DE G DE GRUYTER OPEN

10.1515/bjdm-2017-0011

BALKAN JOURNAL OF DENTAL MEDICINE ISSN 2335-0245

Practical Aspects of Finite Element Method Applications in Dentistry

SUMMARY

The use of numerical methods, such as finite element method (FEM), has been widely adopted in solving structural problems with complex geometry under external loads when analytical solutions are unachievable. Basic idea behind FEM is to divide the complex body geometry into smaller and simpler domains, called finite elements, and then to formulate solution for each element instead of seeking a solution for the entire domain. After finding the solutions for all elements they can be combined to obtain a solution for the whole domain. This numerical method is mostly used in engineering, but it is also useful for studying the biomechanical properties of materials used in medicine and the influence of mechanical forces on the biological systems. Since its introduction in dentistry four decades ago, FEM became powerful tool for the predictions of stress and strain distribution on teeth, dentures, implants and surrounding bone. FEM can indicate aspects of biomaterials and human tissues that can hardly be measured in vivo and can predict the stress distribution in the contact areas which are not accessible, such as areas between the implant and cortical bone, denture and gingiva, or around the apex of the implant in trabecular bone. Aim of this paper is to present – using results of several successful FEM studies – the usefulness of this method in solving dentistry problems, as well as discussing practical aspects of FEM applications in dentistry. Some of the method limitations, such as impossibility of complete replication of clinical conditions and need for simplified assumptions regarding loads and materials modeling, are also presented. However, the emphasis is on FE modelling of teeth, bone, dentures and implants and their modifications according to the requirements. All presented studies have been carried out in commercial software for FE analysis ANSYS Workbench.

Key words: Finite Element Method, Biomechanical Systems, Computer Simulations, Stress Analysis

Aleksandar Grbović, Dimitrije Mihajlović

Faculty of Mechanical Engineering, University of Belgrade, Serbia

REVIEW PAPER (RP)
Balk J Dent Med, 2017;69-77

Introduction

Finite element method (FEM) is one of the most widely used numerical methods for solving the problems of mechanics of continuum. FEM is method of discrete analysis and – unlike other numerical methods which are based on mathematical discretization[a] of equations of boundary problems – it is based on physical discretization of considered domain. Basis for all calculations is represented by the part of the domain (so called sub-domain) which has finite dimensions, also known as finite element. From the perspective of physical interpretation this means that the observed real physical domain with infinite number of degrees of freedom[b] (DOF) can

a Discretization is the process of transferring continuous functions, models, and equations into discrete counterparts. This process is usually carried out as a first step toward making them suitable for numerical evaluation and implementation on digital computers.

b In engineering, the degree of freedom (DOF) of a mechanical system is the number of independent parameters that define its configuration. It's the number of parameters that determine the state of a physical system. For example, the position and orientation of a rigid body in space is defined by three components of translation and three components of rotation, which means that it has six degrees of freedom.

be replaced with discretized geometrical model with finite number of DOF. Such model consists of elements interconnected by finite number of points known as nodes (Figure 1). These finite elements have defined dimensions, physical properties and simple geometry, and together they can "simulate" the behavior of complex physical system. As a part of the process of discretization, choice of shape of finite element and number of elements used in numerical simulation is influenced by the nature of the analyzed problem and the required accuracy of solution.

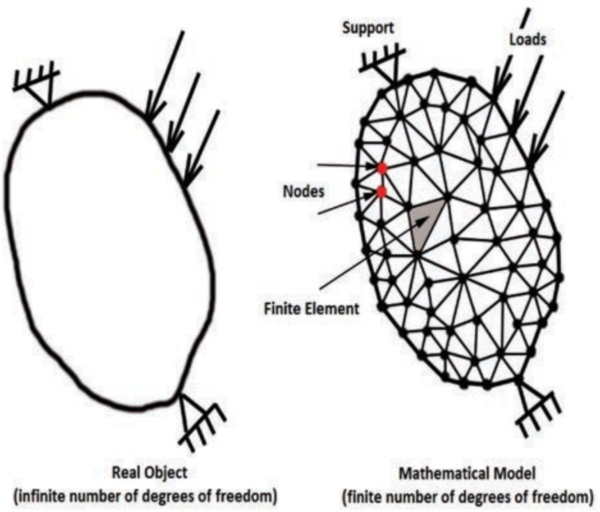

Figure 1. Approximation of the real object using FEM

By analyzing the individual elements along with the characteristics of their mutual connections, the whole complex system can be analyzed. The approach where universal solution is obtained from the individual solutions is known as inductive method. This method is of the greatest importance for solving the problems where exact solutions cannot be achieved directly[1].

Figure 1 shows the sketch of the real object and equivalent mathematical model after one possible discretization. Set of subdomains for the entire domain is called finite element mesh. Each node in mesh has finite number of degrees of freedom. Forces cannot act on the surface of the finite element or edge but only in nodes. After carried out calculations, each node will be assigned displacement values that represent the reaction of the entire system to given loads and boundary conditions[c]. Values of displacements on the finite elements between the nodes are determined by means of mathematical interpolation[d].

The power of the FME resides primarily in its adaptability: analyzed structure might have arbitrary shape, materials, loads and supports. Also, the mesh may consist of elements of diverse types, shapes and physical properties. This great adaptability is usually achieved within a single computer program and the definition of all necessary input variables is controlled by user.

Finite element analysis (FEA) can provide detailed quantitative data at any location within mathematical model; therefore, FEA has become a valuable analytical tool in dentistry. FEM can indicate aspects of biomaterials and human tissues that can hardly be measured in vivo and can predict the stress and strain[e] distribution in the contact areas which are not accessible, such as areas between the implant and cortical bone, denture and gingiva, or around the apex of the implant in trabecular bone. In general, research fields in which FEM is implemented can be classified as follows[1]:

- Investigation of improved shape and design of fillings, crowns, dental implants, removable dentures, dental bridges, etc.;
- Examination of mutual interaction of stomatognathic system supporting structures;
- Study of residual stresses which occur as consequence of mechanical and thermal extension in crowns and dental fillings;
- Research of physiological and biochemical effects of chewing forces, teeth reactions to occlusal forces, their interaction and stress concentration;
- Research and application in orthodontics;
- Research and application in implantology.

In practice, applications of FEM in dentistry implies creation of virtual computer model with properly defined geometry and material properties, precisely defined loads and boundary conditions. These four parameters essentially define numerical model and the accuracy of the results is directly linked to them.

Geometry of the model must be close to the actual structure and unreasonable simplifications will unavoidably result in significant inaccuracy: experience and good judgment are needed to define adequate geometry. To decrease time to numerical solutions, researchers are often performing a two-dimensional (2D) instead of three-dimensional (3D) analyses because a 2D model is as efficient and accurate as a 3D model if it's well defined (Figure 2).

c The set of conditions specified for the behavior of the solution to a set of differential equations at the boundary of its domain. Boundary conditions are important in determining the mathematical solutions to physical problems.

d In engineering, interpolation is a method of constructing new data points within the range of a discrete set of known data points. It is often required to interpolate (i.e. estimate) a value within two known values in a sequence of values. Polynomial interpolation is a widely-used method of estimating values between known data points.

e In continuum mechanics stress is a physical quantity that expresses the internal forces that neighboring particles of a continuous material exert on each other, while strain is the measure of the deformation of the material. The dimension of stress is that of pressure, and therefore is commonly measured in the same units as pressure (pascals), while strain is unitless quantity.

However, 2D models cannot simulate the 3D complexity of the structures and hence results might be of minor clinical values. Time needed to create FE models and obtain results is decreasing with advances in computer technology; thus, three-dimensional FE models are becoming dominant. With the development of digital imaging techniques more efficient methods are available for the direct transformation of 3D information from computed tomography (CT) or magnetic resonance imaging (MRI) into FE mesh. Solid models of a mandibula, crowns, teeth or dental implants may be obtained directly from 3D scanners or constructed using the computer-aided design (CAD) software such as CATIA or SolidWorks.

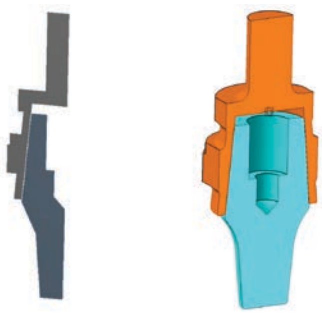

Figure 2. Two-dimensional model (left) vs. three-dimensional model (right)

Material properties considerably influence the stress and strain distribution in a structure. These properties can be modeled as isotropic, orthotropic, anisotropic, hyperelastic, viscoelastic, plastic (plasticity), etc. When the properties are the same in all directions material is linearly isotropic and only two independent material constants (Young's modulus E and Poisson's ratio v) must be defined. In contrast to isotropic, orthotropic materials have properties that differ along three mutually-orthogonal axes of rotational symmetry. This results in unique elastic behavior along the three orthogonal axes of the material, thus three elastic (E) and shear modulus (G) and six Poisson's ratios (v) must be known for model input. Orthotropic materials are a subset of anisotropic materials which are directionally dependent, which implies different properties in different directions. Anisotropy can be defined as a difference, when measured along different axes, in a material's physical or mechanical properties such as absorbance, refractive index, tensile strength, etc. Hyperelastic material model is used to describe the non-linear stress-strain behavior of complex materials such as rubbers, polymers, biological tissue, etc. Plasticity describes the deformation of a (solid) material undergoing non-reversible changes of shape in response to applied forces.

In most reported investigations in dentistry materials were modelled as homogenous and linearly isotropic for two main reasons: 1) It is not easy to accurately determine orthotropic, anisotropic or hyperelastic properties of material, and 2) If material is isotropic analysis is linear, otherwise it is non-linear and convergence[f] problem may arise. There are several methods to determine the physical properties of bone or tooth, such as tensile, compressive, bending and torsion testing, pure shear tests, micro- and nano- indentation tests, acoustic tests, etc. For example, the values from 13.7 to 20.7 GPa and 1.37 to 14.8 GPa have been frequently used for the Young's modulus of cortical and cancellous bone, respectively, and Poisson's ratio was assumed to be 0.3 for both[2]. But, bone is an anisotropic material with properties being directionally dependent. To incorporate realistic material for bone tissues in maxilla or mandible, the FEM may employ full orthotropy for cortical bone as the elastic behavior in cortical bone approximates to an orthotropic material and transversely isotropic for cancellous bone.

Selecting appropriate **loading conditions** is also of immense importance for productive FEA. In general, loads used in FE simulation can be divided into axial forces and horizontal forces (or moment-causing loads). Combinations of these forces (termed as mixed loading) define oblique occlusal loads which are more realistic and usually generate considerable localized stresses in compact bone. An axial force acts down the long axis of the tooth or implant and hence produces compression (which is favorable), whereas horizontal loading transmits tensile stresses and induces a bending (which is considered undesirable). For example, when a crown is to be fabricated for an implant, its shape must be without any cantilevers (which can contribute to torsional or bending movement) and should distribute biomechanical forces in a such way to produce favorable compressive stresses.

Extensive studies of masticatory (bite) force revealed significant variations in magnitude which were related to the area of the mouth, muscle size, bone shape, sex, age and many other factors. In the premolar region, values of masticatory force range from 40 to 600 N, forces from 50 to 400 N have been recorded in the molar region for young adults while forces from 25 to 170 N have been measured in the incisal region[3,4,5]. Clinical studies revealed that the average masticatory force transmitted to implant range between 90 N and 280 N, depending on the location, diameter, length of the implant and the kind of abutment used[6,7]. The choice of point of loading is also very important for successful FE analysis. Loading

f Convergence is a major issue with the use of FEM software. When FE problem is non-linear, solution techniques are based on iterative process to successively improve a solution, until 'convergence' is reached. The exact solution to the iterative problem is unknown, but numerical result must be sufficiently close to the solution for required level of accuracy. This requirement depends upon the purpose to which the solution will be applied.

point changes in accordance to the modeled morphology of the restoration. FEA studies have loaded the cusp tips, distal and mesial fossae of the crowns with the objective of simulating the contact path followed by the functional cusps of a tooth.

It's important to emphasize that all loads can be classified as either static or dynamic. Masticatory forces are dynamic loads, but since these loads are difficult to numerically model most FEA use static loads[8]. However, as an illustration that this modelling is possible, FE simulation with dynamic load will be presented in this paper.

The last but not less crucial step in defining FE model is determination of **boundary conditions**. The boundary conditions (BCs) are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field of interest. In other words, physical constraints such as displacements and supports must be applied on boundaries of the virtual model to ensure an equilibrium solution. The constraints are placed on nodes and they can prevent displacement and rotation in all directions (so called fixed support) or in some directions only (for example, displacement in X direction is allowed, while displacements in Y and Z direction and rotations about all three axes are not allowed). Boundary conditions can sometimes play the role of loads: instead of applying forces (whose magnitudes might be hard to evaluate) displacement on nodes in given direction is applied to simulate the effect of loading (Figure 3).

Figure 3. 2D FE model with symmetrical BCs and applied displacement

One method for efficient use of FE modeling is to exploit the planes of symmetry in an assembly or a part being analyzed. To take the advantage of symmetry, only a portion (a half, or a quarter) of the actual structure should be modeled to reduce the analysis run time and RAM memory required. The lines or planes of symmetry in a FE model can be simulated by providing proper BCs to the symmetrical faces or edges of the geometry, while loads must be completely symmetric, too. The general rule for a symmetry displacement condition is that the displacement vector component perpendicular to the symmetry plane is zero and the rotational vector components parallel to the plane are zero. Figure 3 shows two-dimensional FE model of one half of dental assembly with symmetric BC (named *Frictionless Support*) and displacement of 1 mm used as a load. *Frictionless support* is applied along the axis of symmetry (position C in Figure 3), while *Fixed Support* is applied along edge A (which is the part of the assembly with restricted displacement in all directions). Finally, vertical upward displacement (represented by arrow) of 1 mm is applied along edges denoted by B and D. As it can be seen, no forces were used in this FE simulation.

To conclude, with rapid improvements of digital technologies and user friendly software interfaces, the FEM has become available not only to aerospace and civil engineers but to doctors and dentists who can use this powerful technique to analyze complex biomechanical structures. The modeling and simulation save time and money for conducting the live experiment or clinical trials. By understanding the basic theory, method, application, and limitations of FEA, the clinicians will be better equipped to interpret results of FE studies and apply these results to clinical situations.

The following four case studies – based on the results of FE simulations carried out in collaboration with professors from the Faculty of Dental Medicine, University of Belgrade – demonstrate the full potential of FEM application in dentistry. These studies have proved robustness of FEM in addressing biomedical problems that are challenging for conventional methods.

Case study 1: Evaluation of stress distribution under FPD supported in three different ways

In this study[9] FEA was carried out to evaluate stress distributions in supporting tissues in four-unit tooth-tooth, implant-implant and tooth-implant supported dentures. The section contours of the alveolar bone, teeth, and fixed prosthesis were obtained from ATOS scanner and imported into CAD/CAM software CATIA where three different solid models were obtained. Solid model of the implant was constructed in CATIA and imported – along with bone, teeth and prosthesis models – into the software for FEA Ansys Workbench. After meshing the models, setting up material characteristics, boundary and loading conditions, calculations were performed. Materials were assumed homogenous, elastic and isotropic (with Young's moduli and Poisson's ratios taken from[10]), and 300 N axial and oblique load (making an angle of 30° to the long axis) were applied on pontic and all four units.

FEA showed that maximum stress values occurred at the neck of implant and tooth in all models for occlusal forces, while in a case of axial and oblique loading on the pontic the highest stress value was located at the implant neck for tooth-implant supported fixed partial denture (Figure 4 and Figure 5). After reviewing all load combinations and fixtures it was concluded that implant-implant supported fixed partial denture had a better stress distribution compared to tooth-implant fixture.

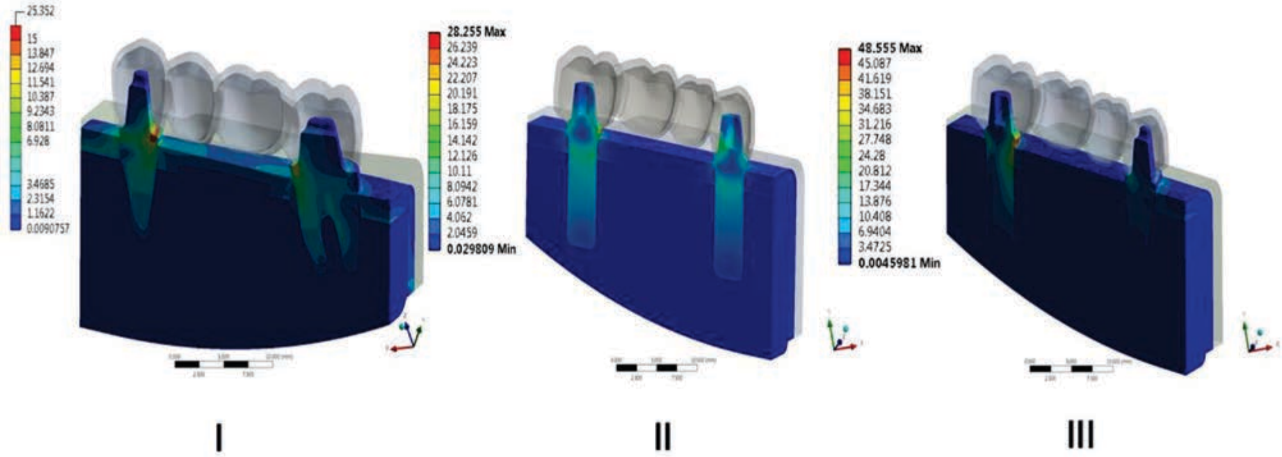

Figure 4. Stress distribution in supporting tissue of four-unit tooth-tooth (I), implant-implant (II) and tooth-implant support under axial load (III)

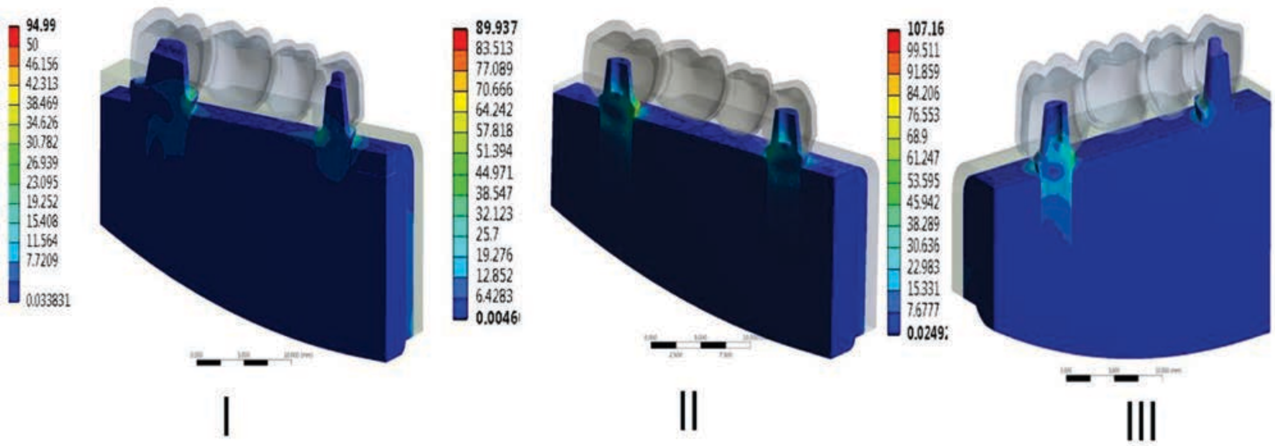

Figure 5. Stress distribution in supporting tissue of four-unit tooth-tooth (I), implant-implant (II) and tooth-implant support under oblique load (III)

Case study 2: Analysis of stress distribution and deformation under FPD for two diverse types of tooth-implant connection

In this case, FEM was used to compute stress distribution in surrounding tissue of a fixed partial denture with tooth-implant connection for two diverse types of implants: resilient Titan Shock Absorber (TSA) abutment was compared to conventional non-resilient abutment[11].

Two 3D models were created for this purpose. It was assumed that first and second molar were missing and implant was mounted in the second molar position for both cases. Modelling of the implant and abutment was performed in accordance with dimensions and recommendations obtained from the manufacturer. Virtual model was comprised of tooth contours, periodontal ligament, mucous membranes, implant, cortical bones and spongiosa, abutment and suprastructure. Figure 6. shows model of tooth-implant supported fixed partial denture.

Again, all used materials were assumed homogenous, linear elastic and isotropic with exception of periodontal ligament which was modelled with 3D non-linear highly elastic spring elements to better replicate its real properties in the FE model. Finite element mesh is shown in Figure 7.

Three different load conditions were taken into consideration with vertical force of 500 N. In first case force was introduced on the tooth, in second above the implant and in third case above all three units. Such models were analyzed using Ansys Workbench software which provided stress distribution for both models and all three load cases. It was observed that the resilient TSA abutment helped load dissipation and that the stress values are lower compared to non-resilient abutment. Deformation results for one type of the load are displayed in Figures 8 and 9 below.

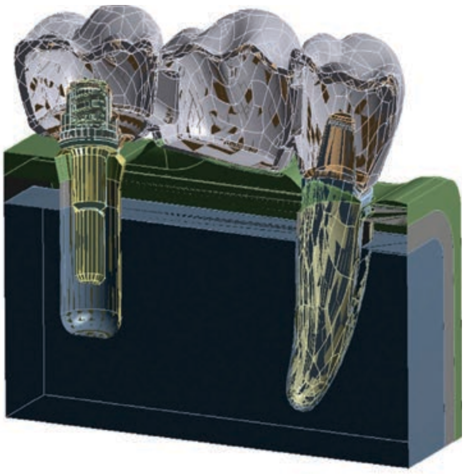

Figure 6. 3D model of tooth-implant supported fixed partial denture

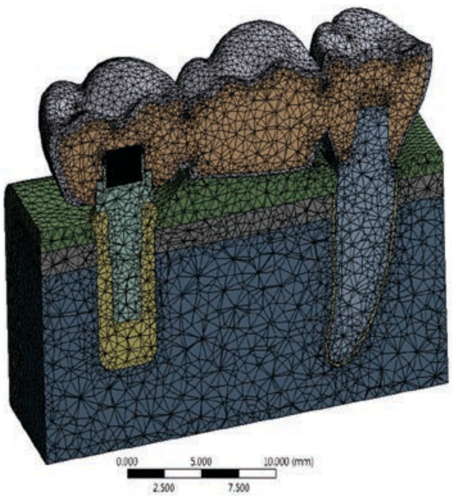

Figure 7. Finite element mesh of the fixed partial denture

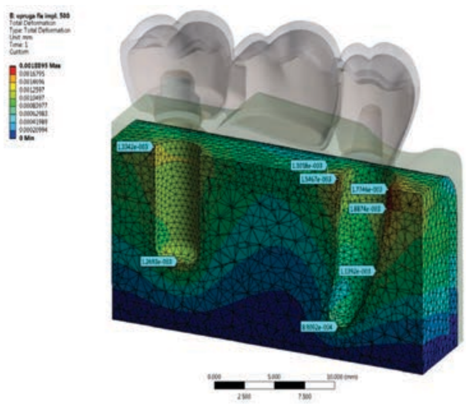

Figure 8. Total deformation in case of force acting above implant with resilient TSA abutment

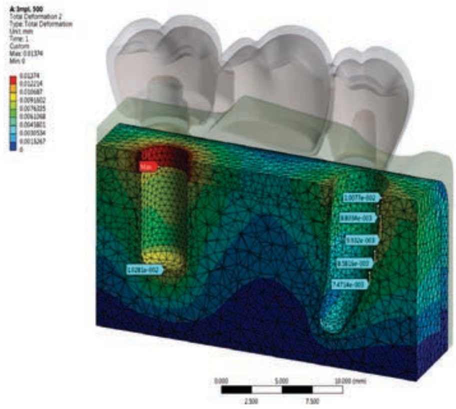

Figure 9. Total deformation in case of force acting above implant with non-resilient abutment

Case study 3: Analysis of stress distribution in adhesive inlay bridge

Adhesive fixed partial dentures are a part of a group of minimally invasive fixed constructions, with inlay retainers instead of classic-shell retainers. Reduced contact area, however, weakens the retention and makes dentures prone to separation from teeth over time. Since this is a consequence of extensive pressure on the cement layer, stress states of this layer and surrounding elements can be observed, analyzed and interpreted using FME[12].

Two extracted teeth were selected and scanned to simulate a lateral dental segment with partial edentulism[13]. A space of 13 mm was left between the teeth which were prepared with interproximal slots of specific dimensions. The corresponding fixed partial denture (FPD) made of ZirCAD was digitized with a scanner and imported into Ansys Workbench. An assumption was made that the teeth are not fixed and that the root is allowed a certain amount of vertical displacement equal to approximate distance between the alveolus and the root (about two hundredths of a millimeter). Therefore, the whole model was supported by elastic springs, which can simulate assumed behavior.

To simulate masticatory force three areas on the outer inclines of the buccal cusps were loaded (Figure 10). The total load of 225 N was applied along three directions in three successive simulations: at 90^0 to the tooth long axis (horizontal force), at 45^0 to the long axis (oblique force) and along the tooth axis (vertical force). Figures 11 and 12 shows stress distributions in the most critical case when horizontal load was applied.

As it can be seen in Figures 11 and 12, maximum stress occurs at contact areas of the bridge and supporting teeth. Stress is concentrated along the edges and over time this stress concentration may lead to tooth damage and, consequently, separation of the denture from tooth. Fatigue cracks always start at stress raisers, so removing them the fatigue strength of the system increases. To prevent cracks or, at least, to reduce the probability of their initiation the fillets at the sharp edges should be created. FEM might be very useful tool in determination of the optimized radii of these fillets.

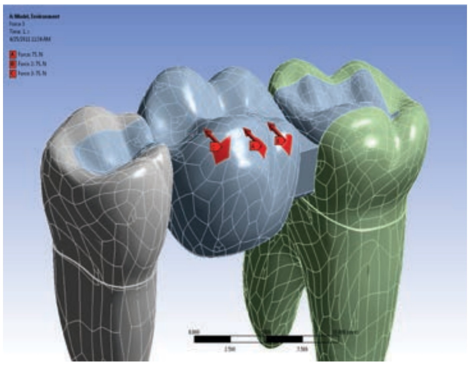

Figure 10 Loads used in FEA of adhesive inlay bridge

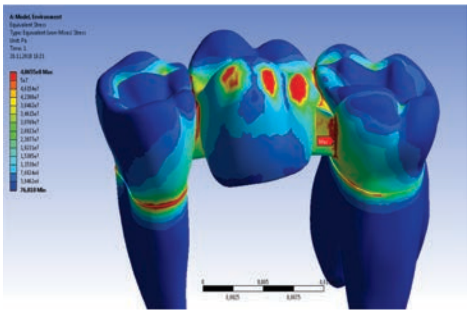

Figure 11 Stress distribution in the case of horizontal loading

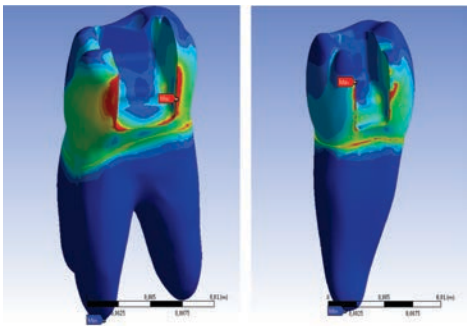

Figure 12 Stress distribution on the second molar and premolar in the case of horizontal loading

Case study 4: Optimization of the number of MDIs to support complete denture. Simulation of crack growth in MDI under dynamic load.

The main goal of this FE study[14] was to determine required number of mini dental implants (MDI) needed to fix complete denture in a case of poorly developed alveolar ridge. MDIs play key role in stabilization of complete dentures, especially lower ones. Due to small dimensions, MDIs can easily be applied even in extremely narrow and small ridges which are common with patients using dentures for a longer period. They contribute to solving problems of retention and stabilization, speech difficulties, etc.

3D model used in this study can be seen in Figure 13. Denture and plaster model of poorly developed alveolar ridge were scanned and imported into CATIA v5, while MDIs were constructed directly in software.

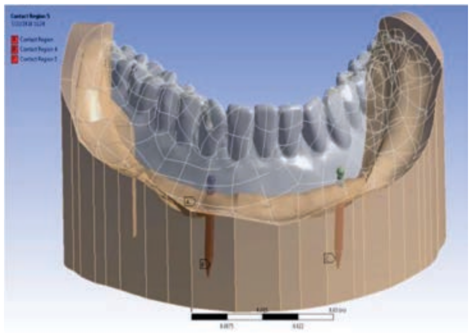

Figure 13. 3D model of full denture with two mini dental implants

Before adding implants, calculations were performed with denture being supported by the alveolar ridge only. Results showed high displacements which indicated denture non-stability. Then, calculations were carried out with denture supported by MDIs and obtained stress distributions on alveolar ridge and implants can be seen in Figure 14 and 15 respectively. It was obvious that the MDIs were asymmetrically loaded which was caused by asymmetry of the 3D model and loads applied. Only in a case of completely symmetrical model stress distributions would be the same on both MDIs. Since the stress values did not exceed the limits defined by material characteristics, it was concluded that two MDIs could provide substantial support for lower complete denture.

Figure 14. Stress distribution on alveolar ridge

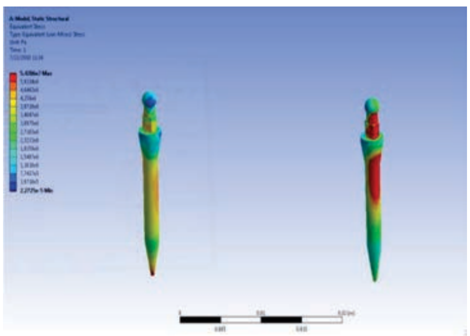

Figure 15. Stress distribution in mini dental implants

Occasionally, during the installation of MDI micro cracks may be initiated on the implant's surface and their growth can cause – after a certain number of chewing cycles – fatigue failure of MDI in cyclic stress environment. To investigate this phenomenon existing FE model of MDI has been used and crack growth in more stressed implant (right MDI in Figure 15) was analyzed[15]. To simulate fatigue crack growth in implant, it is necessary to define realistic dynamic loads. In this study, horizontal and vertical forces of magnitudes 0.005 N were applied on spherical part of MDI during two consecutive time intervals (each interval was 1 second long). Later, these values were multiplied by cyclic load of amplitude 20000 N to simulate fully reversed load with constant amplitude of approximately 100 N. Since load on MDI, caused by changeable masticatory force acting on the denture, is mostly less than 100 N and varies with time, a random load spectrum (maximum amplitude of 20000 N) was used for crack initiation predictions (Figure 16). Time defined for this spectrum was 94 seconds.

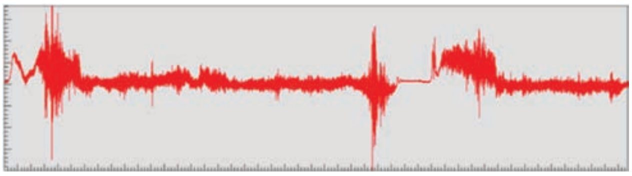

Figure 16 Random load spectrum used for crack initiation analysis

After the horizontal and vertical force had been applied, FE analysis was performed and obtained stress values were 0.0459 MPa and 0.00639 MPa, respectively. This is because small loads (0.005 N) were applied on MDI. But, during the second phase of the simulation these loads were multiplied by previously described random spectrum to get realistic dynamic loads. In Figure 15 the critical area of MDI (in terms of crack appearance) is showed in red color and the number of blocks of load spectrum which would initiate the crack was found to be between 533 and 20000. This means that total time before crack would start to grow is between 14 and 500 hours (total time depends on number of appearances of peak values during chewing).

The first step in setting the crack growth properties was to define the initial crack length. The value of 0.05 mm was chosen. The second step was to define crack geometry. It is assumed that the crack in the implant is much like a semi-circular crack in tension. Once the model was created in ANSYS, the steps necessary to perform crack growth analysis were: read the mesh information, rebuild the mesh around the crack, perform the Ansys analysis and compute new crack length[16].

The program begins the process of inserting the flaw into the original model and then meshes the resulting cracked model. Figure 17 display the computed crack front in the model after five and ten steps of calculation. After each step the extension was scaled, polynomial was adjusted to fit through the new crack front points and the polynomial extrapolation was adjusted to ensure intersection with the model surface.

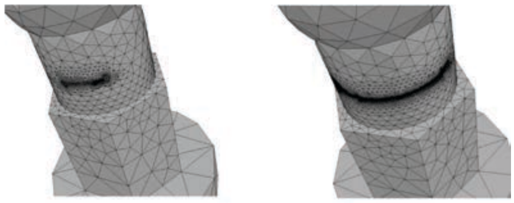

Figure 17 Steps of crack growth in MDI

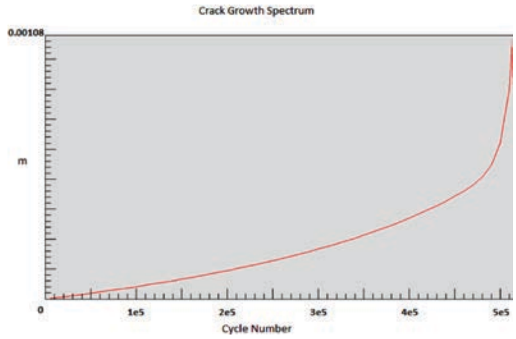

Figure 18. Crack length as a function of number of chewing cycles

The plot in Figure 18 shows crack length versus cycle number. It must be noted that the final crack length was found to be just over 1 mm after 500000 cycles. This result confirmed the expectation that even damaged implant would work well for an extended period. This also concurs with clinical practice experiences, which show that the fatigue failure of MDI rarely occurs during the exploitation. Given that the worst load case scenario was used (load spectrum was very abrupt and rather long), it can be concluded that the least designed life of slightly damaged MDI of 500000 cycles is acceptable.

Discussion

FEM has been shown to be a useful tool when investigating complex systems that are difficult to standardize during in vitro and in vivo studies. It has been used mostly to evaluate the influence of the type of material and geometry on the stress distribution and deformation during chewing cycles. Most of the studies employed linear static models which are valid if the structure exhibits a linear stress–strain relationship and all the volumes are bonded as one unit. However, realistic testing situations give rise to dynamic models and nonlinearities, which can be classified in two main categories: (1) material nonlinearities (that cause the stiffness to change with different load levels) and (2) geometric nonlinearities (such as nonlinearities in the vicinity of a crack tip).

Main dilemma in application of FEM in dentistry is to which extent is the numerical model equivalent to the real biological system. Many studies have shown significant trend regarding advancements and optimizations of FE models. Enhancements in software and hardware have significant positive impact on this trend. However, it is important to emphasize that even with all improvements, it is still impossible (and will be impossible in near future) to fully replicate the complexity of the human body.

Bone, mucoperiosteum or teeth are complex non-homogenous and anisotropic structures that are simplified in FEA to be adapted for calculations. These simplifications do not imply that the results obtained from such models are useless, but should be taken with caution. They are not conclusive and must be supported with clinical researches. Examples presented in this paper showed that the most valuable result of FEA was identification of critical (high stress) areas of the physical model.

Resulting stress distributions are useful, but the stress values that would lead, for example, to the mandible fracture are still not unquestionably defined and there is no evidence of bone rupture due to implant overload in clinical practice. Vast number of researches was pointed to the prediction of bone mass loss in the implant surroundings due to overload. It must be borne in mind that this concept is still insufficiently explored and that there is no reliable evidence of connection between these phenomena. Bone resorption is a dynamic process primarily influenced by biological factors.

For the aforementioned reasons, it can be concluded that the finite element method should be used as a tool for comparison and analysis of similar cases and not as a tool for drawing final conclusions. Future applications should be focused to optimization and verification of bone models, temporo-mandibular joint, teeth, implants, dentures, etc. Usefulness of FEM should be reflected in testing the new hypotheses which can be later confirmed in vivo. Differences (or similarities) in hypotheses identified by means of finite element calculations can be solid foundation for further clinical researches.

Note: The results of this paper were presented as a part of an invited lecture at the 22nd BaSS Congress.

References

1. Stomatološki fakultet. Stomatološki materijali knjiga 2: Beograd; 2012.
2. Ming-Lun Hsu, Chih-Ling Chang, Application of Finite Element Analysis in Dentistry, Finite Element Analysis, ed. David Moratal, 2010.
3. Geng J, Yan W, Xu W (Eds.). Application of the Finite Element Method in Implant Dentistry, ISBN: 978-3-540-73763-6, Springer, 2008.
4. Duygu Koc, Arife Dogan, and Bulent Bek, Bite Force and Influential Factors on Bite Force Measurements: A Literature Review. Eur J Dent, 2010;4:223-232.
5. Olmsted MJ, Wall CE, Vinyard CJ. Human bite force: the relation between EMG activity and bite force at a standardized gape. Am J Phys Anthropol, 2005;40:160-161.
6. Rangert B, Jemt T, Jörneus L. Forces and moments on Branemark implants. Int J Oral Maxillofac Implant, 1989;4:241-247.
7. Mericske-Stern R, Piotti M, Sirtes G. 3-D in vivo force measurements on mandibular implants supporting overdentures. A comparative study. Clin Oral Implants Res, 1996;7:387-396.
8. Akyuz E, Braun JT, Brown NA, Bachus KN. Static versus dynamic loading in the mechanical modulation of vertebral growth. Spine, 2006;31:952-958.
9. Erić J, Tihaček Šojić Lj, Lemić A, Bjelović Lj, Grbović A. Evaluation of stress distributions in supporting tissues in four-unit tooth, implant-implant and tooth-implant supported dentures by finite element analysis, Belgrade, 2016.
10. Lin ChL, Wang JCh, Chang WJ. Biomechanical interactions in tooth– implant-supported fixed partial dentures with variations in the number of splinted teeth and connector type: a finite element analysis. Clin Oral Impl Res, 2008;19:107-117.
11. Glišić M, Stamenković D, Grbović A, Todorović A, Marković A, Trifković B. Analysis of load distribution on tooth-implant supported fixed partial dentures by the use of resilient abutment. Srp Arh Celok Lek, 2016;144:188-195.
12. Grbović A, Vidanović N, Čolić K, Jevremović D. The use of finite element method (FEM) for analyzing stress distribution in adhesive inlay bridges, Third Serbian (28th Yu) Congress on Theoretical and Applied Mechanics, Vlasina lake, Serbia, 5-8 July 2011.
13. Jevremović D. Distribucija i koncentracija napona u adhezivnim mostovima, Doctoral Thesis, Belgrade, 2008.
14. Grbović A, Tanasković M, Vidanović N. Comparative analysis of stress distribution on the toothless alveolar ridge at the bottom of the complete denture prosthesis and overdenture retained with mini implants, 2nd International Congress of Serbian Society of Mechanics, D-14, 2009.
15. Grbović A, Rašuo B, Vidanović N, Perić M. Simulation of crack propagation in titanium mini dental implants (MDI). FME Transactions, 2011;39:165-170.
16. Kastratović, G. Determination of Stress Intensity Factors of Supporting Aero Structures with Multiple Site Damage, PhD thesis, Faculty of Mechanical Engineering, University of Belgrade, Belgrade, 2006.

Received on May 1, 2017.
Revised on May 11, 2017.
Accepted on May 22, 2017.

Correspondence:

Aleksandar Grbović
Faculty of Mechanical Engineering
University of Belgrade, Serbia
e-mail: agrbovic@mas.bg.ac.rs