DOI:10.33667/2782-4101-2023-1-19-23

THE USAGE OF THE FINITE ELEMENT ANALYSIS IN THE DESIGN OF NEW DENTAL IMPLANT SYSTEMS

Yu. A. Sergeev[1], A. A. Dolgalev[1], D. Z. Choniashvili[2], V. M. Avanisyan[1]

[1]Stavropol State Medical University of the Ministry of Health of the Russian Federation, Stavropol, Russian Federation
[2]Federal State Budgetary Educational Institution of Higher Professional Education «Kosta Khetagurov North Ossetian State University» (NOSU), Vladikavkaz, Russian Federation

SUMMARY
Introduction. *The development of new dental implants in the context of the booming domestic industry makes it possible to find alternative options in the treatment of clinically difficult situations, to select the necessary individual solution during dental implant surgery and consequently to perform the surgery in an error-free manner and achieve the desired results. The development of a dental implant is a multistep process, and the characteristics of the implant material and its biophysical characteristics must be studied in detail until the implant is integrated into the bone tissue.*
The aim of the study: *to estimate the opportunities and prospects of applying the finite elements method by developing the new systems of dental implants according to the literature data.*
Materials and methods. *A search was carried out in the national digital libraries e-library, CyberLeninka, as well as PubMed, Medline, Web of Science and Google Scholar using the following keywords: dental implant, finite-element analysis, mathematical model. Sixty-nine papers were selected and analysed.*
Results. *The finite element method is an accurate method to analyse the implant being developed, but it has certain limitations, because in the finite element mesh, the implant-bone interface is a continuous relationship. The absence of micro-movement at the implant-bone interface during loading is different from the actual clinical situation. The expected 100% osseointegration based on 3D-modelling can't be an ideal option and never corresponds to the reality in the clinical situation. However, the use of the finite element method makes it possible to test single loads and inclination angles, which in the clinical situation is very rare.*

KEYWORDS: *biocompatibility, osseointegration, biomechanics, dental implant, finite element analysis, mathematical model.*

CONFLICT OF INTEREST. *The authors declare no conflict of interest.*

Introduction

Today, issues related to achieving positive results in dental implant surgery have received considerable attention. This is due to the expansion of a number of techniques, surgical techniques, a wide range of dental surgeons, especially in view of the widespread development and implementation of digital dentistry, and new types of implants [1, 2].

At the time of dental implant development, the study of biophysical characteristics is traditionally considered of paramount importance. This is due to a number of factors, which undoubtedly include the response of bone tissue to the foreign body, the loading ratio of the dental implant and bone, as well as the distribution of forces at the implant-abutment junction. In the case of dental implants, the following features should be noted:

- nature of the failure: fatigue or temporary (static);
- the condition of the bone, both at the implant site itself and in the surrounding bone tissue;
- the condition of the implant fixation nodes;
- the degree of osseointegration of the implant at different sites.

In the body of knowledge required by the dental implantologist using dental implants, the fundamentals of biomechanical reasoning for the decisions he or she makes are of particular importance [3].

Inadequate biomechanical analysis in the design, fabrication and application of dental implants can lead to a lack of formation of the bone-implant interface as well as short duration of their function, which in turn affects the patient and leads to negative feedback.

The aim of the work was to examine the publications available in digital libraries, which included information on the possibilities and peculiarities of applying the finite element method to the development of new dental implant systems**.**

Materials and methods of research

Scientific sources indexed in PubMed, Medline, Web of Science and Google Scholar were analysed in this study. The sources describing the original research in this area focused on the prototypes of the implants being developed, stress distribution on the adjacent bone, the biomechanics of the dental implant and bone, and the implant-bone interface.

Results and conclusions

Dental implants made of titanium (Ti) and its alloys (roxolide, titanium-zirconium alloy) are one of the most reliable alternatives for replacing missing teeth due to their optimal

a б

Fig. 1. Dental implants (a) and 3D model of the mandible with dental implants with abutments (b)

biomechanical properties [66, 68]. Almost all existing implants are made of titanium, which has a hard structure and a higher strength that is 5–10 times stronger than bone [4]. It is worth noting that a significant mismatch between implant strength and bone strength can contribute to overloading, aseptic inflammation and bone resorption [5–10].

Currently, the development of new dental implant systems is based on the principles of optimising implant physical density and bone simulation [11–16] to overcome the aforementioned complications and achieve complete bone remodelling.

Thus, this process can be influenced by many factors such as implant material and design [17–21], implant surface [22–25] and bone quality [9, 26–29].

Implants have a higher physical density and are able to absorb more load and transfer less stress to the surrounding bone tissue. The amount of deformation in the surrounding bone determines the remodelling process under occlusal loading [30, 31], as proven by Wolf's transformation law.

Harold Frost [32] described the mechanostatic theory as changes in strain levels and corresponding changes in bone density. Bone density is a value that is often used to assess clinical evidence of bone health. Each individual may have a different bone density. In addition, bone density varies in different age and gender groups [33].

To overcome the aforementioned complications, many researchers are working on the development of new materials and bioinspired structures using both standard solutions and additive technologies (3D-printing) to recreate mechanical properties that are well compatible with bone tissue [23, 34, 35, 36]. To simulate different bone densities, implant designs and various mechanical loads, computer methods are actively used in the evaluation of implant biomechanics [37, 38]. The most relevant method for modelling and calculating the strength and reliability of developed products is the usage of modern computer-aided engineering analysis packages, with ANSYS [39] being the most effective computer-aided engineering (CAE) system for modelling the functional processes of such products.

The main method of computer simulation is the finite element analysis (FEA), which allows the calculation of the stress-strain state (SSS) arising within a mechanical system under the influence of external forces, as well as displaying the areas of the structure where material deformation and subsequent failure occurs [40]. The calculated finite element method of stress analysis may not exceed the stress limit values.

It is important to consider that the transfer of stress between the implant and the bone depends on a number of factors and the description of this process is quite extensive. In order to realise the biomechanical effects, a 3D mathematical model is used, in which the geometric data of the implant, the mechanical properties of the bone and the parameters of the bone-implant interface are defined, otherwise known as a finite element network [41–44]. By modifying the individual elements of this system, it is possible to obtain data on the clinical performance of the dental implant.

Thus, by studying the biomechanical features of dental implants, biomechanical changes and VAT can be fully determined, which in turn will help to further improve implantation techniques and increase the effectiveness of the treatment performed [45].

At the time of creation of a dental implant, methods for biomechanical evaluation of its effect on the bone-implant interface are actively used to predict the behaviourof the implant directly in the bone tissue and to assess its advantages and disadvantages [46]. To study the biomechanical effects, a computer model (*Figure 1*) with predetermined dental implant specifications is used to digitally construct a finite element model of the jaw region with the dental implant (bone tissue parameters are determined in advance based on already available data).

Korioth, T.W. and Hannam, A.G. [44] reflected one of the first works on the application of the finite element method to biomechanical analysis in dentistry. More recently, Van Staden, R.C. et al. [47] in their work indicated that finite element method (FEM) should be considered a numerical method for the analysis of strains and stresses in any given structure. Today, FEM is a widely used method in the field of dental biomechanics [48].

A study based on the FEM using simplified models, allows «pure experimentation», i.e. to exclude all irrelevant factors inherent to the real object, with the properties of FEM models being as close as possible to the real object [49].

During the development phase of a dental implant, a finite element model is used to assess the technical characteristics and will overwhelmingly consist of several parts (*Figure 2*), namely the jaw cortical bone, jaw trabecular bone, dental implant, abutment and abutment screw, and the crown and its fixation method [9,46,50].

The properties of the dental implant materials to be used in the biomechanics simulation must be specified in advance. The materials must be homogeneous, isotropic and linearly elastic.

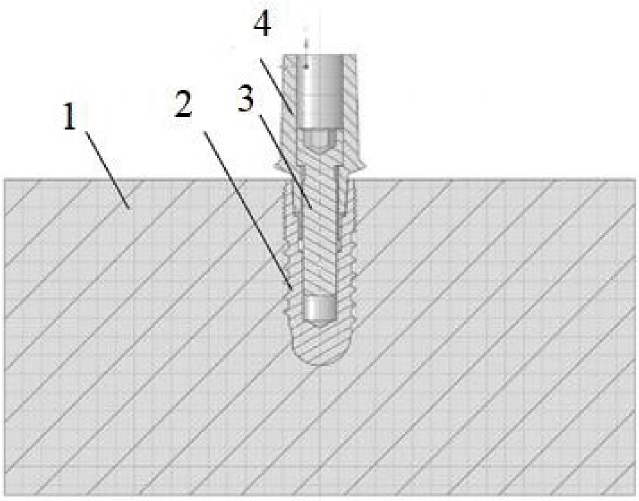

Fig. 2. Finite element model for dental implant design: 1 – bone block, 2 – implant body, 3 – screw, 4 – abutment

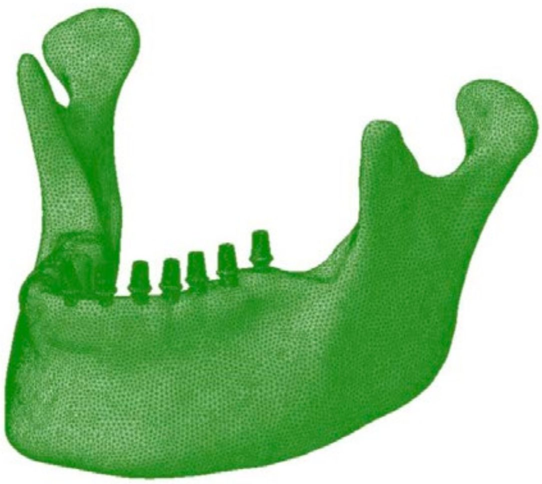

Fig. 3. Grid element in the computer model

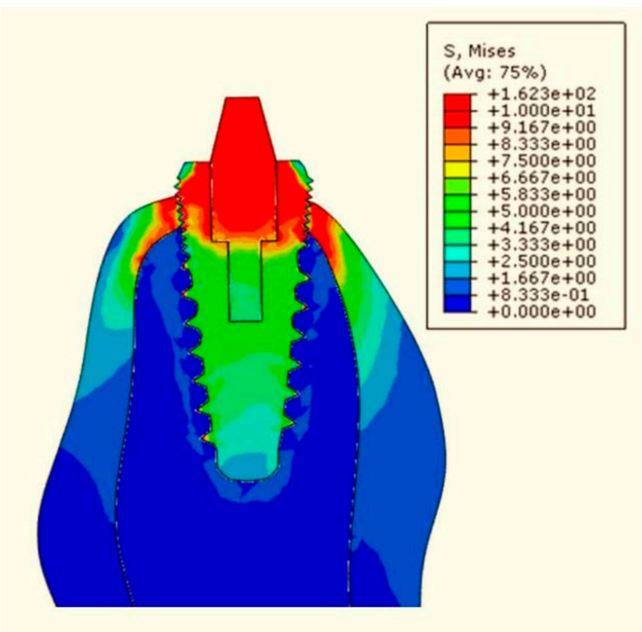

Fig. 4. Von Mises load distribution

Therefore, 2 independent parameters (Young's modulus (E) and Poisson's ratio (ν)) are often used to indicate material properties.

The mesh element used in the finite element computer model is a tetrahedral mesh [51,52] (*Figure 3*).

In most cases, it is necessary to use a finite element mesh model to determine the various occlusal conditions on the dental implant system to be designed.

FEM makes it possible to assess the distribution of reaction forces and structural stresses at the fixed upper end of the abutment in a dental implant system [20,53].

With the mathematical model, it is possible to estimate the von Mises stress distribution in the modelled bone area (*Figure 4*). Moreover, when examining the load on the alveolar bone in the areas of a single tooth, it can be seen that the areas in connection with the cortical bone have a higher stress. The occurrence of this high stress can be explained by Hooke's law (stress = Young's modulus × strain). When a dental implant system receives an external force, it will have a slight downward displacement, compressing the alveolar bone and creating the same displacement at the junction of the cortical bone and the trabecular bone (creating the same deformation here) [12]. The stress is proportional to the Young's modulus. The cortical bone above the alveolar bone therefore has a higher stress (54). Therefore, the design of some implants increases the number of microthread coils on the implant neck, increasing its contact area with the alveolar bone surface, preventing the dental implant system from loosening when exposed to external forces, thereby increasing implant success.

The von Mises load distribution of the bone-implant system shows that high loads on dental implants due to external forces mainly occur near the dental implant neck, where the dental implant contacts the abutment. Therefore, when a tooth receives an external force, it directly deforms the neck of the dental implant. According to Hooke's law, high stress will be generated in this area. [10].

It is worth noting that the alveolar bone adjacent to the implants that have been exposed to external forces will also be highly stressed due to deformation (44).

When assessing the stress on the abutment and abutment screw, it can be seen that high stress on the abutment occurs at its junction with the dental implant [55, 56]. High stress on the abutment screw occurs in the area where the screw head is connected to the abutment, also corresponding to the area where the geometric shape of the screw head and screw are bent. Therefore, the design of abutments and abutment screws should avoid the generation of high stresses by the geometric shape. Otherwise, since the patient will be using the structure for an extended period, the dental implant system may be damaged due to material fatigue (57, 58).

In obtaining unique dental implant models, and specifically in the design phase of their development today, a self-adaptive 3D model is used, which in contrast to traditional approaches in parameterised self-modifying implant models, assembling self-adaptive 3D models, transferring bi-directional parameters and adjusting variables [59].

Building a parametrized self-modifying implant model means that the implant model is built based on the diameter and length of the implant. In other words, the amount of implant threading can be changed with automatic changes in implant diameter and length. Assembling self-adaptive 3D models means that all models are rebuilt based on implant parameters. That is, the parameters of other parts (bone types) were changed while automatically changing the implant parameters [4, 58].

CAD (Pro/E) and CAE (ANSYS Workbench) bidirectional parameter transfer tools can transfer model parameters mutually and seamlessly. The variable settings include input variables (D and L) and output variables (max EQV stress in the mandible and max displacement in the implant-abutment complex) [38, 39].

Results

FEM is an accurate method to analyse the implant under development, but has certain limitations because in a finite element mesh, the implant-bone interface is a continuous relationship. The absence of micro-movement at the implant-bone interface during loading is actually different from the actual clinical situation [28, 60].

The expected 100% osseointegration based on 3D modelling cannot be ideal and never corresponds to reality in the clinical situation. The bone (cortical, cancellous) and the implant are thought to be isotropic and homogeneous, but in real, the bone is anisotropic and heterogeneous. The implant is rigidly fixed in the bone. Loads were only applied at point locations. The duration of force application in implants and the oral cavity is different [8, 61, 62].

In addition, the use of FEM, allows the testing of single loads and tilt angles, which is very rare in the clinical situation [25, 45, 57, 63, 64, 65].

In many scientific studies using the finite element method, most authors use optimum values and loads [66], but for a complete understanding of the biomechanical behaviourof dental implants, attention must be paid to all existing biomechanical modelling features.

Conclusions

The finite element method is an important tool in dental implantology, because it makes it possible to test prototypes of implants under development and to study the behaviour of existing modified implants in order to study stress distribution in adjacent bone, the biomechanics of the dental implant and bone, and the implant-bone interface.

A mathematical model including finite element analysis allows for predicting possible risks associated with overloading of the implant or possible complications at the time of loading.

The combined use of fatigue, aging, thermal and continuous mechanical cyclic loading in the analysis of dental implant prototypes makes it possible to generate the most effective medical devices from a clinical point of view.

Reference

1. *Ananth, H.; Kundapur, V.; Mohammed, H.; Anand, M.; Amarnath, G.; Mankar, S. A review on biomaterials in dental implantology. Int. J. Biomed. Sci. 2015, 11, 113.*
2. *Kawahara et al.//Image synthesis with deep convolutional generative adversarial networks for material decomposition in dual-energy CT from a kilovoltage CT/Comput. Biol. Med.(2021)*
3. *Marcián P. et al.//Micro finite element analysis of dental implants under different loading conditions/Comput. Biol. Med.(2018)*
4. *Chang, P.-K.; Chen, Y.-C.; Huang, C.-C.; Lu, W.-H.; Chen, Y.-C.; Tsai, H.-H. Distribution of micromotion in implants and alveolar bone with different thread profiles in immediate loading: A finite element study. Int. J. Oral Maxillofac. Implant. 2012, 27, e96-e101.*
5. *Dos Santos, M.C.L.G.; Campos, M.I.G.; Line, S.R.P. Early dental implant failure: A review of the literature. Braz. J. Oral Sci. 2002, 1, 103–111.*
6. *Han et al.//Continuous functionally graded porous titanium scaffolds manufactured by selective laser melting for bone implants/J. Mech. Behav. Biomed. Mater.(2018)*
7. *Himmlova, L.; Dostalova, T.; Kacovsky, A.; Konvickova, S. Influence of implant length and diameter on stress distribution: A finite element analysis. J. Prosthet. Dent. 2004, 91, 20–25*
8. *Jafari et al.//A comparative study of bone remodeling around hydroxyapatite-coated and novel radial functionally graded dental implants using finite element simulation/Med. Eng. Phys.(2022)*
9. *Kang, X.; Li, Y.; Wang, Y.; Zhang, Y.; Yu, D.; Peng, Y. Relationships of Stresses on Alveolar Bone and Abutment of Dental Implant from Various Bite Forces by Three-Dimensional Finite Element Analysis. Biomed Res. Int. 2020, 2020, 7539628.*
10. *Schwitalla, A.; Abou-Emara, M.; Spintig, T.; Lackmann, J.; Müller, W. Finite element analysis of the biomechanical effects of PEEK dental implants on the peri-implant bone. J. Biomech. 2015, 48, 1–7.*
11. *Bozkaya, D.; Muftu, S.; Muftu, A. Evaluation of load transfer characteristics of five different implants in compact bone at different load levels by finite element analysis. J. Prosthet. Dent. 2004, 92, 523–530.*
12. *Cozzolino et al.//Implant-to-bone force transmission: a pilot study for in vivo strain gauge measurement technique/J. Mech. Behav. Biomed. Mater.(2019).*
13. *Dutta et al.//Design of porous titanium scaffold for complete mandibular reconstruction: the influence of pore architecture parameters/Comput. Biol. Med.(2019)*
14. *Li et al.//3D porous Ti6Al4V-beta-tricalcium phosphate scaffolds directly fabricated by additive manufacturing/Acta Biomater.*
15. *Sato et al.//The effects of bone remodeling on biomechanical behaviour in a patient with an implant-supported overdenture. Biol. Med. (2021)*
16. *Wang Juncheng & Yang Sefei. (2017). Risk factors affecting osseointegration of implants: (eds.) Proceedings of the 8th Academic Conference of the General Stomatology Committee of the Chinese Stomatological Association (pp.338)*
17. *Cynthia S. Petrie DDS, MS and John L. Williams Ph D. Shape optimization of dental implant designs under oblique loading using the p-version finite element method[J]. Journal of Prosthodontics, 2002, 11(4): 333–334.*
18. *Ghaziani A.O. et al.//The effect of functionally graded materials on bone remodeling around osseointegrated trans-femoral prostheses/J. Mech. Behav. Biomed. Mater.(2021)*
19. *Günther F. et al.//Design procedure for triply periodic minimal surface based biomimetic scaffolds/J. Mech. Behav. Biomed. Mater.(2022).*
20. *Verri, F.R.; de Souza Batista, V.E.; Santiago, J.F., Jr.; de Faria Almeida, D.A.; Pellizzer, E.P. Effect of crown-to-implant ratio on peri-implant stress: A finite element analysis. Mater. Sci. Eng. C 2014, 45, 234–240.*
21. *Wu, S.-W.; Lee, C.-C.; Fu, P.-Y.; Lin, S.-C. The effects of flute shape and thread profile on the insertion torque and primary stability of dental implants. Med. Eng. Phys. 2012, 34, 797–805.*
22. *Li et al.//Early osteointegration evaluation of porous Ti6Al4V scaffolds designed based on triply periodic minimal surface models/J. Orthop. Transl.(2019)*
23. *Mehboob et al.//Finite element modelling and characterisation of 3D cellular microstructures for the design of a cementless biomimetic porous hip stem/Mater. Des.(2018).*
24. *Salou et al.//Enhanced osseointegration of titanium implants with nanostructured surfaces: an experimental study in rabbits/Acta Biomater.*
25. *Shim, H.W.; Yang, B.-E. Long-term cumulative survival and mechanical complications of singletooth Ankylos Implants: Focus on the abutment neck fractures. J. Adv. Prosthodont. 2015, 7, 423430.*
26. *Azcarate-Velázquez et al./Influence of bone quality on the mechanical interaction between implant and bone: a finite element analysis/J. Dent.(2019)*
27. *Kasani R. et al.//Stress distribution of overdenture using odd number implants – a Finite Element Study/J. Mech. Behav. Biomed. Mater.(2019).*
28. *Peyroteo M.M.A. et al.//A mathematical biomechanical model for bone remodeling integrated with a radial point interpolating meshless method/Comput. Biol. Med.(2021)*
29. *Piccinini et al.//Numerical prediction of peri-implant bone adaptation: comparison of mechanical stimuli and sensitivity to modeling parameters/Med. Eng. Phys.(2016)*
30. *Negmatova D.U., Kamariddinzoda M.K. «Modern Approaches to Solving Biomechanical Problems in Dental Implantology» Voprosy Materialovedeniya Nauk, no. 7 (53), 2019, pp. 227234.*
31. *Cheong V.S. et al.//Bone remodelling in the mouse tibia is spatio-temporally modulated by oestrogen deficiency and external mechanical loading: a combined in vivo/in silico study/Acta Biomater.(2020)*
32. *Tyrovola J.B. (2015). The «Mechanostat Theory» of Frost and the OPG/RANKL/RANK System. Journal of cellular biochemistry, 116(12), 2724–2729. https://doi.org/10.1002/jcb.25265*

33. Gubaua J. E. et al.//Techniques for mitigating the checkerboard formation: application in bone remodeling simulations/Med. Eng. Phys.(2022)
34. Santiago Junior, J.F.; Pellizzer, E.P.; Verri, F.R.; de Carvalho, P. S. Stress analysis in bone tissue around single implants with different diameters and veneering materials: A 3-D finite element study. Mater. Sci. Eng. C Mater. Biol. Appl. 2013, 33, 4700–4714.
35. Yao Y. et al.//A personalized 3D-printed plate for tibiotalocalcaneal arthrodesis: design, fabrication, biomechanical evaluation and postoperative assessment/Comput. Biol. Med.(2021)
36. Zheng et al.//Bone remodeling following mandibular reconstruction using fibula free flap/J. Biomech.(2022)
37. Alzahrani F. S. et al.//Analytical estimations of temperature in a living tissue generated by laser irradiation using experimental data/J. Therm. Therm. Biol.(2019)
38. Chakraborty et al.//Finite element and experimental analysis to select patient's bone condition specific porous dental implant fabricated using additive manufacturing/Comput. Biol. Med.(2020)
39. Zanichelli, A., Colpo, A., Friedrich, L., Iturrioz, I., Carpinteri, A., & Vantadori, S. (2021). A Novel Implementation of the LDEM in the Ansys LS-DYNA Finite Element Code. Materials (Basel, Switzerland), 14(24), 7792. https://doi.org/10.3390/ma14247792
40. Bulaqi, H.A.; Mashhadi, M.M.; Safari, H.; Samandari, M.M.; Geramipanah, F. Effect of increased crown height on stress distribution in short dental implant components and their surrounding bone: A finite element analysis. J. Prosthet. Dent. 2015, 113, 548–557.
41. Kong, L.; Liu, B.; Li, D.; Song, Y.; Zhang, A.; Dang, F.; Qin, X.; Yang, J. Comparative study of 12 thread shapes of dental implant designs: A three-dimensional finite element analysis. World J. Model. Simul. 2006, 2, 134–140.
42. Kong, L.; Zhao, Y.; Hu, K.; Li, D.; Zhou, H.; Wu, Z.; Liu, B. Selection of the implant thread pitch for optimal biomechanical properties: A three-dimensional finite element analysis. Adv. Eng. Softw. 2009, 40, 474–478.
43. Koolstra, J.H.; van Eijden, T. M. Combined finite-element and rigid-body analysis of human jaw joint dynamics. J. Biomech. 2005, 38, 2431–2439.
44. Korioth, T.W.; Hannam, A. G. Mandibular forces during simulated tooth clenching. J. Orofac. Pain 1994, 8, 179–189.
45. Su, K.-C.; Chang, C.-H.; Chuang, S.-F.; Ng, E.Y.-K. Biomechanical evaluation of endodontic post-restored teeth-finite element analysis. J. Mech. Med. Biol. 2013, 13, 1350012.
46. El-Anwar, M.I.; El-Zawahry, M.M. A three dimensional finite element study on dental implant design. J. Genet. Eng. Biotechnol. 2011, 9, 77–82.
47. Van Staden, R.C.; Guan, H.; Loo, Y. C. Application of the finite element method in dental implant research. Comput. Methods Biomech. Biomed. Eng. 2006, 9, 257–270.
48. Chieruzzi, M.; Pagano, S.; Cianetti, S.; Lombardo, G.; Kenny, J.M.; Torre, L. Effect of fibre posts, bone losses and fibre content on the biomechanical behaviour of endodontically treated teeth: 3Dfinite element analysis. Mater. Sci. Eng. C Mater. Biol. Appl. 2017, 74, 334–346.
49. Hijazi, L.; Hejazi, W.; Darwich, M.A.; Darwich, K. Finite element analysis of stress distribution on the mandible and condylar fracture osteosynthesis during various clenching tasks. J. Oral. Maxillofac. Surg. 2016, 20, 359–367.
50. Baggi, L.; Cappelloni, I.; Di Girolamo, M.; Maceri, F.; Vairo, G. 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, 422–431.
51. Luo, D.; Rong, Q.; Chen, Q. Finite-element design and optimization of a three-dimensional tetrahedral porous titanium scaffold for the reconstruction of mandibular defects. Med. Eng. Phys. 2017, 47, 176–183.
52. Su, K.-C.; Chang, C.-H.; Chuang, S.-F.; Ng, E.Y.-K. Biomechanical evaluation of endodontic post-restored teeth-finite element analysis. J. Mech. Med. Biol. 2013, 13, 1350012.
53. Jo, J.-Y.; Yang, D.-S.; Huh, J.-B.; Heo, J.-C.; Yun, M.-J.; Jeong, C.-M. Influence of abutment materials on the implant-abutment joint stability in internal conical connection type implant systems. J. Adv. Prosthodont. 2014, 6, 491–497.
54. Amid, R.; Ebrahimi, N.; Kadkhodazadeh, M.; Mirakhori, M.; Mehrinejad, P.; Nematzadeh, F.; Dehnavi, F. Clinical evaluation of a new device to measure maximum bite force. Dentist. Case. Rep. 2018, 2, 26–29.
55. Ashley, E.T.; Covington, L.L.; Bishop, B.G.; Breault, L. G. Ailing and failing endosseous dental implants: A literature review. J. Contemp. Dent. Pract. 2003, 4, 35–50.
56. Li, T.; Hu, K.; Cheng, L.; Ding, Y.; Ding, Y.; Shao, J.; Kong, L. Optimum selection of the dental implant diameter and length in the posterior mandible with poor bone quality-A 3D finite element analysis. Appl. Math. Model. 2011, 35, 446–456.
57. A. A. Muraev, S. Y. Ivanov, S. Leonov. Y., Leonov S. B. [et al.] // Comparative analysis of biomechanics at different implant-abutment interface nodes based on three-dimensional finiteelement modeling data / Dentistry.– 2019.– T. 98.– № 1.– C. 11–16.– DOI 10.17116/stomat20199801111.
58. Chang, Y.-H.; Chan, M.-Y.; Hsu, J.-T.; Hsiao, H.-Y.; Su, K.-C. Biomechanical Analysis of the Forces Exerted during Different Occlusion Conditions following Bilateral Sagittal Split Osteotomy Treatment for Mandibular Deficiency. Appl. Bionics Biomech. 2019, 2019, 4989013.
59. Ryu, H.-S.; Namgung, C.; Lee, J.-H.; Lim, Y.-J. The influence of thread geometry on implant osseointegration under immediate loading: A literature review. J. Adv. Prosthodont. 2014, 6, 547554.
60. Dolgalev A. A., Muraev A. A., Lyakhov P. A., Lyakhova U. A., Choniashvili D. Z., Zolotayev K. E., Semerikov D. Yu., Avanisyan V. M. Determining the optimal neural network structure for the development of decision support programmes in dental implantation. Medical alphabet. 2022;(34):54–64.
61. Huang, H.-L.; Hsu, J.-T.; Fuh, L.-J.; Tu, M.-G.; Ko, C.-C.; Shen, Y.-W. Bone stress and interfacial sliding analysis of implant designs on an immediately loaded maxillary implant: A nonlinear finite element study. J. Dent. 2008, 36, 409–417.
62. Marcián, P.; Borák, L.; Valášek, J.; Kaiser, J.; Florian, Z.; Wolff, J. Finite element analysis of dental implant loading on atrophic and non-atrophic cancellous and cortical mandibular bone-a feasibility study. J. Biomech. 2014, 47, 3830–3836.
63. Savransky F. Z., Grishin P. O., Kushnir E. N. [et al] // The use of the method of mathematical modeling of the stress-strain state of bone tissue in dental implantation (literature review) / Modern orthopaedic dentistry.– 2018.– № 30.– C. 30–33.
64. Huang, H.-L.; Su, K.-C.; Fuh, L.-J.; Chen, M.Y.; Wu, J.; Tsai, M.-T.; Hsu, J.-T. Biomechanical analysis of a temporomandibular joint condylar prosthesis during various clenching tasks. J. Cranio-MaxilloFac. Surg. 2015, 43, 1194–1201.
65. Van Eijden, T. Three-dimensional analyses of human bite-force magnitude and momentum. Arch. Oral Biol. 1991, 36, 535–539.
66. Merdji et al.//Stress distribution in dental prosthesis under an occlusal combined dynamic loading/Mater. Des.(2012)

Received 28.04.2023
Revised 05.05.2023
Accepted 10.05.2023

INFORMATION ABOUT AUTHORS

Sergeev Yuriy Andreevich, *Postgraduate Student, Department of General Practice and Pediatric Dentistry, Stavropol State Medical University of the Ministry of Health of the Russian Federation, 355000, Stavropol, Russian Federation. ORCID: https://orcid.org/0000-0002-6183-2586. Tel. +7-906440-18-89, email: serg_yuriy@mail.ru.*

Dolgalev Alexander Alexandrovich, *MD, Head of the Center for Innovation and Technology Transfer, Professor of the Department of General Practice Dentistry and Pediatric Dentistry of the Stavropol State Medical University of the Ministry of Health of the Russian Federation, Professor of the Department of Clinical Dentistry with a course of OS and MFS of the Pyatigorsk Medical and Pharmaceutical as Institute-branch of the Volgograd State Medical University Stavropol, Russian Federation. ORCID: https://orcid.org/0000-0002-6352-6750. Tel. +7-962-440-48-61, email: dolgalev@dolgalev.pro*

Choniashvili David Zurabovich, *Candidate of Medical Sciences, Associate Professor of the Department of Therapeutic, Surgical and Pediatric Dentistry with courses in Implantology, Reconstructive Oral Surgery, Pediatric maxillofacial surgery, Dean of the Medical Faculty of the Federal State Educational Institution of Higher Professional Education «North Ossetian State University named after Kosta Levanovich Khetagurov (NOSU), Head Doctor of the Clinical and Diagnostic Center of Dentistry of NOSU, Vladikavkaz, Russian Federation. Tel. +7-918-828-11-72, email: davidchoniashvili@mail.ru*

Avanisyan Vazgen Mikhailovich, *1-st year resident at the Department of Therapeutic Dentistry, Stavropol State Medical University of the Ministry of Health of the Russian Federation, 355000, Stavropol, Russian Federation. ORCID: https://orcid.org/0000-0002-0316-5957, SPIN code: 1207-9234. Tel. +7-928-633-89-95, email: avanvaz@yandex.ru*

Contact person: *Vazgen Mikhailovich Avanisyan, 1st year resident, department of Therapeutic Dentistry, 310 Mira Street, Stavropol, Russian Federation. For correspondence: avanvaz@yandex.ru, +7-928-633-89-95.*