Applied Medical Informatics
Communication Vol. 26, No. 2/2010, pp: 73-82

The Use of Numerical Applications in the Study of Dental Contacts

Mihai-Cătălin FRUNZĂ[1,*], Gheorghe FRUNZĂ[2], and Rodica LUCA[1]

[1] University of Medicine and Pharmacy, Faculty of Dentistry, 37 Dionisie Lupu Street, Bucharest, Romania.
[2] University of Suceava, Faculty of Mechanical Engineering, Mechatronics and Management, 13 University Street, Romania.
E-mail(s): frunzamc@yahoo.com; frunza@fim.usv.ro.

* Author to whom correspondence should be addressed; Tel.: +40723-423102.

Received: 8 May 2010 /Accepted: 10 June 2010 / Published online: 21 June 2010

Abstract

This paper seeks to explore the numerical analysis methods used in dentistry in general and those regarding teeth contacts, in particular. Typically, such an analysis consists of the following steps: modelling the actual object, mesh generation, numerical modelling and computer programming. The best known and mostly used of all is the finite element method. The paper also presents other more refined methods, for instance: CATIA and fast Fourier transform (FFT). The study of the living tissue based on numerical analysis exceeds the limitations of in vivo experiments but computers can never replicate the body adaptation capacity.

Keywords: Dental stress analysis; Finite element analysis; Dental occlusion.

Introduction

In the etiology of the dental-periodontal disease, in addition to microbial factors an important place is occupied by occlusal factors. Clinically, they are known as cracks and fractures, abrasion, cervical noncarious lesions, migration and tooth mobility, periodontal lesions, joint dysfunction. These occlusal factors are specifically related to the type of dental contacts and the muscular force developed. Following the approach of the two arches, as a consequence of the contraction of the mastication muscle, opposite teeth come into contact and are subjected to loads with a specific force. This will result in a stress state in the tooth, bone and periodontal structure. Over the years there have been made attempts at calculating these tensions, as it is assumed that they play a determining role in all of biophysical, biochemical and metabolic processes.

The paper presents the concept of numerical modeling, classical methods used for stress state computation and highlights some special methods.

Stress computation

The value of studying stresses consists in a better understanding of the distribution of occlusal loads, failures of different direct and indirect restorations and stomatognathic system homeostasis.

The stress computation itself can be achieved by three methods: analytical, numerical and experimental. Briefly, analytical methods require the creation of a mathematical model, characterized by a system of equations, which can be solved by methods which are characteristic to mathematical analysis. However, the calculation procedures are very complicated and almost impossible to realize.

Experimental studies can be performed in vivo or in vitro. They have traditionally been performed by means of strain gauges, photoelastic methods or on simple dental contacts, and are nowadays carried out by using the atomic force microscope or by employing nanoindentation. One of the main drawbacks is the high cost of equipment; in addition, it requires specialized personnel and conducting the experiment is rather difficult. But if the experimental model is well designed, the results will be very accurate.

Numerical studies are the most common and constitute the subject of the present paper.

Numerical Modeling

Mathematical models are an integral part of several problem solving methods encountered in bioengineering, medicine, biology, physics and chemistry. These mathematical models are frequently based on engineering knowledge or scientific principles and others derive from experimental data [1]. Mathematical models generally require the use of mathematical proceedings based on differential equations. To solve these equations, an approximate solution using numerical methods is often preferred.

The computer-assisted approach to biomechanical problems typically consists of the following next steps [2]:

a. real system modeling

This corresponds to an idealization of the real system, taking into account only the most important elements in the analysis. The physical model thus obtained is often accompanied by partial differential equations, which together form a mechanical model.

The mechanical model retains the configuration and the essential characteristics of the real body. It should be noted that the effectiveness of the resolution depends heavily on mastering all the essential elements even since the level of physical modeling.

b. discrete representation of the physical model

The physical models are then typically subjected to meshing. Discretization can be complete, resulting in a discrete model represented by systems of algebraic equations or only partial (semi-discrete), which often leads to a system of ordinary differential equations without consideration of time.

The most common methods of meshing are offered by numerical methods, such as the finite element method, border element method and finite differences method. It should be noted that some real systems do not require generating a physical model, because the discrete model can be created directly from real system considerations.

c. numerical modeling of discrete model

This step corresponds to the selection of numerical algorithms for solving the equations that describe the meshed model. The choice of specific computer applications may significantly influence the efficiency of the solving procedure. At this stage, the analyst's familiarity with modern numerical algorithms is generally required, as it can contribute to solving large systems of linear and nonlinear algebraic equations efficiently, ordinary differential equations, linear or nonlinear mathematics programming problems, etc.

d. computer programming

The computer implementation step creates a code written in one of the programming language. The code is just another version of the system model, which was created to suit the available "hardware".

e. Generating the input data in a prescribed form.

f. Running the program.

g. Verification of the used models and interpretation of results.

Next we shall present the method of numerical analysis most frequently used and two lesser-known methods.

Modeling by Finite Element Method (FEM)

The finite element theory, a branch of numerical analysis, first appeared in the work of Courant (1943) [3]. The finite element method is an approximate solving method on computers of a wide variety of problems. In general, the physical phenomena are described mathematically by differential equations, by whose solving the exact solution of the problem is obtained. However, such an analytical approach is rarely possible due to computing difficulties.

In such cases, two solving methods can be employed:
- obtaining exact solutions by simplifying the real model
- obtaining approximate solutions for the real problem, using numerical methods.

Research has shown that an approximate solution of the real phenomenon is often preferable, as it can reflect reality better than an exact solution of the simplified model. The finite element method belongs to the second approach.

In essence, the basic idea of FEM refers to approximating unknown functions (temperature, strain, flow rate, pressure) with a specific function (type "splines"), so that on sufficiently small subdomains called finite elements, the result would be sufficiently close to the exact solution [4].

The method comprises three main stages: preprocessing, analysis and post processing.

I. Preprocessing

At this stage, the realization of the analyzed object model and its discretization take place. Meshing the model means dividing it in smaller regions and is performed with a view to seeking a solution to the proposed equations, not for the whole field, but for each part separately [5].

Meshing the domain of analysis (surface, body, and structure) includes:
- selection of a limited number of elements in the body, in which, instead of exact solutions, we shall have approximate solutions for the parameters to be analysed;
- choice of one-, two- or three-dimensional geometric patterns, depending on the body geometry and on the number of independent spatial coordinates.

The whole is consequently divided into a number of smaller regions called finite **elements**, while their contours may be lines and/or plans, straight or curved. Naturally, examining the properties and characteristics of a single finite element is more easily performed than that of the whole body.

Intersections of contour lines are called **nodes** and the border between the finite elements is called **nodal line** or **nodal plane,** as appropriate.

A finite element belonging to a whole features:
- Geometric properties (shape, size);
- Physical properties (elasticity, density, viscosity, etc. - by problem type);
- Functional properties - approximates one or more problem variables in the space occupied by real body (displacement, stresses, strains, etc.).

A proper meshing of the global domain in finite elements involves certain conditions, such as:
- Two adjacent finite elements can have in common only points located on the common border;
- The join of all elements must lead to a field that bears considerable similarity to the initial field. No openings are allowed between elements or their overlap.

Depending on the geometry of the analyzed body and the number of independent spatial coordinates used to define the geometric, physical and functional properties, we can distinguish three types of finite elements, as shown in Table 1.

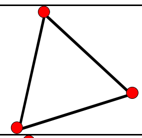
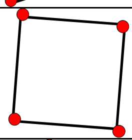
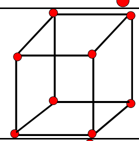
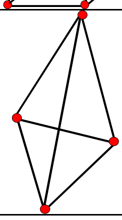
Basically, to solve a problem on the basis of the finite element method, the analyst must decide at the outset, the type or types of finite elements to be used. On this activity does the time required for running computer programs and data preparation time depend? The geometric shape of the analyzed body offers a first indication of the types of items to be used.

In some cases, it may take two or more types of finite elements, such as: two-dimensional and three-dimensional elements with triangular and quadrilateral faces coupled with one-dimensional elements.

The dimensions and the number of finite elements must be carefully chosen to obtain a minimum difference between the approximate and the exact solution of a given problem. This difference is called error and refers to one or more unknown nodes (displacements, specific deformation and stress). The exact solution can be known, in simple cases, by analytical methods, while the real solution is obtained by experimental measurements or other means.

A comparative analysis of several variants on the size, number and shape of finite elements is often performed. A modern version is preprocessing, i.e. the use of specialized software to solve this problem.

Table 1. Types of finite elements

Finite element name	Type	Geometric Representation
One-dimensional finite element	Straight nodal lines	
	Curved nodal lines	
Two-dimensional finite element	triangular	
	quadrilateral	
Tridimensional finite element	hexahedron	
	Tetrahedron	

II. Analysis

The discretization results as well as the date prepared in the first phase are introduced in the finite element program, which will build and solve a linear or nonlinear algebraic system of equations.

III. Post processing

The highlighting of the system of equations solutions occurs in graphical form or imaging. Furthermore, the stress level of the model is chromatically depicted.

The computer applications that use this method are known as: Ansys, Cosmos, and Nastran.

Advantages:

- It can be used by persons less specialized, the calculation being quite easy and fast;
- It minimizes the subjective contribution of the user in that it requires only information about how to mesh the structure;
- The degree of generality is high enough so that it can solve an extensive number of issues encountered in dentistry;

Disadvantages:

- Optimally meshing a structure is a rather difficult issue;

- Imposing boundary conditions requires maximal attention. Their modification may lead to totally different results. They specifically refer to how to fix the tooth in the alveolar bone or to the crowns alone, to the enamel-dentin bond and so on;
- The geometric configurations of the tooth cannot be achieved with precision using finite elements, so the result is only an approximate outline.

The 2nd Table includes several representative studies in which researchers have been using this method, from its beginnings up to the present day.

Table 2. Studies using FEM

No.	Author, Year [ref]	Modeling approach	Topic
1	Farah and Craig, 1973 [6]	2D of the 1st molar	Type of the abutment preparation: knife edge, chamfer, beveled shoulder
2	Weinstein, 1976 [7]	2D of the mandibular canine area with an implant	Root form implant of porous Co-Cr-Mo alloy
3	Yettram et al., 1976 [8]	2D of the premolar in two cases: isotropic and orthotrop enamel	Normal and restored tooth with a gold crown
4	Derand, 1977 [9]	2D of a premolar, charged only on the vestibular cusp at 30º	Marginal behavior of an amalgam restoration
5	Spierings et al., 1984 [10]	2D of a normal mandibular molar, and then with an amalgam filling and with different bases	Temperature distribution and heat transfer
6	Anusavice et al, 1986 [11]	2D of a maxillary central incisor, restored with a ceramic-metal crown	The influence of metal cap thickness and metal type
7	De Groot et al, 1987 [12]	3D of a mechanical testing composite samples	Applying different mechanical criteria to predict a fracture
8	Peters et al, 1993 [13]	2D of a restored premolar using ceramic inlay MOD	Occurrence of cracks in concentrated and then distributed loading
9	Korioth and Hannam, 1994 [14]	3D of the entire mandible, including TMJ and teeth, cortical bone - found ortotrop	Mandible deformations occurring at different tooth contacts
10	Lin and Douglas, 1994 [15]	3D of an enamel-dentin-composite specimens	The fracture behavior of the enamel-dentin assembly
11	Spears, 1997 [16]	3D of the enamel prisms, including their crystal	Enamel elasticity modulus (anisotropy)
12	Oh et al, 2002, [17]	3D of all ceramic three elements bridges	Connector shape and the risk of bridge fracture
13	El Zohairy et al, 2004 [18]	3D of composite specimens for testing fracture	The influence of size and ways of fixing upon results
14	Soares et al, 2008 [19]	3D of ceramic-dentin specimens for testing the adhesion	The influence of shape and ways of fixing upon the results

Numerical Modeling using CATIA Application

This is a powerful CAD-CAM software (computer-aided design and computer-aided manufacturing) designed by Dessault Systemes and marketed by IBM. CATIA can be used to design a wide range of objects: airplanes, jewelry and clothing.

When talking about CATIA, the notion of **digital modeling** is often used. This term indicates all computerized data that allow analyzing an object as or better than one could do it with a real model or prototype. This digital model can be tested in different conditions and its viability can be checked. It also allows lowering costs and time and increasing quality, because it avoids passing through prototype or actual model phases. Furthermore, subsequent changes become easier to accomplish. By virtue of the design and the development module, one can simulate the manufacturing of an element from a structure on digital operational machines and can automatically generate the digital file of production, which is used by the real digital machine [20].

The CATIA application includes the following facilities:

- The ability to design various parts and assemblies directly in the 3D mode, without drawing their plan;

- The common CAD-FEM environment and generative and associative working capacity allows a large number of evaluations of the mechanical behavior of components and assemblies in the early stages of development;
- The analysis can be carried out under static or dynamic conditions, which represents a major feature of the program.

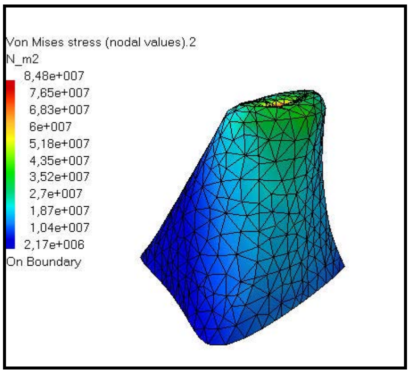


Figure 1. Upper incisor model using CATIA software

Next we shall present an analysis by FEM using the CATIA computer application. It was desired to calculate the stresses and strains in the superior incisor enamel layer [21].

The steps of the method's application were as follows:

1. **Designing the graphical model**. One can use either the module Part Design of the CATIA application or can import it from an external CAD application. Average anatomical shapes and sizes known in literature were considered [22]. The difference from a normal design is that in this program each point of the design is digitized (e.g. a binary code is attached). For the most accurate results, the graphic model must be as close to reality as possible. This is the most important and delicate stage.
2. **Choosing the material**, either by specifying its physical and mechanical characteristics (modulus of elasticity, Poisson ratio, yield and fracture strength, etc.), or by its choice from the libraries of CATIA.
3. **Boundary conditions**, using CATIA specific functions, from the windows of the program. Given a 3D model of a dental crown, in order for it to be loaded, it must be fixed somehow, as the crown is fixed in the bone by means of the dental root.
4. **Placing loads** by specifying the type of load (concentrated or distributed), the intensity and direction of its application and the operating surface.
5. **Meshing the domain in finite elements**. CATIA software automatically generates this operation but the user has provided many tools and features to manage the type and size of the finite elements. Optionally, finer meshes can be achieved in the areas of interest.
6. **Launching the calculation and interpretation of results**. One obtains data about the stress and strain state, the von Mises equivalent stress (representative for the local plastic deformation moment). Their numerical values are correlated with a color code that can facilitate viewing highly stressed areas.

Advantages:

- It uses MEF alongside the border element method (a variant of the first) so that the tooth outline is copied exactly by means of finite elements;
- Accomplishment of automatic meshing and increase refinement in desired areas;
- Speed of calculation;
- Simulating deformations and stresses in operational conditions and gathering all in a small movie;

- Finally, a CATIA report is obtained, which contains all data, all steps and all necessary information for analysis and interpretation. This report may be submitted as a Word or PDF document type, so installing the program on another computer is not necessary;
- It uses the graphical model for its physical realization with a CAD-CAM machine interface, which means that different dental restorations can be performed practically.

Disadvantages:

- The high cost of the CATIA program;
- 3D design thorough implementation, which requires specialized training;
- The use of composite structures requires imposing strict boundary conditions at interface (enamel-dentin). The computation is therefore preferable either in the enamel or in the dentin.

Numerical Modeling using Fast Fourier Transforms

With a view to solving differential equations (which are the mathematical language of physics) and convolutions (a type of mathematical operation), one of the Fourier mathematical analysis forms may be used, namely the Discrete Fourier Transform (DFT). It transforms a function into another function. DFT has a wide range of applications. But the possibility of using it depends crucially on the existence of a fast computing algorithm. The most widely used algorithm is the Fast Fourier Transform (FFT). It reduces the working time a few hundred times.

This algorithm may also be used in order to calculate stresses in a given body. As one can notice, the process is based on advanced mathematical knowledge; only the main steps shall therefore be further described below:

1. Obtaining data on the tooth **surface topography** in numerical form (digitized). This can be done by 3D scanning or 2D laser profilometry, as shown in Figures 2 and 3. From a mathematical point of view, what is to be obtained will be an m x n matrix, with which we'll carry on.
2. Consider the situation of a dental hertzian, non-hertzian or on the area contact that can be calculated using the **equations of mechanic contact**.
3. The initial matrix is taken with a **computer application** type Mathlab or Mathcad.
4. Under the aforementioned applications, perform a **program that is based on DFT and / or FFT**.
5. By employing this numerical method a **stress/strain state** in the tooth body will be obtained.

Figure 2. μScan Technology [23]

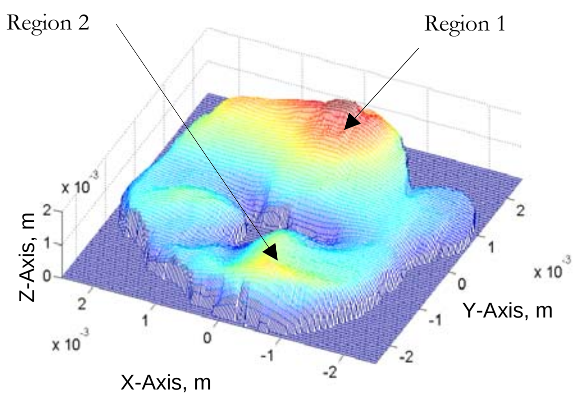
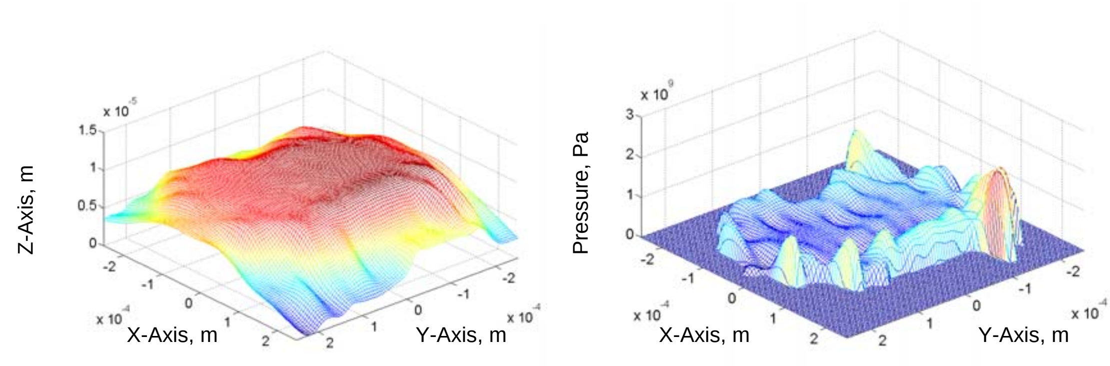


Figure 3. Digitized occlusal surface topography [23]

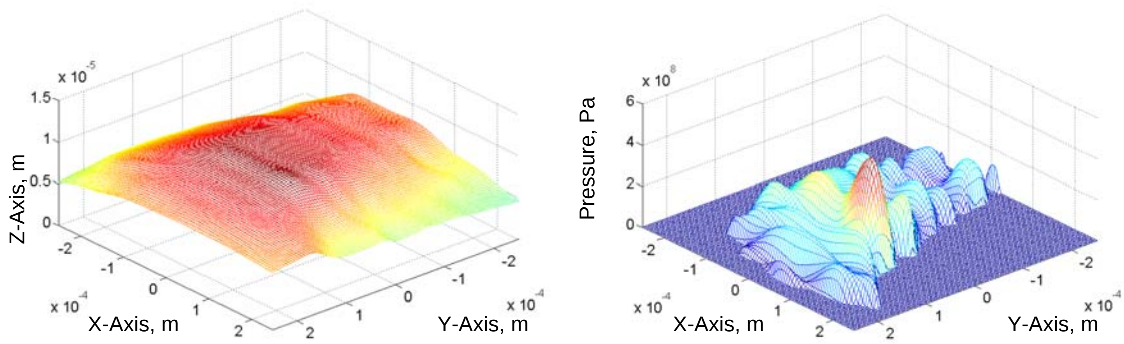
The two regions, depicted in Fig. 3, which correspond to active contacts, were selected for analysis. For each region, a topography sample was obtained from the original array by clipping the edges of the data set. The two resulting grids, of 128 × 128 cells each, are represented in Figure 4.a and Figure 5.a. The choice of these sets was made so that the height would not vary very abruptly on the selected region, a condition required by the program.

The following data was inputted to the elastic contact solver: normal load transmitted through contact, Q = 50N, friction coefficient, μ = 0.025, enamel Young modulus, E = 67.5 GPa, enamel Poisson's ratio ν = 0.31.



a. *b.*

Figure 4. Region 1, roughness sample (a) and pressure distribution (b) [23]



a. *b.*

Figure 5. Region 2, roughness sample (a) and pressure distribution (b) [23]

Figure 6 depicts the von Mises equivalent stress induced in the enamel layer. Immediately in the subsurface, the stresses vary abruptly due to surface roughness irregularities. A small step on the z-axis would be advisable in order to achieve detailed information. However, a longer depth interval can be adopted for deep layers, where stress field is much smoother. If a full detail is required even for deep layers, additional information can be obtained using interpolation between computed nodal values [23].

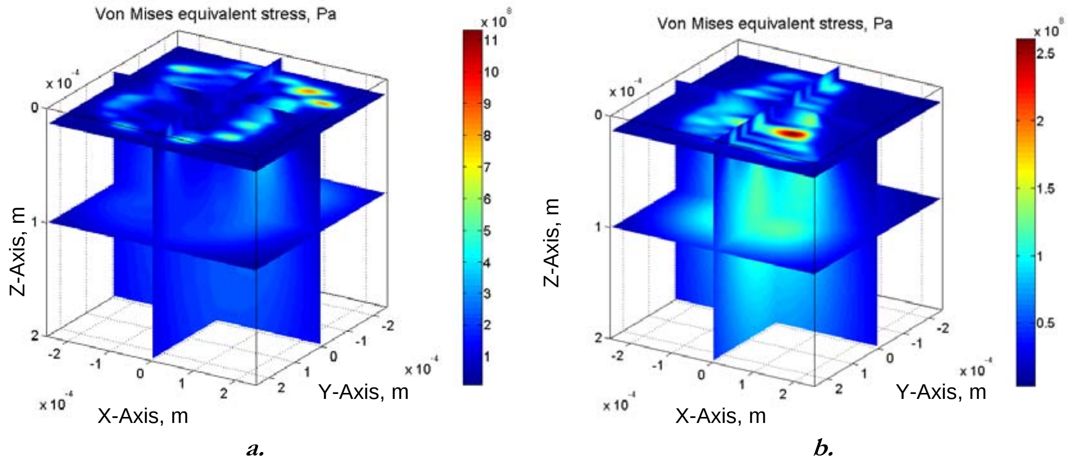


Figure 6. Subsurface Von Mises equivalent stress: a. region 1; b. region 2 [23]

Characterization of the Method and its Advantages

- It does not require the creation of a model that would subsequently be meshed. Obtaining the area in a digitized form (directly on computer) in the first stage is, in fact, a real meshing.
- It combines the analytical part (equations of contact mechanics) with the numerical part (computer programs for solving equations).
- Stress calculation can be performed only in small areas, in the interest areas or throughout the area.
- It uses clear equations of contact mechanics that can be customized by tooth and by surface.
- It is closer to the right solution because it uses a real dental surface and not an imagined one.
- It does not require the formulation of boundary conditions (often a source of error in FEA programs).

Disadvantages:

- Similar to CATIA program, working on composite materials, consisting of several layers (for e.g. the tooth) is difficult.
- Requires obtaining a very accurate topography.
- Profilometry can be executed only in a single plane (for e.g., the occlusal surface and axial ones are being approximated).

Conclusions

- The use of numerical methods in studying various dental issues led to clear results, with many clinical applications.
- Dental contacts, as a key element in the emergence of tensions inside the tooth, can be numerically modeled.
- The study by numerical analysis of living tissue exceeds the limitations of the in vivo experiments, but computers can never replicate the body adaptation capacity.

Acknowledgments

The research was financial supported by Romania CNCSIS-MECT through grant 757/2007-2008.

References

1. Kaw A, Introduction to Numerical Methods [online] [2010 aprilie]. Available from: URL: http://numericalmethods.eng.usf.edu/mws/gen/01aae/mws_gen_aae_spe_introduction.pdf.
2. Kleiber M. Handbook of computational solid mechanics. Springer, 1998.
3. Pletea AL. Numerical integration of differential equations: finite element method. PhD thesis, Iaşi, 1990.
4. Marinescu G. Finite element method. Numeric analysis and termoelastic aplication. Ed. CIA, Bucharest, 1996.
5. Blumenfeld M. Introducere în metoda elementelor finite. Tehnică, Bucureşti, 1995.
6. Farah JW, Craig RG. Finite Element Stress Analysis of a Restored Axisymmetric First Molar. J Dent Res 1974;53(4):859-866.
7. Weinstein AM, Klawitter JJ, Anand SC, Schuessler R, Stress Analysis of Porous Rooted Dental Implants. J Dent Res 1976; 55:772-777.
8. Yettram AL, Wright KWJ, Pickard HM. Finite Element Stress Analysis of the Crowns of Normal and Restored Teeth. J Dent Res 1976; 55:1004-1011.
9. Dérand T. Marginal Failure of Amalgam Class II Restoration. J Dent Res 1977;56:481-485.
10. Spierings TA, Vree JH, Peters MC, Plasschaert AJ. The influence of restorative dental materials on heat transmission in human teeth. J Dent Res 1985;63:1096-1100.
11. Anusavice KJ, Hojjatie B, DeHoff PH. Influence of metal thickness on stress distribution in metal-ceramic crowns. J Dent Res 1986;65:1173-1178.
12. De Groot R, Peters MCRB, De Haan YM, Dop GJ, Plasschaert AJM. Failure Stress Criteria for Composite Resin. J Dent Res 1987;66:1748-1752.
13. Peters MCRB, De Vree JHP, Brekelmans WAM. Distributed Crack Analysis of Ceramic Inlays. J Dent Res 1993;72:1537-1542.
14. Korioth TWP, Hannam AG. Deformation of the Human Mandible During Simulated Tooth Clenching. J Dent Res 1994;73:56-66.
15. Lin CP, Douglas WH. Structure-Property Relations and Crack Resistance at the Bovine Dentin-Enamel Junction. J Dent Res 1994;73:1072-1078.
16. Spears IR. A Three-dimensional Finite Element Model of Prismatic Enamel: A Re-appraisal of the Data on the Young's Modulus of Enamel. J Dent Res 1997;76:1690-1697.
17. Oh W, Götzen N, Anusavice KJ. Influence of Connector Design on Fracture Probability of Ceramic Fixed-partial Dentures. J Dent Res 2002;81:623-627.
18. El Zohairy AA, de Gee AJ, Jager N, van Ruijven LJ, Feilzer AJ. The Influence of Specimen Attachment and Dimension on Microtensile Strength. J Dent Res 2004;83:420-424.
19. Soares CJ, Soares PV, Santos-Filho PCV, Armstrong SR. Microtensile Specimen Attachment and Shape-Finite Element Analysis. J Dent Res 2008;87:89-93.
20. Wadeleux V. Guide d'utilisation du logiciel CATIA V5. [online] [2001 ianuarie]. Available from : URL: http://www2.ulg.ac.be/ltas-cao/pdf_cao/guide_CATIA_v5.pdf
21. Frunză G, Frunză MC. Stress state in enamel layer of dental contacts. J Biomech 2008;41(suppl 1):S217.
22. Scheid R. Woelfel's dental anatomy. 7th ed. Lippincott Williams&Wilkins, Philadelphia, 2007.
23. Frunză G, Spânu S, Frunză MC. Numerical simulation of pressure distribution in multiasperity dental contacts. Annals of the Oradea University. Fascicle of Management and Tehnological Engineering, 2009;VIII:236-239.