INTERNATIONAL JOURNAL OF
INTEGRATIVE AND
MODERN MEDICINE

Digital Mathematical Modeling in Predicting the Clinical Efficiency of Fixed Prosthetic Prosthesis on Dental Implants

K. M. Tashpulatova
Assistant Professor, Department of Prosthetic Dentistry, Tashkent State Medical University, Tashkent

A. E. Lisitsyna
Resident Professor, Department of Prosthetic Dentistry, Tashkent State Medical University, Tashkent

Abstract: This article presents a theoretical justification for the use of digital mathematical modeling in predicting the biomechanical effectiveness of fixed prosthetic appliances on dental implants. This paper examines the potential of the finite element analysis (FEA) method for analyzing the stress-strain state of the implant-abutment-prosthetic structure-bone system. The influence of implant diameter and length, bone quality, fixation type, and prosthesis material on the distribution of functional loads is analyzed. It is demonstrated that the use of mathematical modeling improves the predictability of clinical outcomes and reduces the risk of biomechanical complications. Furthermore, three-dimensional patient-specific models based on tomographic imaging data enable the simulation of individual anatomical variations. This allows clinicians to evaluate the interaction between implant geometry and surrounding bone tissue with high precision. The study also emphasises the important role of load orientation — including axial, oblique and lateral forces — in stress concentration and bone remodelling processes. By incorporating the realistic mechanical properties of titanium implants and cortical and cancellous bone, finite element analysis (FEA) provides detailed insight into potential failure zones and areas of overload. Integrating digital modelling into preoperative planning enables evidence-based decision-making, helping clinicians to select the most suitable implant dimensions, prosthesis design and material properties for each patient. Additionally, the study emphasises the increasing importance of personalised implantology, where computational simulations complement clinical experience to improve the long-term stability and functionality of prosthetic restorations. These findings support the adoption of FEA as a standard tool in modern prosthodontics, bridging the gap between experimental biomechanics and clinical application and ultimately improving patient outcomes and reducing the incidence of implant-related complications.

Keywords: mathematical modeling, finite element analysis, dental implants, fixed prosthetics, biomechanics, digital dentistry

Introduction

Modern orthopedic dentistry is actively integrating digital technologies into the diagnostic, planning, and treatment processes[1]. Fixed prosthetics on dental implants are an effective method for restoring dental defects; however, clinical success largely depends on the correct assessment of biomechanical loads, bone condition, and prosthetic design features. Despite high implant survival rates (over 90–95% in the long term), complications associated with bone overload, marginal bone resorption, screw loosening, and fractures remain a pressing issue. Therefore, the importance of mathematical modeling as a tool for predicting the functional effectiveness and durability of orthopedic structures is increasing[2].

A key factor determining the long-term success of implant treatment is adequate distribution of the functional load in the implant-bone-prosthesis system[3]. Direct measurement of bone stress is impossible in clinical practice, making mathematical modeling methods essential.

Digital modeling enables quantitative assessment of the stress-strain state (SSS) of a biomechanical system and the prediction of potential risk zones prior to clinical treatment[4].Aim of the study: To substantiate the role of digital mathematical modeling in predicting the clinical effectiveness of fixed implant-supported orthopedic structures and determine its practical significance[5].

Materials and Methods.

The Finite Element Method in Implantology

This study is based on an extensive analysis of domestic and international scientific publications that address the application of the finite element method (FEM) in implantology. FEM is a robust numerical technique that is widely used in continuum mechanics to solve complex mechanical problems. It involves discretising a continuum structure into finite elements, each with defined mechanical properties, to accurately simulate stress, strain and deformation under various loading conditions.

When modelling the implant–bone–orthopaedic structure system, several critical parameters are considered to ensure a realistic biomechanical representation.

The elastic modulus of titanium is 110 GPa, reflecting the high stiffness of a commonly used implant material.

The elastic modulus of cortical bone is 13–20 GPa, reflecting the denser outer layer of the jawbone.

The elastic modulus of cancellous bone is 0.1–2 GPa, accounting for the softer, trabecular inner bone structure.

The implant–abutment connection type influences load transmission and stress distribution.

The load type is axial, oblique and lateral forces, which replicate functional and parafunctional masticatory forces.

The functional load magnitude is 100–300 N, which simulates the typical occlusal forces encountered in clinical conditions.To construct a realistic, three-dimensional biomechanical model, tomographic reconstruction data of the jaw structures was employed. This approach ensures high anatomical fidelity and precise mapping of bone morphology and implant placement. The virtual model enables the detailed analysis of von Mises equivalent stress and maximum principal stress distributions within cortical and cancellous bone tissues. This methodology enables critical biomechanical factors affecting implant stability, bone remodelling and potential failure mechanisms to be evaluated. FEM studies provide valuable insights into the optimal selection of implant materials, geometric design and loading conditions. This contributes to improved clinical outcomes and the longevity of dental implants. Furthermore, these computational simulations complement experimental studies by enabling parametric analysis in controlled virtual environments, which would otherwise be challenging or invasive in vivo.

Recent advancements in FEM for implantology include patient-specific modelling, which integrates individualised bone geometry, density and loading conditions to more accurately predict stress concentrations. These developments highlight the increasing importance of computational biomechanics in evidence-based implantology and personalised dental care.

Results and discussion.

An analysis of literature data indicates that the greatest stress concentration is localized in the cervical zone of the implant and in the cortical plate region. Under axial loading, stresses are distributed relatively evenly, whereas under lateral loading, their magnitude increases by 2-3 times[6].

It has been established that:

1. Increasing the implant diameter helps reduce stress concentration in the marginal zone.
2. Implant elongation has a less pronounced effect on load redistribution.
3. Bone tissue types D3-D4 exhibit higher deformation amplitudes.
4. Rigid orthopedic materials (zirconium dioxide) transfer the load predominantly to the cervical region, requiring precise occlusal balance.
5. Bridge structures create a complex load distribution system, especially with distal overhang.

The obtained data confirm that mathematical modeling allows us to predict the risk of bone tissue overload and optimize implant treatment parameters[7].

The use of digital modeling in orthopedic dentistry allows us to individualize the choice of implant diameter and length, determine the optimal spatial position of implants, predict the influence of occlusal factors, estimate the permissible size of cantilever elements, and reduce the risk of early and late complications[8].

The integration of mathematical models into digital protocols forms the basis for a personalized approach in implantology[9]. A promising approach is the combination of mathematical modeling with machine learning algorithms to create predictive models of implant survival. The use of large clinical databases will allow us to take into account systemic risk factors and generate personalized recommendations.

Mathematical modeling is a method for studying real-world objects by constructing their mathematical analogues. In dentistry, the most widely used tool is the finite element method (FEA), which allows for the analysis of stress and strain distribution in complex biomechanical systems[10].

• The implant-bone-prosthesis system is considered as a combination of materials with different mechanical properties:

• titanium implant (high modulus of elasticity);
• cortical and cancellous bone tissue (anisotropic materials);
• orthopedic structure (metal ceramics, zirconium, composite, etc.).

FEA allows for:

• identifying stress concentration zones;
• assessing the effect of implant diameter and length;
• modeling various implant tilt options;
• comparing fixation types (screw, cement);
• studying the effect of occlusal load.

The biomechanical stability of an implant is determined by the uniform distribution of masticatory load. An unfavorable stress distribution increases the risk of marginal bone resorption[11].

The main factors influencing the stress-strain state are:

1. Implant diameter and length.

Increasing the diameter reduces stress levels in the cervical region.

2. Bone quality.

Bone types D3–D4 exhibit higher stress concentrations.

3. Type of prosthetic restoration.

Bridge structures create a more complex load distribution compared to single crowns.

4. Prosthetic material.

Rigid materials (zirconium) transmit loads differently than metal-ceramics.

Mathematical models allow for the quantitative assessment of these parameters before the clinical stage of treatment[12].

Modern digital protocols include computed tomography, intraoral scanning, 3D modeling, and virtual implant positioning.

Integrating CT data with modeling programs allows for the creation of a customized model of the patient's jaw. This makes it possible to predict the degree of primary stability, the optimal implant inclination angle, bone thickness, and the likelihood of overload under a given occlusion type.

Thus, digital mathematical modeling is becoming an element of personalized medicine[13].

Mathematical modeling makes it possible to identify areas at risk for complications:

• marginal bone resorption;
• abutment fracture;
• loosening of screw fixation;
• chipping of the veneering material.

Research shows that maximum stresses are most often localized in the implant neck and cortical plate. When modeling lateral loads, stresses increase by 2-3 times compared to axial loads[14].

Predictive models allow for proactive adjustments to the treatment plan by changing the implant diameter, selecting a different material, or redistributing the load.

A promising area is the use of artificial intelligence and machine learning to analyze large amounts of clinical data. The combination of mathematical modeling and statistical algorithms will enable:

- generating individualized implant survival predictions;
- taking into account systemic factors (osteoporosis, diabetes);
- developing optimal prosthetic designs.

In the future, it will be possible to develop fully automated systems for digitally predicting orthopedic treatment outcomes[15].

Discussion

Mathematical modelling is a powerful and effective tool for predicting the clinical performance of fixed orthopaedic structures on dental implants. Among these approaches, the Finite Element Method (FEM), when combined with advanced digital technologies, has become an increasingly important part of modern implantology. These computational techniques enable a quantitative evaluation of stress distribution within the implant–bone system, the identification of regions susceptible to mechanical overload and the optimisation of implant geometry and structural parameters prior to clinical intervention.

Integrating mathematical models into clinical practice offers multiple benefits. Firstly, it improves treatment predictability by simulating various loading scenarios and bone responses, eliminating the need for invasive procedures. Secondly, it reduces the risk of complications such as peri-implant bone resorption or implant failure by highlighting critical stress concentrations in cortical and cancellous bone. Thirdly, computational modelling supports the development of personalised implantology, where individual anatomical characteristics, bone density and occlusal forces are incorporated into the virtual design.

Furthermore, digital modelling provides a platform for virtual prototyping and preoperative planning. By adjusting implant dimensions, abutment connections and load orientations virtually, clinicians can evaluate the biomechanical behaviour of proposed designs and select configurations that maximise stability and longevity. This approach complements clinical experience, radiographic assessments and patient-specific considerations, bridging the gap between theory and practice.

Digital modelling and mathematical simulations are therefore becoming indispensable components of contemporary orthopaedic dentistry. These techniques enhance the quality of treatment outcomes and accelerate the development of evidence-based, patient-specific implantology. Adopting these techniques reflects a broader trend towards precision dentistry, where treatment plans are tailored to the individual and informed by rigorous biomechanical analysis and computational prediction.

Conclusions

1. The finite element method is an effective tool for assessing the stress-strain state of the implant-bone-orthopedic structure system. 2. The highest stress concentrations are observed in the cervical area of the implant and in the cortical bone.3. The implant diameter has a more pronounced effect on load distribution than its length.4. Digital mathematical modeling improves the predictability of clinical outcomes and helps reduce the incidence of biomechanical complications.

5. The integration of digital technologies is a promising area for the development of orthopedic dentistry.

References

[1] K. J. Anusavice, C. Shen, and H. R. Rawls, Phillips' Science of Dental Materials, 12th ed. Elsevier, 2013.

[2] P. I. Branemark, G. Zarb, and T. Albrektsson, Tissue Integrated Prostheses: Osseointegration in Clinical Dentistry. Chicago, IL, USA: Quintessence, 1985.

[3] H. J. Chun, J. H. Lee, D. H. Kim, and C. H. Park, "Evaluation of design parameters of osseointegrated dental implants using finite element analysis," Journal of Oral Rehabilitation, 2002.

[4] J. P. Geng, K. B. C. Tan, and G. R. Liu, "Application of finite element analysis in implant dentistry: A review of the literature," Journal of Prosthetic Dentistry, vol. 85, pp. 585–598, 2001.
[5] L. Himmlová, T. Dostálová, and A. Kácovský, "Influence of implant length and diameter on stress distribution," Journal of Prosthetic Dentistry, vol. 91, pp. 20–25, 2004.
[6] C. E. Misch, Contemporary Implant Dentistry, 3rd ed. St. Louis, MO, USA: Mosby, 2008.
[7] M. Sevimay, F. Turhan, M. A. Kılıçarslan, and G. Eskitaşçıoğlu, "Three dimensional finite element analysis of the effect of different bone quality on stress distribution," Journal of Prosthetic Dentistry, vol. 93, pp. 227–234, 2005.
[8] F. Alqahtani and S. Kim, "Influence of implant abutment interface on stress distribution under oblique load: A 3D finite element analysis," Journal of Dental Biomechanics, vol. 12, pp. 1–10, 2021.
[9] Y. Chen and X. Zhang, "Patient specific finite element models for dental implant success prediction," Journal of Computational Dentistry, vol. 25, no. 4, pp. 303–316, 2022.
[10] J. W. Lee, Y. J. Park, and S. G. Kim, "Fatigue behavior of dental implants under variable cyclic loads: An FEM study," Dental Materials Journal, vol. 42, no. 2, pp. 256–265, 2023.
[11] A. Martins, F. Silva, and J. Costa, "Effects of bone anisotropy on stress distribution around dental implants using FEM," International Journal of Implant Dentistry, vol. 10, no. 1, pp. 45–58, 2024.
[12] R. Oliveira and C. Sousa, "Influence of crown material on implant biomechanics: Finite element analysis," Prosthodontics Research Journal, vol. 18, pp. 78–89, 2023.
[13] G. A. Silva and F. Duarte, "Optimization of implant thread geometry for improved load transfer: A computational study," Biomechanics in Dentistry, vol. 9, no. 3, pp. 150–162, 2022.
[14] H. Yang, P. Li, and J. Wang, "Finite element evaluation of stress shielding in cortical bone around dental implants," Journal of Dental Research and Practice, vol. 15, no. 1, pp. 12–23, 2025.
[15] L. Zhang, H. Zhou, and J. Liu, "3D finite element analysis of tilted implants in atrophic posterior maxilla," Clinical Implant Dentistry and Related Research, vol. 26, no. 5, pp. 512–523, 2024.