

PACS: 62.20.F

ISSN 1729-4428 (Print)
ISSN 2309-8589 (Online)

B.I. Hrynkiv¹, L.O. Borushchak², Z.R. Ozhogan¹, L.V. Miziuk¹, R.Z. Ozhogan¹,
I.V. Viklyuk¹

Computer modeling and research of the stress-deformed state of the system "dental implant - bone tissue"

¹*Ivano-Frankivsk National Medical University, Ivano-Frankivsk, Ukraine, b.grynkiv@gmail.com*

²*Ivano-Frankivsk National Technical University of Oil and Gas, Ivano-Frankivsk, Ukraine, liubomyr.borushchak@nung.edu.ua*

The stress–strain state of the “dental implant–bone tissue” system was determined using three-dimensional computer models created in the SolidWorks 2024 parametric solid modeling software with the application of the finite element method. The actual investigations were performed using the Simulation package. Parametric solid models were developed in SolidWorks 2024, including a fragment of the jawbone with a dental implant and prosthetic elements. The physical and mechanical properties of the prosthetic elements (crown and abutment), the dental implant, as well as the cortical and trabecular bone tissues of the jaws were applied in the analysis. The analysis of the stress–strain state was conducted under conditions corresponding to average strength and density values of the trabecular bone tissue of the jaws. The simulation of maximally asymmetric masticatory loading applied to the prosthetic elements and the dental implant made it possible to identify unfavorable functional conditions of the crown–implant–bone tissue system.

Keywords: stress; deformation; osseointegration; three-dimensional computer model; abutment teeth; dental implant; bone density; trabecular bone tissue; finite element method (MCE).

Received 25 January 2026; Accepted 06 May 2026; Published 22 June 2026.

Introduction

The process of osseointegration of a dental implant primarily depends on its biomechanical behavior because the nature of the stress distribution in dental implants is completely different from that of a natural tooth. Natural tooth is characterized by the presence of a periodontal ligament which acts as a shock absorber for occlusal forces. The main factor in the success or failure of osseointegration of a dental implant is the nature of the load transfer from the dental implant to the adjacent alveolar bone tissue [1].

The process of osseointegration of dental implants is influenced by a number of factors: the clinical skills of the doctor, implant placement technique and surgical protocol. Successful osseointegration requires careful consideration of stress distribution within the implant and jawbone, as well as their biomechanical interaction [2].

Occlusal overload is considered the primary cause of

peri-implant bone loss, often resulting in the failure of prosthetic restorations and dental implants. Therefore, biomechanics is a main factor in predicting implant disintegration and directly affects the long-term survival of dental implants and prosthetic reconstructions [3].

The demand for dental rehabilitation using dental implants has increased annually due to traumatic injuries and periodontal diseases. The number of unsuccessful implant integration cases has increased accordingly. Methods for modeling the behavior of a dental implant in terms of stress and deformation in the jaw bone tissue under the influence of masticatory forces are necessary to predict the behavior of a dental implant during the osseointegration process. One of the most widely applied approaches for this purpose is the finite element method (FEM) [4].

The finite element method is used to determine the state of stress, strain and displacement in dental implants and surrounding bone tissue. It enables analysis of

biomechanical factors affecting implant behavior before and after osseointegration and supports the optimization of dental implant design.

The study of the osseointegration process remains an important research topic in dental implantology [4].

The finite element method (FEM) provides valuable insights into the mechanical behavior and stability of implant systems and allows highly accurate predictions of structural responses and potential deformation within both the implant and adjacent bone tissue [5].

This method allows to determine the state of stress and displacement immediately after implant placement, during the osseointegration process and after the osseointegration process until complete healing [6].

Computer-based software is used to convert both simple and complex structures into meshes. The resulting models consist of nodes, elements and predefined boundary conditions. During the modeling process, loads are applied to certain elements or nodes and then stresses and displacements are evaluated to perform the modeling analysis. This method is widely used in dentistry: to determine the design and shape of restorations, removable and fixed prostheses, posts, retention pins, dental implants, etc. [7].

The advantage of this method is the ability to predict the mechanical behavior of a dental implant without the need for destructive testing or in vivo experimentation [8].

However, the finite element method (FEM) has a number of disadvantages that must be considered when interpreting FEM results for making clinical decisions, including the inability to simulate biological factors such as temperature, pH, and biofilm formation and the use of isotropic materials [9].

Nevertheless, FEM enables detailed prediction of spatial stress distribution in areas where implants contact with cortical bone, as well as in the area around the apex of dental implants in trabecular bone. Factors that influence the regulation of biomechanical loading include the size of the functional surface and the type of applied force. The finite element method (FEM) can provide accurate quantitative data at any point in the mathematical structure. Therefore, the finite element method (FEM) is a useful analytical tool in the modern dental implantology. [10].

I. Materials and methods

The analysis of the obtained data will make it possible to compare the calculated values of static stresses, displacements of elementary internal volumes of the model and the degree of their deformation with the permissible stress limits, thereby enabling conclusions to be derived regarding the nature of bone tissue response to the mechanical loading caused by the dental implant. The comparative results will indicate the probability level of trabecular bone tissue failure which represents the most vulnerable component of the system.

The studies were conducted using computer-based models of a jawbone fragment, the dental implant and prosthetic components.

1.1. Research tools

Three-dimensional computer models were created in the solid-state parametric modeling program Solid Works 2024 and the actual research was performed using the Simulation package. The application is based on the finite element method.

The values of equivalent static stresses are displayed in the legend in megapascals (MPa) according to Mises (third strength theory).

The values of displacements of internal volumes are displayed in mm and the values of their deformation - ESTRN - in units of equivalent deformation of the body relative to the three coordinate axes.

1.2. Initial data for conducting research

The initial data for the study include:

- the geometric structure of the jawbone segment, the dental implant and the prosthetic elements;
- the design of the prosthetic elements (the dental implant, abutment and ceramic crown);
- the magnitude of the masticatory load force acting on the crown and the remaining prosthetic elements;
- the physical and mechanical properties of the jawbone components (cortical and trabecular bone) and prosthetic elements.

The fastening element (screw) which ensures a reliable and rigid connection between the abutment and the implant was excluded from the model for the purpose of reasonable simplification of the investigated computer model.

The jawbone fragment at the site of dental implant placement represents a composite structure consisting of a relatively dense, hard and strong cortical layer with a thickness of 0.6–1.25 mm at the surface while the internal volume is composed of porous trabecular bone tissue characterized by comparatively lower hardness and strength.

The results of experimental studies of the properties of the trabecular and cortical components of the jaw bones are in slightly different, overlapping ranges. Generalized material properties are presented in Table 1.

Table 1.
Physical and mechanical parameters of bone structures

Parameter	Symbol	Trabecular bone	Cortical bone
Modulus of elasticity	E, MPa	2.5÷150	(2.5÷20)×10 ³
Poisson's ratio	μ	0.23÷0.49	0.25÷0.35
Density	ρ, g/cm ³	0.27÷1.1	1.4÷2.1
Ultimate Tensile Strength	σ, MPa	0.25÷32	22÷125
Ultimate Compressive strength	σ, MPa	0.3÷12.5	15.5÷82

The average values of the trabecular bone tissue parameters were established and presented in Table 2 for the purpose of conducting studies.

Table 2.

Selected values of physical and mechanical parameters of trabecular bone tissue of the jaws

Parameter	Symbol	Average values
Modulus of elasticity	E, MPa	65
Poisson's ratio	μ	0.3
Density	ρ , g/cm ³	0.6
Ultimate Tensile Strength	σ , MPa	12.0
Ultimate Compressive strength	σ , MPa	5.5

We determine that during the preparation process the implant site is widened in such a way that the superficial (cortical) layer has almost no contact with the prosthetic elements, which facilitates proper placement of the dental implant. The analysis of the numerical values of the strength parameters of the cortical structure shows that the elastic modulus is approximately two orders of magnitude higher than that of trabecular bone tissue. Therefore, the hardness is also higher which causes much smaller deformations of the cortical layer under the influence of force factors. The averaged values of these parameters are adopted for the computer simulations which are presented in Table 3.

Table 3.

Selected values of physical and mechanical parameters of cortical bone tissue of the jaws

Parameter	Symbol	Cortical bone
Modulus of elasticity	E, MPa	13.5×10^3
Poisson's ratio	μ	0.25
Density	ρ , g/cm ³	1.65
Ultimate Tensile Strength	σ , MPa	65
Ultimate Compressive strength	σ , MPa	45

We set the masticatory load force acting on the crown to 100 N which is based on research data.

Table 4.

Physical and mechanical parameters of other elements of a dental implant-based prosthetic structure

Parameter	Symbol	Implant	Abutment	Crown
Modulus of elasticity	E, MPa	120×10^3	115×10^3	220×10^3
Poisson's ratio	μ	0.32	0,3	0,22
Density	ρ , g/cm ³	4.37	4,43	3,0
Ultimate Tensile Strength	σ , MPa	937	800	2300
Ultimate Compressive strength	σ , MPa	930	810	551

1.3. Creating a solid 3D model

The first stage to perform the research is to create a

solid-state parametric model of a jawbone fragment with a dental implant and full-size prosthetic elements using the Solid Works 2024 program.

Static studies of the stress-strain state are performed in the Simulation application of the specified program in the Static category.

The basis of the prosthetic structure included the KDA0F4008 implant which was chosen for premolar replacement with a diameter of $\varnothing 4.25$ mm and a length of 10.0 mm from the Spanish company GMI Dental Implantology, S.L. with an abutment and a locking screw (Fig. 1).

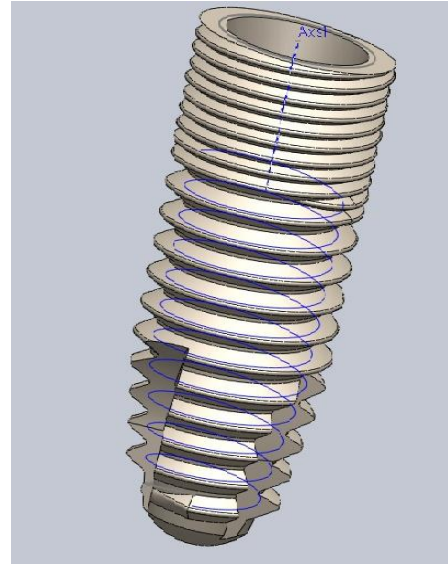


Fig. 1. The model of a dental implant.

We will make the following assumption during the modeling process. The abutment is based in the dental implant with a high-precision conical surface and is fixed in the implant with a screw which exclude any mutual movements. Therefore, we will exclude the fixing screw from the prosthetic structure in order to simplify the model.

The thickness of the cortical layer of the jaw bone will be taken as 1.25 mm according to average statistical data.

The configuration and dimensions of the premolar crown are set for the same reasons.

The model of a dental implant is presented in Fig. 2.

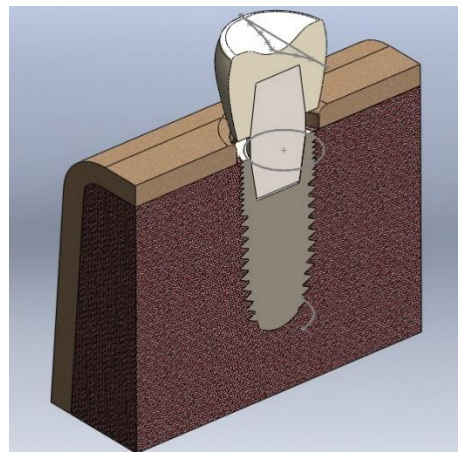


Fig. 2. Longitudinal section of the assembly model of the studied jawbone-implant system.

1.4. Research methodology

The algorithm for conducting a force study includes the following steps. At first, we will establish the direction and place of application of the masticatory load force to the crown. Its greatest concentration in the case of hard pieces of food usually falling on the lateral areas of the upper part of the tooth (shown by purple arrows). Moreover, such a choice makes it possible to determine the nature of the stress-strain state of the model under the most unfavorable loading condition.

In the second stage we replace the rest of the jaw bone with rigid (fixed) ligaments along defined surfaces. The two lateral vertical and horizontal base surfaces of the imaginary jaw sections are presented in our case on Fig 3.

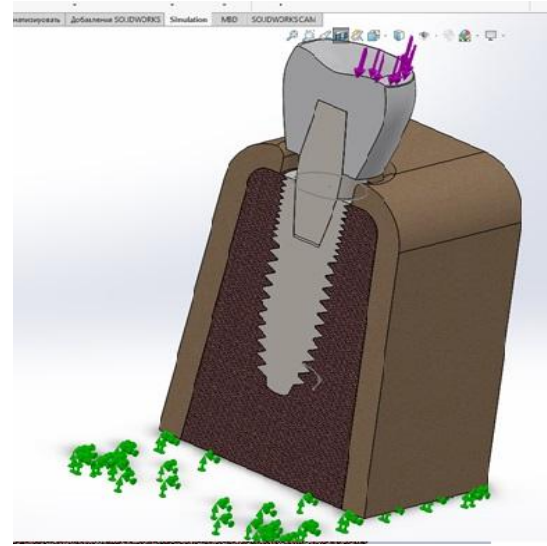


Fig. 3. The model under study with an applied masticatory load of 100 N and imposed equivalent rigid constraints on the surfaces.

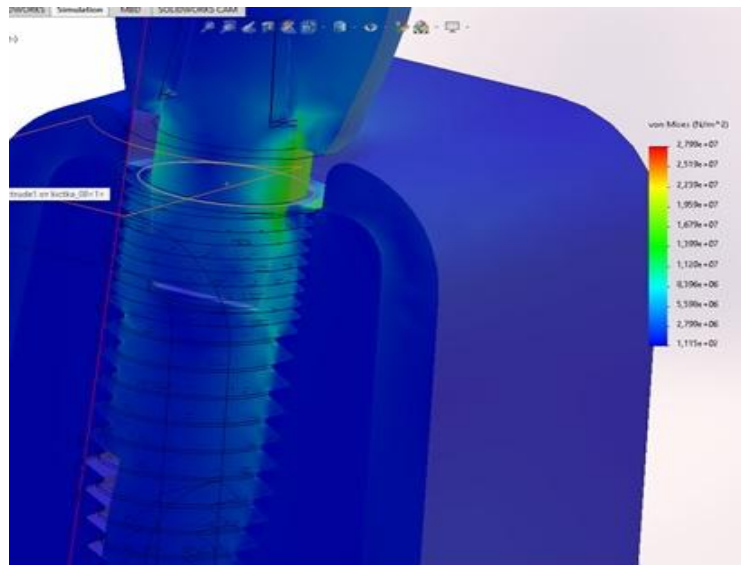


Fig. 4. Volumetric stress plot at average values of trabecular bone tissue strength parameters.

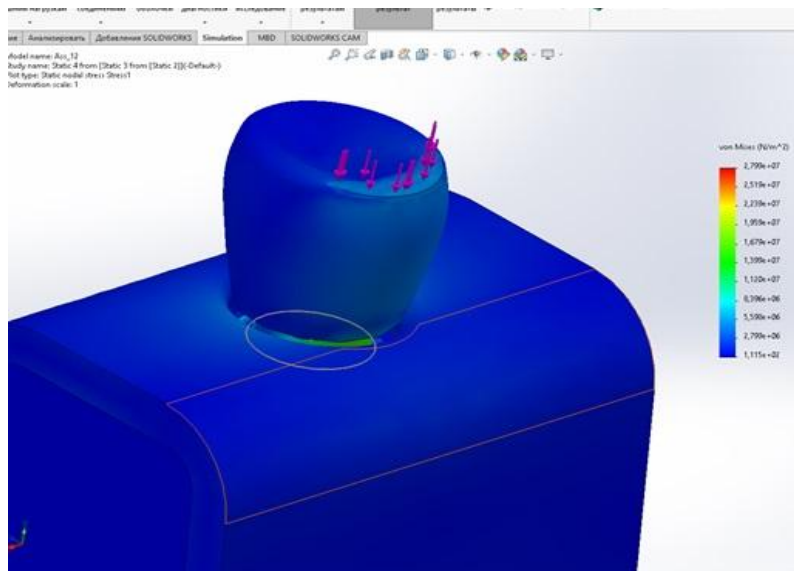


Fig. 4,a. Volumetric stress plot in the surface layer of cortical bone tissue at average strength parameters of trabecular bone tissue.

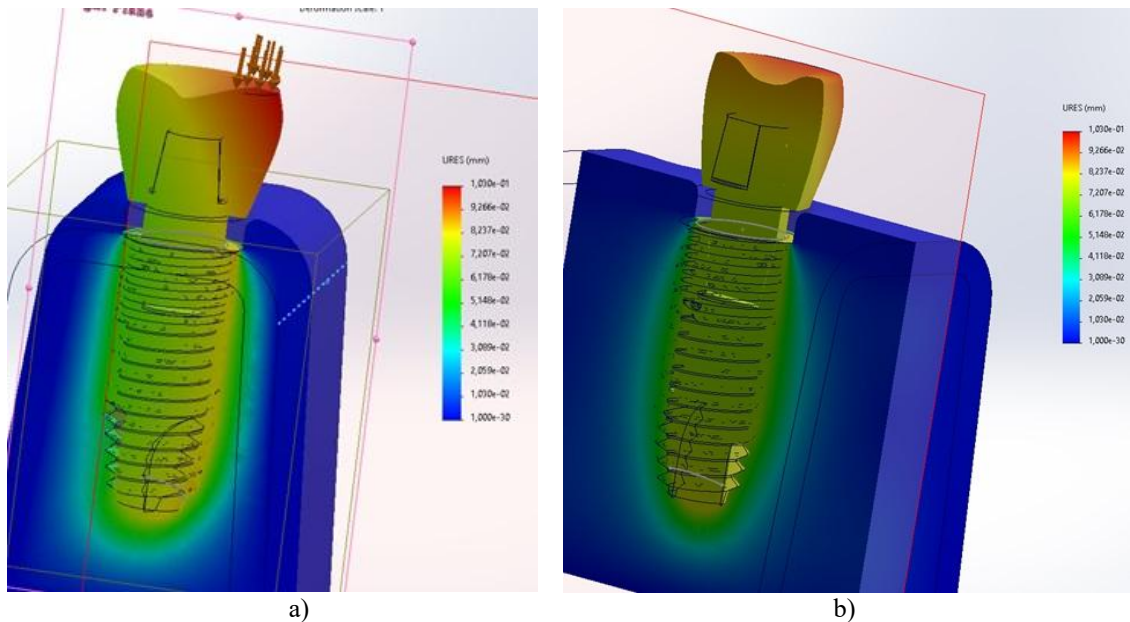


Fig. 5. Volumetric displacement plot in transverse (a) and longitudinal (b) sections at average strength parameters of the trabecular bone tissue.

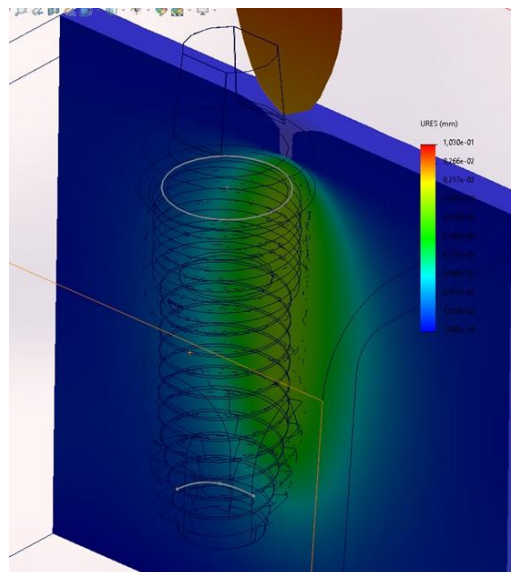


Fig. 5.a. Volumetric displacement plot in a longitudinal section of the volumes surrounding a dental implant at average strength parameters of the trabecular bone tissue.

The created 3D model is an assembly, so it is necessary to define the contact conditions between individual components of the model, including the possibility of their relative displacements or constraints. Based on the dental implant design, the placement method and the jawbone structure, any relative movement of the components is excluded in this study. Therefore, a global contact between the components is defined which eliminates their mutual mobility.

II. Results and discussion.

The analysis of the stress-strain state was conducted under conditions of average strength and density of the jaw trabecular bone tissue.

Average values of trabecular bone strength parameters

The obtained results are described in more detail below. We determine that the equivalent stresses increase to 15 MPa in the upper girdling volumes of the trabecular bone tissue (Fig. 4). The stresses decrease to 9 MPa as they approach the base of the dental implant and at the base the stresses are in the range of 0.14 – 0.18 MPa. The minimum equivalent stress values recorded by the software amount to 0.09 MPa in the thread-adjacent zone.

An evident explanation for this behavior is that the bone tissue volumes adjacent to the dental implant in its threaded region are constrained by the thread profile and therefore cannot undergo significant deformation when the upper layers of trabecular bone tissue are essentially open and unlike the lower layers, experience tensile deformation (see Fig. 6). The trabecular bone tissue, which is porous and relatively soft, appears to separate from the upper cortical layer.

At the same time, the equivalent stresses in the

annular peri-implant region decrease uniformly from a maximum value of approximately 5–7 MPa to a minimum of 0.12–0.13 MPa across almost the entire upper zone.

We determined that the maximum downward displacement of the upper part of the crown reaches 0.103 mm. The displacement of the elementary volumes of trabecular bone tissue ranges from 0.075 to 0.09 mm and attains its highest values in the vicinity of the implant's threaded surface which causes a downward pulling effect. The displacements gradually decrease to 0.032–0.052 mm with increasing distance from the implant body (Figures 5 and 5a).

Figure 6 demonstrates that the most significant deformation occurs in the band-like volumes of the trabecular bone within the gap between the implant top and the outer cortical layer of the jawbone, ranging from 0.26 to 0.29 ESTRN units. Minor deformations are observed along the spiral of the implant thread crest between 0.08 and 0.12 ESTRN units. However, deformations increase to 0.18–0.22 ESTRN specifically along the crest of the first (bottom) thread turn. The minimum value is 0.06 ESTRN units. This distribution of deformation factors results from the increased pressure area on the bone tissue along the first thread turn. These values correlate well with the magnitudes of equivalent stresses and displacements.

The nature of the relative volumetric deformations of the system is examined (Fig. 6), as this criterion is potentially critical for predicting the failure probability of trabecular bone tissue. This assumption is based on the fact that this bone tissue is sufficiently porous and less rigid, allowing it to cushion the impact of masticatory loads during biting or chewing.

The obtained computer modeling data are compared with the natural 'tooth-periodontium-bone tissue' system. It should be noted that bone tissue is softer and more compliant than the materials of the crown, abutment, and the dental implant itself; therefore, it also undergoes

elastic deformation under load. Besides, periodontal tissue is a natural shock absorber of loads. Consequently, after the placement of a dental implant, the jawbone tissue is subjected to sharp dynamic loads from the latter.

Conclusions

Computer simulations with eccentric masticatory load application to the artificial crown surface allow for identifying the most probable zones of equivalent stress concentration, their values and the areas of maximum deformation in the trabecular bone tissue. We have created conditions for the maximum asymmetric application of masticatory load force to the prothetic elements and the dental implant which allows us to identify unfavorable conditions for the functioning of the crown-implant-bone tissue system.

The distribution and configuration of internally stressed volumes and deformation zones in trabecular bone tissue demonstrate that the most heavily loaded and maximally deformed areas are the annular volumes surrounding the dental implant, specifically near the bone surface and at the implant base.

The volumes of bone tissue near the surface of the dental implant thread evenly surround the latter and have a small thickness (0.12–0.25 mm). We determined that in the specified zone there are tensile stresses in the spiral zone above the crest of the implant thread and joint stresses from the lower edge. During mastication, the location of masticatory load concentration shifts along the crown surface. Accordingly, the stress-strain state of the jawbone changes.

The potentially dangerous zone of trabecular bone tissue, characterized by low strength and elasticity, experiences alternating compressive and tensile stresses and deformations. This leads to loosening of the dental

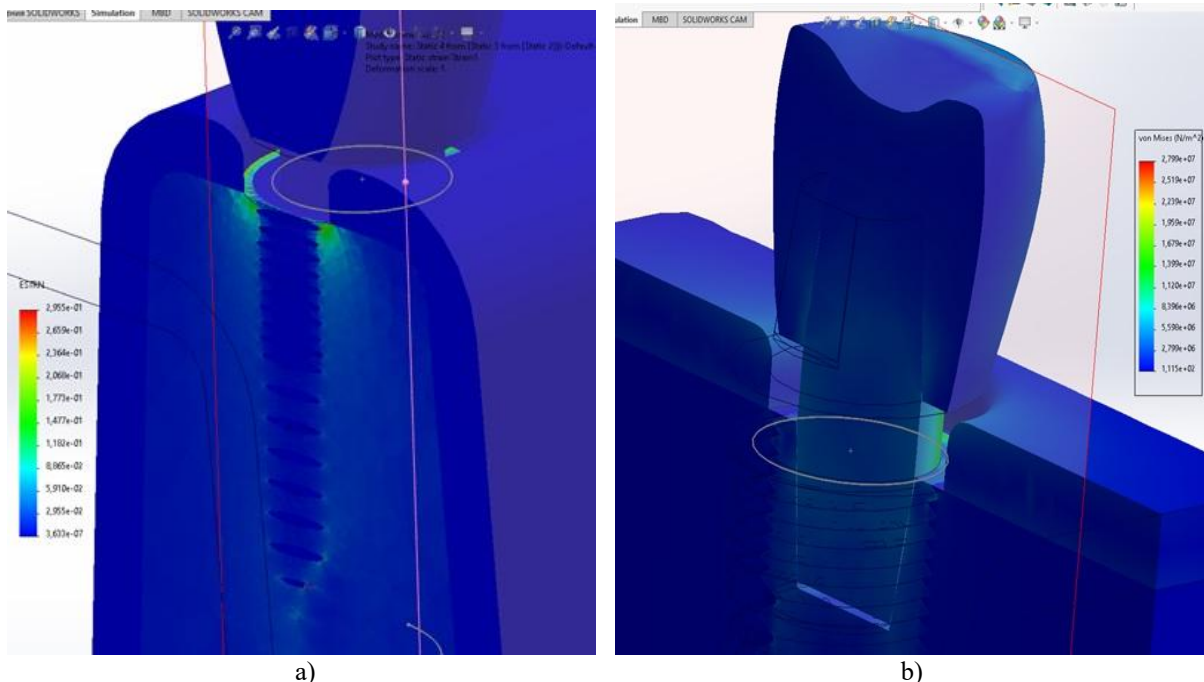


Fig. 6. Volumetric strain plot of model elements at average strength parameters of trabecular bone tissue (a – transverse section, b – longitudinal section).

implant, weakening of its fixation with the destruction of adjacent bone tissue volumes. This may be particularly significant for patients with impaired strength and elasticity characteristics of the jaw trabecular bone tissue.

Conflicts of interest. *There are no conflicts of interest.*

Hrynkiv B.I. – postgraduate student at the Department of Orthopedic Dentistry;
Borushchak L.O. – Ph.D., Associate Professor of the

Department of Computerized Mechanical Engineering;
Ozhogan Z.R. – Dr.Med.Sci., Professor, Honored Worker of Science and Technology of Ukraine, Head of the Department of Orthopedic Dentistry;
Miziuk L.V. – Ph.D., Associate Professor of the Department of Orthopedic Dentistry;
Ozhogan R.Z. – Ph.D., Assistant Professor, Department of Dentistry, Postgraduate Education;
Viklyuk I.V. – Ph.D., Assistant Professor of the Department of Orthopedic Dentistry;

- [1] M.S. Reddy, R. Sundram, H.A.E. Abdemagyd, *Application of Finite Element Model in Implant Dentistry: A Systematic Review*. Journal of pharmacy & bioallied sciences, 11(2), 85 (2019); https://doi.org/10.4103/JPBS.JPBS_296_18.
- [2] V.V. Dam, H.A. Trinh, D.T. Dung, T.D. Hai, *Applications of Finite Element in Implant Dentistry and Oral Rehabilitation*. The Open Dentistry Journal, 15(3), 392 (2021); <https://doi.org/10.2174/1874210602115010392>.
- [3] M.P.M. Silveira, L.M. Campaner, M.A. Bottino, R.S. Nishioka, A.L.S. Borges, J.P.M. Tribst, *Influence of the dental implant number and load direction on stress distribution in a 3-unit implant-supported fixed dental prosthesis*. Dental and medical problems, 58(1), 69 (2021); <https://doi.org/10.17219/DMP/130847>.
- [4] D.C. Costa, J. Belinha, R.M. Natal Jorge, *Biomechanical Simulation of a Dental Implant using Finite Element Method Analysis*. 2019 IEEE 6th Portuguese Meeting on Bioengineering (ENBENG), Lisbon, Portugal, 1- (2019); <https://doi.org/10.1109/enbeng.2019.8692548>.
- [5] S.A. Elsheikh, M.I. El-Anwar, T. Hong, C. Bourauel, A. Alhotan, N.M. Anany, I.G. Shahin, A.H. Diab, T.M. Elshazly, *Biomechanical Analysis of Various Connector Designs of Dental Implant Complex: A Numerical Finite Element Study*. International dental journal, 75(4), 100873, (2025); <https://doi.org/10.1016/j.identj.2025.100873>.
- [6] I. Roatesi, S. Roatesi, *Modeling of Dental Implant Osseointegration Progress by Three- Dimensional Finite Element Method*. Applied Sciences, 10(16), 5561, (2020); <https://doi.org/10.3390/app10165561>.
- [7] L.A. Sabri, F.A. Hussein, A.R. Al-Zahawi, B.Y. Abdulrahman, K.N. Salloomi, *Biomechanical finite element analysis of a single implant threaded in anterior and posterior regions of maxilla bone*. Indian Journal of Dental Research, 31(2), 203 (2020); https://doi.org/10.4103/ijdr.IJDR_510_18.
- [8] Y. Yi, S.J. Heo, J.Y. Koak, S.K. Kim, *Mechanical complications of implant-supported restorations with internal conical connection implants: A 14-year retrospective study*. The Journal of prosthetic dentistry, 129(5), 732 (2023); <https://doi.org/10.1016/j.prosdent.2021.06.053>.
- [9] L.M. Campaner, M.P.M. Silveira, G.S. de Andrade, A.L.S. Borges, M.A. Bottino, A.M.O. Dal Piva, R. Lo Giudice, P. Ausiello, J.P.M. Tribst, *Influence of Polymeric Restorative Materials on the Stress Distribution in Posterior Fixed Partial Dentures: 3D Finite Element Analysis*. Polymers, 13(5), 758, (2021); <https://doi.org/10.3390/polym13050758>.
- [10] N.S. Sahai, D. Yadav, S. Madhav, N. Kumar, A. Dubey, N. Qadeer, R.S. Makkad, *Comparative Study of the Influence of Dental Implant Design on the Stress and Strain Distribution using the Finite Element Method*. Journal of pharmacy & bioallied sciences, 16(3), S2776 (2024); https://doi.org/10.4103/jpbs.jpbs_432_24.

Б.І. Гриньків¹, Л.О. Борушчак², З.Р. Ожоган¹, Л.В. Мізюк¹, Р.З. Ожоган¹,
І.В. Виклюк¹

Комп'ютерне моделювання та дослідження напружено-деформованого стану системи «дентальний імплантат – кісткова тканина»

¹Івано-Франківський національний медичний університет, м. Івано-Франківськ, Україна, b.grynkiw@gmail.com

²Івано-Франківський національний технічний університет нафти і газу, м. Івано-Франківськ, Україна, liubomyr.borushchak@nung.edu.ua

Визначено характер напружено-деформованого стану системи “дентальний імплантат – кісткова тканина” за допомогою тривимірних комп'ютерних моделей, які були створені у програмі твердотілого параметричного моделювання Solid Works 2024 із застосуванням методу скінченних елементів. Власне дослідження були виконані засобами пакету Simulation. За допомогою програми Solid Works 2024 розроблено твердотільні параметричні моделі, які включали фрагмент щелепної кістки з дентальним імплантатом та ортопедичними елементами. Для проведення дослідження застосовано фізико-механічні параметри ортопедичних елементів (коронки та абатмента), дентального імплантата, кортикальної та трабекулярної кісткової тканини щелеп. Дослідження напружено-деформованого стану проведено за умови середніх показників міцності та щільності трабекулярної кісткової тканини щелеп. Під час виконання дослідження було створено умови максимально асиметричного прикладання сили жувального навантаження на ортопедичні елементи та дентальний імплантат, що дозволило виявити несприятливі умови функціонування системи коронка – імплантат – кісткова тканина.

Ключові слова: напруження; деформація; остеоінтеграція; тривимірна комп'ютерна модель; опорні зуби; дентальний імплантат; щільність кісткової тканини; трабекулярна кісткова тканина; метод скінченних елементів (МСЕ).