of 0°, 7°, 15°, and 22° were analyzed under a load of 20 N.
Results: Both experimental and numerical methods revealed stress concentration patterns dependent on implant placement angles. Maximum stress values were observed at larger angles, and the FEA results showed satisfactory agreement with the experimental data.
Conclusions: The developed methodology effectively predicts the stress-strain state of the "jaw-implant" system. The findings demonstrate the importance of optimizing implant placement angles to minimize stress and improve implant longevity. The integration of FEA and photoelastic methods provides a robust tool for enhancing dental implant design and placement strategies.

KEY WORDS: Stress-strain state, jaw-implant system, finite element analysis, photoelastic analysis, biomechanics, implant placement angle

Wiad Lek. 2025;78(9):1758-1764. doi: 10.36740/WLek/212507 DOI

INTRODUCTION
The biomechanical interaction between the jaw and dental implants is a critical area of research in biomedical engineering, particularly given the increasing prevalence of dental implant procedures. Understanding the stress-strain state of the "jaw-implant" system is essential for optimizing implant design and ensuring long-term success [1, 2].

The intricate relationship between biomechanical systems and their structural integrity is pivotal in the field of biomedical engineering, particularly in the design and assessment of dental implants [3].

One of the most critical interfaces in this realm is that of the jaw-implant system, where the mechanical performance and biological compatibility determine the success of implant therapies. Understanding the stress-strain state of this system is essential, as it directly affects the stability of the implant and the surrounding bone. Research has shown that the mechanical interactions between the jawbone and the implant can significantly impact clinical outcomes, making it imperative to investigate these dynamics comprehensively [4, 5].

The implications of this research extend beyond theoretical modeling; they hold significant potential for improving the design of dental implants, optimizing surgical procedures, and ultimately enhancing patient care [6]. By systematically exploring the stress-strain distribution within the jaw-implant system, we aim to identify critical failure mechanisms and suggest design modifications that could lead to more resilient and effective implant solutions. Studies [7] highlight the importance of load distribution in the jaw-implant system, noting that uneven stress can lead to complications such as bone resorption and implant failure. Moreover, the work [8] emphasizes the role of mechanical loading in osseointegration, suggesting that a thorough understanding of stress patterns is vital for improving implant design and surgical techniques.

Furthermore, the work [8] underscores the necessity of integrating biological factors into mechanical assessments. Our methodology not only evaluates mechanical stress but also considers the biological responses that occur at the implant interface, thereby providing a more holistic understanding of the system's performance. Recent advancements in computational modeling, particularly finite element analysis (FEA), offer powerful tools for simulating the mechanical behavior of the jaw-implant interface. As demonstrated by [9], FEA enables researchers to assess how varying loads affect the stress distribution and strain responses within the system, providing insights that are crucial for optimizing implant performance [2, 10].

This article aims to delve into the stress-strain state of the jaw-implant biomechanical system, building on existing literature while presenting new findings from our analyses. By examining the mechanical interactions and their implications for clinical practice, we seek to contribute to the ongoing development of more effective and resilient dental implants, ultimately enhancing patient care in implant dentistry. By systematically exploring the stress-strain distribution within the jaw-implant system, we aim to identify critical failure mechanisms and suggest design modifications that could lead to more resilient and effective implant solutions.

AIM

To investigate the stress-strain state of the "jaw-implant" system under static loading using the finite element analysis and experimental method of photoelastic analysis as a validation.

MATERIALS AND METHODS

Experimental study [11]. Using a metal matrix with dimensions of 90x35x9 mm, a silicone mold was made, which allows for easy removal of samples without damage. The samples were made of a transparent material in which the implant was placed. The jewelry epoxy resin MagicCrystal 3D was used as the specimen material. To determine the mechanical properties of the epoxy resin, a specimen was made in the form of a straight cylinder with a height of 30 mm and a diameter of 20 mm. For this purpose, a plastic matrix was printed on a 3D printer. After that, the matrix was processed to reduce its roughness and a silicone mold was made. Since the properties of epoxy resin vary each time and also change over time, it is necessary to simultaneously produce both specimens for the photoelasticity test and specimens for determining the mechanical properties of the epoxy resin. It is also necessary to test the specimens on the same day for better accuracy of the results.

The compression tests were carried out using an UTM TIRATEST-2151.

The deformation rate was 5 mm/min. The specimen was placed on the test bench and subjected to an axial compressive load. At the same time, the diagram was recorded.

To place the specimen and conduct experiments using the photoelasticity method, a specially designed device, a polariscope, is used. A polariscope consists of a light source, a polarizer, and an analyzer.

The specimen is placed between the polarizer and the analyzer. The analyzer, for its part, transmits only horizontal projections of the light vectors. Thus, the light passing through the analyzer will be horizontally polarized.

As light passes through the sample, its polarization changes due to the distribution of stresses and strains. With a polarizer changes in light polarization can be observed and analyzed, which allows you to determine the stress and strain distribution fields in the specimen.

The load on the implant was transmitted through a steel ball with a diameter of 5 mm using a universal testing machine TIRATEST-2151.

Before starting the tests, a photo of an unloaded specimen was taken to ensure that there were no stresses on the models. After that, the axial load was consistently increased. During the test, extraneous light sources should be turned off in the room.

When certain load values are reached, the specimen is photographed to record the stress distribution in it.

At the end of the experiment, all the photos were analyzed to check the direction and intensity of the stresses. In this case, the higher the order of the stripes and the number of stripes, the greater the stress intensity. In addition, the closer the stripes are to each other, the higher the stress concentration.

Numerical modeling. The SolidWorks 2021 software package was used to build the simulation model. We considered 4 options for implant placement: at angles of 0°, 7°, 15°, and 22° (Fig. 1). The simulation model was made without a thread, the diameter of the model corresponds to the outer diameter of the thread.

Ansys 2019 R2 software package was used to analyze the simulation model. The model is considered in a plane stress-strain state.

Boundary conditions: a force of 20 N was applied vertically (parallel to the side faces of the resin model) to the implant model and lower edge of epoxy resin. The mechanical properties of the materials of the computational model are shown in Table 1.

RESULTS

The results of this study are presented in three successive parts to ensure a comprehensive understanding of the stress–strain state of the jaw–implant system. First, the mechanical properties of the epoxy resin used as

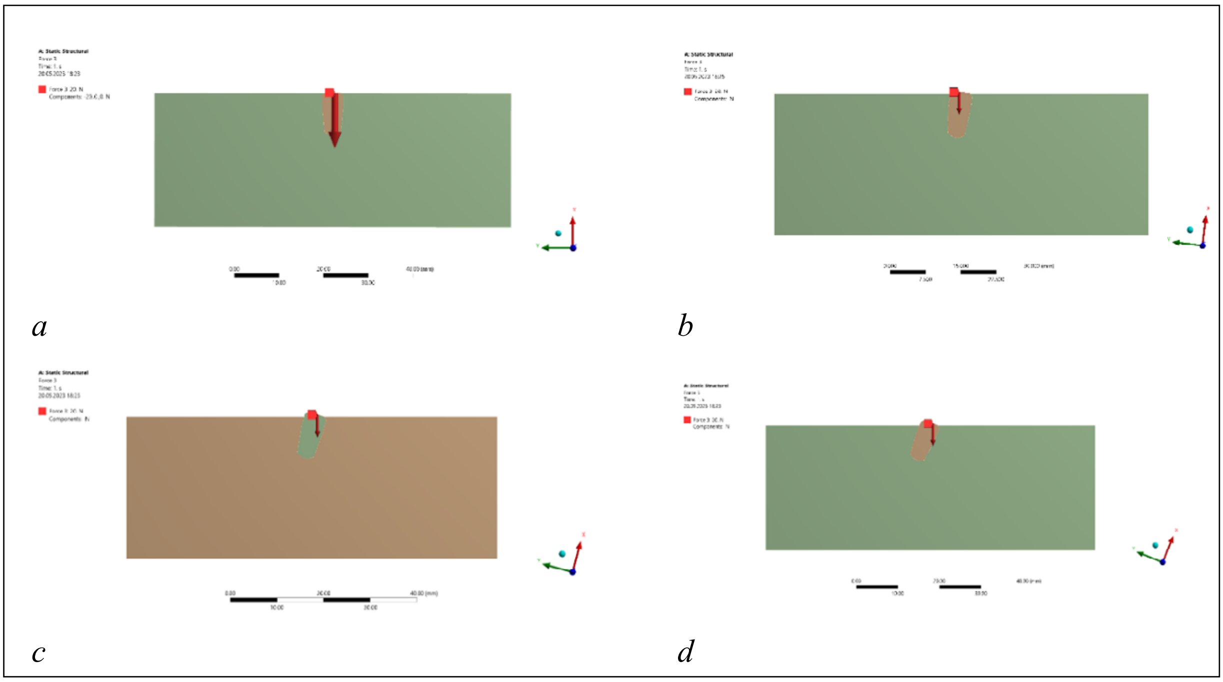

Fig. 1. Installed implant model with inclination angles: (a) 0°, (b) 7°, (c) 15°, (d) 22°
Picture taken by the authors

Table 1. Mechanical properties of materials of the computational model

Object	Material	E, MPa	μ
Implant	Titanium alloy Ti-6Al-4V	$111{,}2 \cdot 10^3$	0,3387
Epoxy resin		7,327	0,3

Source: compiled by the authors of this study

a model material were determined, since these values were required for both experimental interpretation and numerical simulations. Second, photoelastic analysis was employed to visualize stress distribution fields under different loading and inclination conditions. Finally, the findings from finite element modeling were obtained and compared with the experimental observations to validate the numerical approach and quantify the influence of implant inclination on peri-implant stress concentrations.

Compression tests revealed distinct time-dependent variations in the mechanical behavior of the epoxy resin. The stress–strain curves (Fig. 2) obtained from cylindrical specimens demonstrated that samples tested 7 days after curing exhibited significantly higher stiffness compared to those tested 6 months post-curing. The calculated elastic modulus averaged $E = 7.327$ MPa with Poisson's ratio $\nu = 0.30$, values which were subsequently used for finite element simulations.

Photoelastic experiments provided a detailed visualization of stress distribution around the implant under axial loading. In the unloaded state (Fig. 3a), the specimens exhibited no stress-induced birefringence. Under a 20 N vertical load at 0° inclination (Fig. 3b), concentric isochromatic fringes appeared at the implant neck, confirming the presence of localized stress concentrations. Increasing the load to 200 N at 0° (Fig. 3c) led to a marked increase in fringe density, reflecting nonlinear growth of stress intensity.

When the implant was inclined at 15° under 20 N load (Fig. 3d), asymmetric fringe patterns were observed. The isochromes became denser on the side opposite to the direction of inclination, indicating higher tensile stresses and less favorable load distribution. This confirmed that implant tilting introduces significant biomechanical asymmetry in stress transfer.

Numerical modeling was performed for the presented systems and stress distribution fields were obtained (Fig. 4). Finite element analysis was performed for implant inclinations of 0°, 7°, 15°, and 22°. The obtained stress distribution fields (Fig. 4) demonstrated a progressive increase in both the magnitude and localization of stress concentrations as the inclination angle increased.

For the 0° inclination model, the stress distribution exhibited axial symmetry with the highest von Mises stresses localized in the cervical peri-implant region.

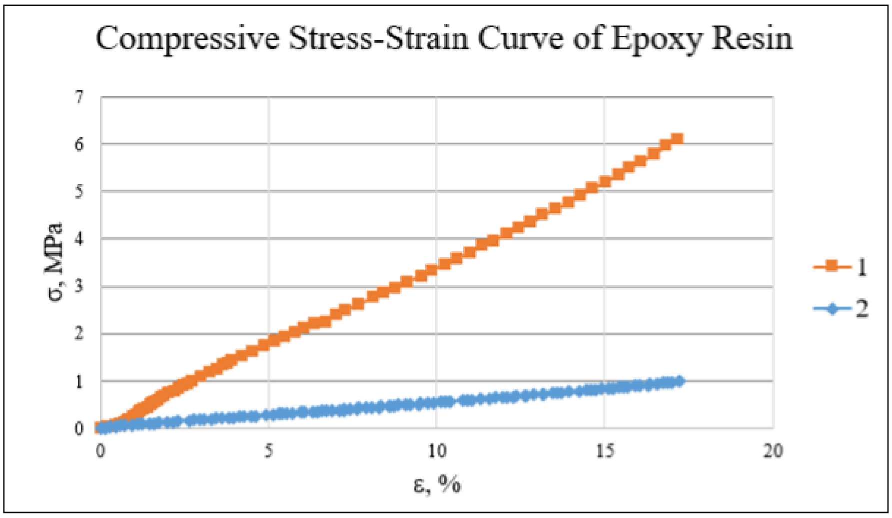

Fig. 2. Mechanical properties of epoxy resin: (1) 6 months after curing; (2) 7 days after curing
Picture taken by the authors

Peak values were concentrated at the implant neck, gradually diminishing along the apical direction. This configuration corresponded to favorable load transfer and minimal shear components.

At an inclination of 7°, the stress field showed distinct asymmetry relative to the implant axis. Maximum stresses shifted toward the cortical region on the side opposite to the loading vector, accompanied by an approximately 12% increase in peak von Mises stress compared to the vertical model. The stress gradients indicated an onset of unfavorable load redistribution.

In the 15° model, stress redistribution became more pronounced. High-intensity zones extended from the cervical margin into the apical third of the implant–resin interface. Peak stresses increased by approximately 25–27% relative to the 0° configuration, and the stress field revealed broader areas of critical concentration, which is biomechanically unfavorable for osseointegration.

The 22° inclination model exhibited the most unfavorable conditions, with stress maxima simultaneously localized at the cervical and apical implant regions. The von Mises stresses at this angle exceeded the threshold values considered biomechanically tolerable for peri-implant bone ($\sigma\_max > 30$ MPa in the simulated resin analogue, extrapolated to cortical bone properties). Such conditions are likely to accelerate peri-implant bone resorption and compromise implant stability.

The simulated stress distribution patterns corresponded well with the photoelastic fringe orders obtained experimentally. In particular, the densification of isochromatic fringes at increasing tilt angles was consistent with the numerical prediction of elevated von Mises stresses. The highest degree of correlation was observed for the 0° and 15° models, confirming the adequacy of the finite element model as a predictive tool for clinical implant biomechanics.

DISCUSSION

Our review of the literature has shown the relevance of studies devoted to the determination of the stress-strain state of jaw-implant systems, which is discussed in [12-14]. In the cited works, the authors use the finite element method as a tool for determining the stress-strain state of the jaw and the implant installed in it. However, the above-mentioned works did not validate the numerical model by means of analytical calculations or experimental studies. We have developed a methodology for validating a numerical model of the jaw-implant system based on the method of photoelasticity at different implant angles.

In contrast to this research, we carried out a study on axial loads applied through a ball that simulated a food lump during the chewing cycle. This scientific work aimed to determine how the distribution of these loads depends on the diameter of the implant [15].

This scientific research studied the distribution of loads in short implants with different types of abutment fixation to the implant. In contrast to this research, we studied an implant that had a connection with an abutment featuring an internal cone of 11 degrees [16].

The use of photoelastic analysis as a methodology for verifying numerical modeling of experimental work provides a powerful impetus for studying the stress-strain states that occur in the jawbone during chewing loads. However, there is a challenge related to the differing densities of the cortical and spongy layers of bone tissue. In numerical modeling, we account for these differences, but in full-scale experiments, this issue remains unresolved because the resin we use is homogeneous. Therefore, the development and search for a filler that can closely mimic the cortical and spongy layers is an urgent task for scientific research.

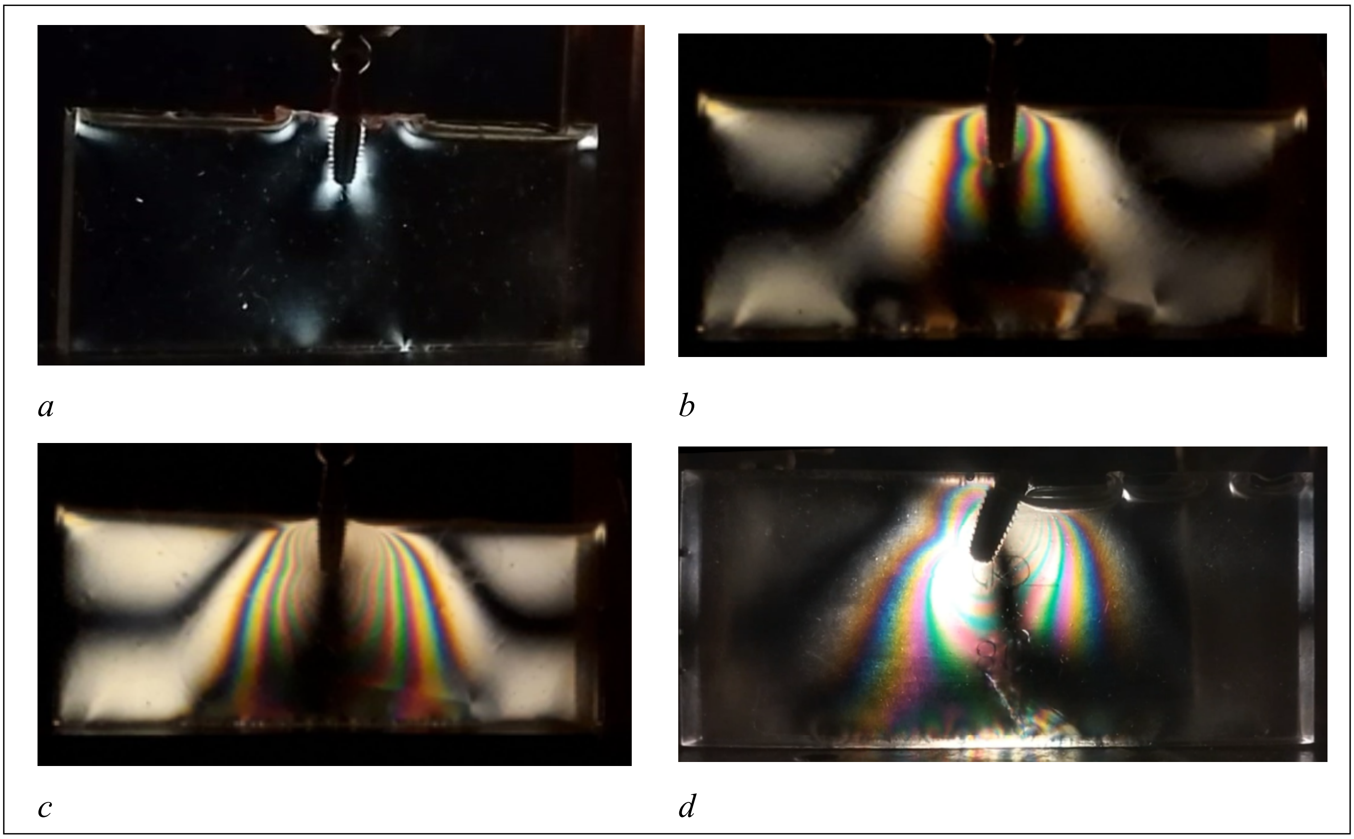

Fig. 3. (a) Unloaded specimen in the polarizer; (b) Stress distribution in a specimen under a force of 20N with an inclination of 0^0; (c) Stress distribution in a specimen under a force of 200N with an inclination of 0^0; (d) Stress distribution in a specimen under a force of 20N with an inclination of 15^0
Picture taken by the authors

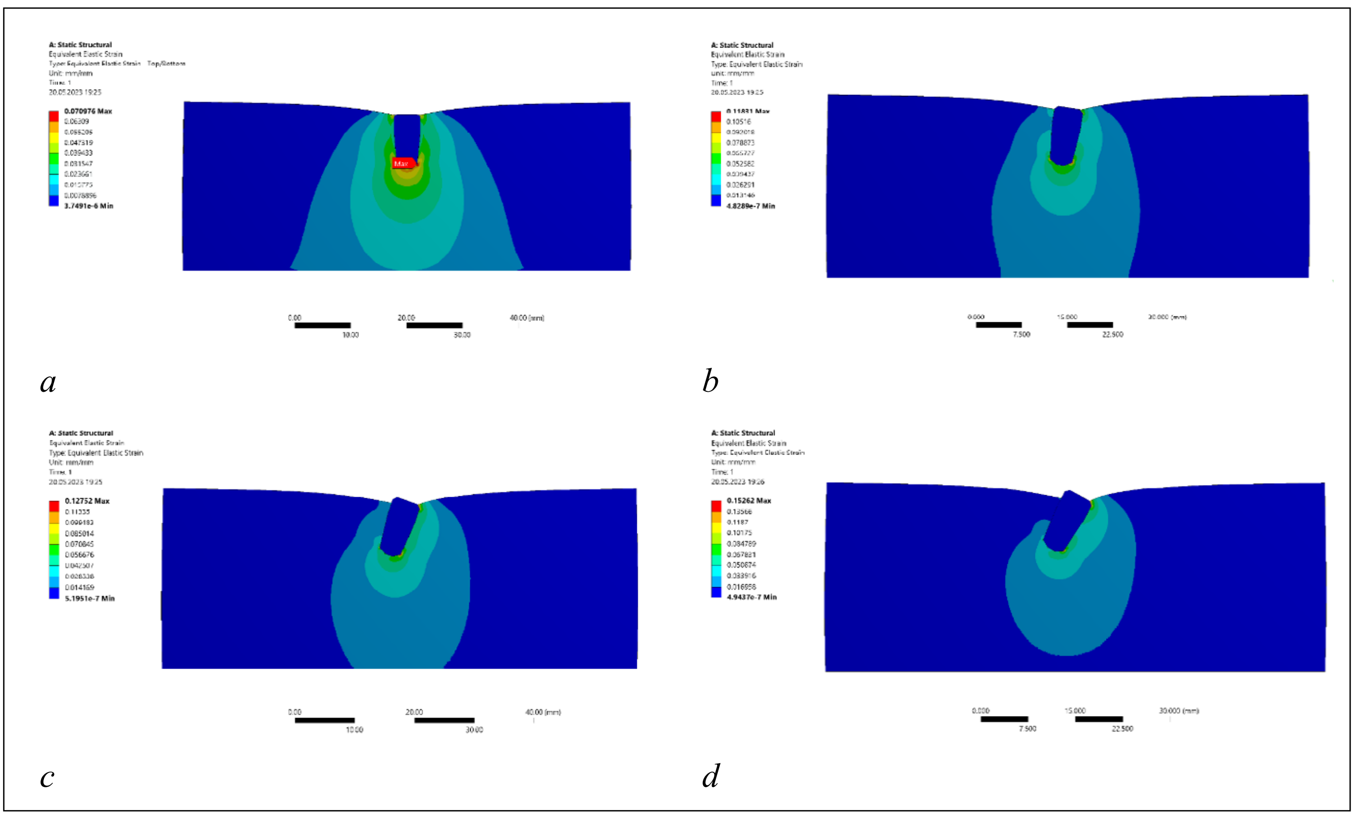

Fig. 4. Stress-strain state of the computational model at a load of 20N with implant tilt angles (a) 0°, (b) 7°, (c)15°, (d) 22°
Picture taken by the authors

CONCLUSIONS

The created numerical model showed satisfactory results. As a result, it can be used to analyze the behavior of the implant in the human jaw and to predict the allowable angles of implant placement if it is impossible to place it at a right angle. The photoelasticity method has also proven to be one of the methods for validating numerical solutions.

Based on the results of the study, the stress distribution fields were obtained and the maximum stresses in the system were determined. The dependence of the stresses on the angle of implant placement was revealed. It was also proved that the maximum stresses do not exceed the allowable ones. Thus, the use of these data will help in choosing the optimal implant placement angle.

This experimental study is important because having an understanding of the load distribution in the superstructure-implant-bone system, which occurs when they are installed at different angles, allows us to plan dental implantation in such a way as to minimize the risk of potential biomechanical complications. The application of digital technologies and navigational methods in implantology enables predictable implant placement at angles that will reduce the risk of biomechanical complications during the functioning of the orthopedic structure supported by dental implants.

REFERENCES

1. Cheng HC, Peng BY, Chen MS et al. Influence of deformation and stress between bone and implant from various bite forces by numerical simulation analysis. Biomed Res Int. 2017;2017:2827953. doi:10.1155/2017/2827953. DOI
2. Baggi L, Cappelloni I, Di Girolamo M et al. The influence of implant diameter and length on stress distribution of osseointegrated implants related to crestal bone geometry: A three-dimensional finite element analysis. J Prosthet Dent. 2008;100(6):422-431. doi:10.1016/s0022-3913(08)60259-0. DOI
3. Lahoti K, Pathrabe A, Gade J. Stress analysis at bone-implant interface of single- and two-implant-retained mandibular overdenture using three-dimensional finite element analysis. Indian J Dent Res. 2016;27(6):597-603. doi:10.4103/0970-9290.199587. DOI
4. Alsrouji MS, Ahmad R, Abdul Razak NH et al. Premaxilla stress distribution and bone resorption induced by implant overdenture and conventional denture. J Prosthodont. 2019;28(2): e764-e770. doi:10.1111/jopr.12954. DOI
5. Pham DQ, Sulyma O, Musiienko O et al. Development of a finite element model for optimization of the bone fixator design. Procedia Struct Integr. 2023;47:945-951. doi:10.1016/j.prostr2023.07.026. DOI
6. Coskunses FM, Tak O. Clinical performance of narrow-diameter titanium–zirconium implants in immediately loaded fixed full-arch prostheses: a 2-year clinical study. Int J Implant Dent. 2021;7(1):51. doi:10.1186/s40729-021-00312-3. DOI
7. Meijer HJA, Slagter KW, Gareb B et al. Immediate single-tooth implant placement in bony defect sites: A 10-year randomized controlled trial. J Periodontol. 2024;95(8):1002-1013. doi:10.1002/JPER.24-0251. DOI
8. Misch CE. Single-tooth implant restoration. In: Misch CE, ed. Dental Implant Prosthetics. St. Louis: Elsevier; 2015:499-552. doi:10.1016/B978-0-323-07845-0.00029-4. DOI
9. Li G, Keyes KM, Huang X. Respond. Am J Public Health. 2018;108(10):e2. doi:10.2105/AJPH.2018.304672. DOI
10. Doroshenko OM, Sirenko OF. Prediction of biomechanical complications in patients with implant supported fixed dental prostheses in different terms of functional loading. Zaporozhye Med J. 2017;(4):482-487. doi:10.14739/2310-1210.2017.4.104923. DOI
11. Herráez-Galindo C, Torres-Lagares D, Martínez-González AJ et al. A comparison of photoelastic and finite elements analysis in internal connection and bone level dental implants. Metals (Basel). 2020;10(5):648. doi:10.3390/met10050648. DOI
12. Thomková B, Marcián P, Borák L et al. Biomechanical performance of dental implants inserted in different mandible locations and at different angles: A finite element study. J Prosthet Dent. 2023;130(6):925-934. doi:10.1016/j.prosdent.2023.10.012. DOI
13. De Stefano M, Lanza A, Faia E, Ruggiero A. A novel ultrashort dental implant design for the reduction of the bone stress/strain: A comparative numerical investigation. Biomed Eng Adv. 2023;5:100077. doi:10.1016/j.bea2023.100077. DOI
14. Akbas O, Greuling A, Stiesch M. The effects of different grading approaches in additively manufactured dental implants on peri-implant bone stress: A finite element analysis. J Mech Behav Biomed Mater. 2024;154:106530. doi:10.1016/j.jmbbm.2024.106530. DOI
15. Goiato MC. Photoelastic stress analysis in prosthetic implants of different diameters: mini, narrow, standard or wide. J Clin Diagn Res. 2014;8(5):268-272. doi:10.7860/jcdr/2014/8489.4902. DOI
16. Sotto-Maior BS, Senna PM, Silva-Neto JP et al. Influence of crown-to-implant ratio on stress around single short-wide implants: a photoelastic stress analysis. J Prosthodont. 2015;24(1):52-56. doi:10.1111/jopr.12171. DOI

CONFLICT OF INTEREST

The Authors declare no conflict of interest

CORRESPONDING AUTHOR
Vitalii V. Parii
Bogomolets National Medical University.
34 Beresteisky Avenue, 88000 Kyiv, Ukraine
e-mail: vitaliy-pariy@ukr.net

ORCID AND CONTRIBUTIONSHIP
Olha S. Musiienko: 0000-0001-8255-3909 A B D E F
Vitalii V. Parii: 0009-0005-2207-5307 A C E F
Oleksii I. Morhun: 0009-0000-9611-1829 B D
Ihor G. Diadik: 0009-0006-2962-2155 D E

A – Work concept and design, B – Data collection and analysis, C – Responsibility for statistical analysis, D – Writing the article, E – Critical review, F – Final approval of the article

RECEIVED: 06.04.2025
ACCEPTED: 22.08.2025