The Influence of Food Consistency on the Abutment Teeth in Fixed Prostheses

A FEA study

MONICA TATARCIUC, ANCA VITALARIU*, ODETTE LUCA, ANDRA AUNGURENCEI, OVIDIU AUNGURENCEI, DRAGOS FRATILA, DIANA DIACONU POPA

Grigore T. Popa University of Medicine and Pharmacy, 16 Universitatii Str., 700115, Iasi, Romania

The aim of our study was to evaluate through Finite Elements Analysis (FEA), the stress induced into the abutments and into a metallic bridge by an alimentary bolus of different consistency. Research was carried using the FEA on a model of the bridge with full crowns as retainers in the posterior teeth (34-36), obtained using a contact scanner and computer aided design (CAD) system. We surveyed the stress induced by different food consistency with elasticity modulus between 0 MPa and 60000 MPa. A 6MPa stress was induced by the bridge when the elasticity modulus was equal to 200 MPa. For the maximal value of the elasticity modulus, the stress was 13.68Mpa. The highest values of stresses are registered for the maximal values of the elasticity modulus.

Keywords: FEA analysis, elasticity modulus, food consistency

There were numerous ways and attempts of experimental research, but due to the complexity of dental structures, composed of various tissue materials mechanically and chemically interconnected, and due to complex tooth morphology and surrounding structures, these attempts failed to obtain precise and reliable results [1]. A successful prosthetic therapy depends on the patient's oral health status and, on the biological and biomechanical requirements of the bridge, on the materials chosen and the alimentary behaviour (food consistency). The finite element analysis (FEA) is a significant tool for biomechanical analyses in biological research. It is an ultimate method for modeling complex structures and analyzing their mechanical properties. FEA is widely accepted as a non-invasive and excellent tool for studying the biomechanics and the influence of mechanical forces on the biological systems [2]

Since it is fairly difficult to conduct an in vivo or in vitro assessment of the forces acting during mastication, the finite element analysis is preferred as, if the modeling is accurate, it may provide very useful information on the stresses [3-4]. Finite element analysis is a numerical method of analyzing stresses and deformations in structures which originated from the need of solving complex structural problems in civil and aeronautical engineering. In the field of dentistry, FEA has been used to simulate the bone remodeling process, to study internal stresses in teeth and different dental materials, and to optimize the shape of restorations. In order to achieve this goal, the structures are discretized into the so called 'finite elements' connected through nodes, each with specific physical properties.The type, arrangement and total number of elements impact the accuracy of the results [5-7].The steps followed are generally constructing a finite element model, followed by specifying appropriate material properties, loading and boundary conditions so that the desired settings can be accurately simulated [2].

Experimental part

Material and method

The modeling was done starting from the hypothesis that the dental elements are deformable structures under the action of various variable demands such as intensity, application point [8, 9]. A FE model representing a single tooth gap in the lateral left mandible, represented by the second premolar was created. The first premolar and first molar served as abutment teeth. The missing premolar was replaced by one unit pontic. The 3D images of the bridge with full crowns as retainers were obtained using a contact scanner and computer aided design (CAD) system (fig. 1, 2).

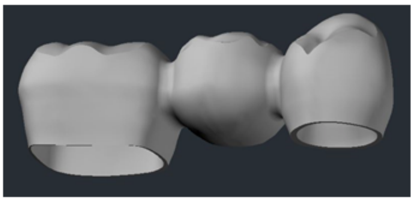

Fig.1 Model of the dental bridge

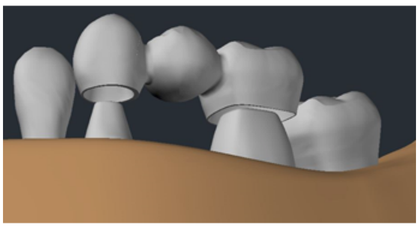

Fig.2. Model of the bridge with full crowns on the abutments

Mesh Generation

Meshing is a process that discretifies a certain solid volume in finite elements of the parallelepiped or tetrahedron type. Each element behaves as an entity with the same characteristics as the base material. After the type of finite element was chosen the discretization can be done manually or through a program [10, 11]. In our study the Autodesk Simulation Mechanical 2014 was used to perform the model. Load application considered the maximum force developed by the masseter and pterygoid muscles.The action of the forces developed by the manducatory muscles during mastication produces reaction forces in the temporomandibular joint and on the contact area between occlusal surfaces and the food.Depending on the loading on an element, it will support a certain stress and transmit it to the neighboring elements through the nodes.Although, the muscle activity and

* email: ancavitalariu@yahoo.com

craniofacial morphology affect the occlusal load in actual clinical situation, it is difficult to simulate individual muscle forces to FEA modeling. So, usually vertical or oblique load on the teeth is used as an input load in FEA [12, 13].

The model is exported to Autodesk and after determining the type of static stress, the mesh command is given. Absolute mesh size and absolute mesh dimension are set to 1mm for the purpose of an accurate analysis.In order to get accurate results with the finite element analysis, the loads should be similar to the physiological ones. Stress levels were calculated according to the Von Mises criteria for each node. The geometry of the healthy standard tooth as abutment has been taken from literature. The prepared surfaces of the abutments were: 14.015 mm^2 for premolar and 17.56 mm^2 for the molar.The analyzed model is presented in figure 3 and 4.

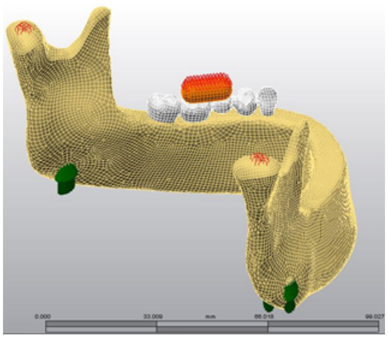

Fig.3. Model of the mandible with the bolus applied on the bridge

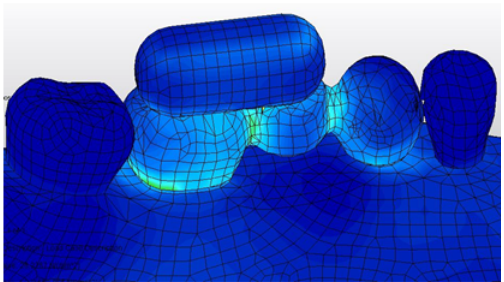

Fig.4. Model of the mandible with the bolus applied on the pontic and abutment 36

A more realistic modeling situation is the one in which a deformable food fragment is applied on the bridge, thus lowering the direct loading exerted on the dental bridge and, implicitly, on the abutment teeth [14]. We surveyed the stress induced by different food consistency with elasticity modulus between 0 MPa and 20000 MPa. Every finite element was ascribed with the biomechanical caracteristics of the component represented by the group (Modulus of elasticity, Poisson's ratio). The differences in elastic modulus are believed to affect the clinical performance of the bridge. All the materials were assumed to be isotropic, homogeneous, and linear elastic [15]. For the bridge we choosed a Ti alloy due to its high biocompatibility and biomechanical behaviour [16]. The properties of materials used into the simulation were adopted from those available in the literature (table 1) [17, 18].

Table 1

MATERIAL PROPERTIES USED IN THIS STUDY

Element	Elasticity modulus [Mpa]	Poisson modulus	Density [kg/m^3]
Bone	13800	0.30	1450
Dentin	17600	0.25	1900
Ti6A14V	110000	0.40	4381.7

Oral rehabilitation is inherently difficult, due to the functional and parafunctional forces within the mouth that result in extremely complex structural responses by the oral tissue. The applied forces for this simulation were F= 400-800 N (on the molar), F= 220-450 N (on the premolar). The force value was increased every 100 MPa for each determination and the maximum value in abutments was recorded.

Loading conditions were vertical and distributed on a15 degrees to the vertical and concentrated. The principal stresses were calculated and compared for the retainers (first molar and first premolar) and pontic (second premolar).

Result and discussions

The results obtained from a FEA on the restored system contain information about the stress distribution of each component of the restoration, instead of only a single value of failure load typical of in vitro results. A correct interpretation of FEA results should be based on the stresses and strength of each component of the system.

For the analyzed items (dental abutments, dental bridge) a similar stress distribution is observed. The values for von Misses stresses were 54,33MPa into the bridge and 15.798 MPa in the supporting teeth (fig.5, 6)

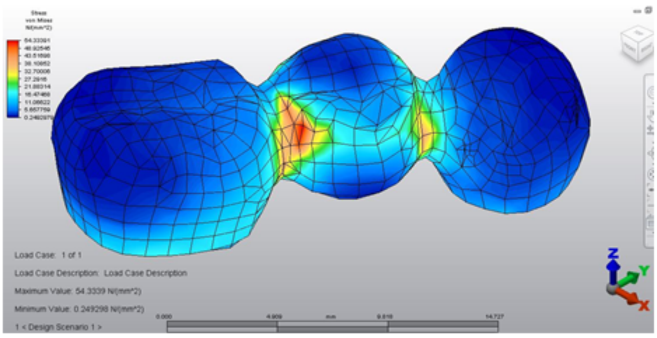

Fig.5. The von Misses Stress distribution and values registered on the bridge

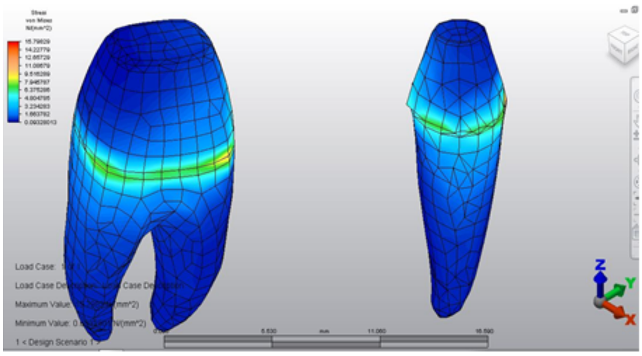

Fig.6. The von Misses Stress distribution and values registered into the abutments

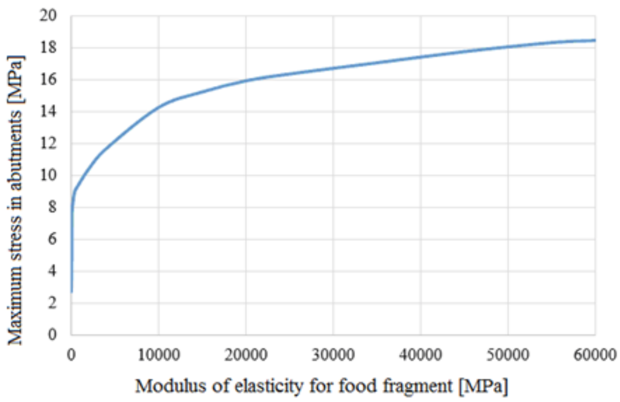

Fig.7.The von Misses stress / food elasticity modulus diagram

Upon to the analysis of the forces acting on the bridge, we noticed certain stress concentrators, especially between the pontic and the abutment 34 when the food fragment was located on the occlusal surface of the mesial abutment, and on the distal surface of the abutment 36 when the food fragment was placed on the occlusal surface of the distal abutment, indicating the highest stress area, therefore the highest breaking risk.

Evolution of the highest tensions into the abutments for variable modulus elasticity of the foodis presented in fig.7.No significant stress was registered until 200MPa modulus of elasticity, when the fist stress value recorded was 8,5Mpa.The maximum von Misses stress value on abutments was 18MPa, for a modulus of elasticity of the food fragment of 60000 MPa.

Conclussions

From a stress standpoint, the distribution on the abutments has the maximum values on the cervical area and on the bridge the stress is increasing distal between the retainer and the pontic related to the modulus of elasticity of the food fragments. The clinical longevity of the supporting teeth depends therefore also on the alimentary behaviour of the patient. The Finite Element Analysis method has advantages over methods that use real patterns. Analyzes are repeatable, there are no ethical considerations, and working hypotheses can be changed or modified sequentially.

References

1.LIN CL, HSU KW, WU CH. J Dent. 2005; 33(9):711-20.

2.TRIVEDI S. J Oral BiolCraniofac Res 2014 Sep-Dec; 4(3): 200–203.

3. DIACONU, D., TATARCIUC, M., VITALARIU, A., STAMATIN, O., FOIA, L., CHECHERITA, L., Mat.Plast.,**51**, no.3, 2014, p.271-274

4.TANCULESCU, O., DOLOCA, A., VIERIU, R.M., MOCANU, F., IFTENI, G., VITALARIU, A., SOLOMON, S., IOANID, N., IOVAN, G., Rev.Chim. (Bucharest), **68**, no.1,2017, p. 94

5.VICECONTI M., ZANNONI C., TESTI D. Comput Methods Programs Biomed. 2007;85:138-151.

6.VAN STADEN R.C., GUAN H., LOO Y.C. Comput Methods Biomech Biomed Engin. 2006;9:257–270

7.DIACONU POPA, D., VITALARIU, A., TATARCIUC, M., MUNTEANU, F., Rev. Chim.(Bucharest), **67**, no. 8, 2016, p.1571

8.GAO J., XU W., DING Z. Comput Methods Programs Biomed. 2006;28:916–924

9.MOLIN MK, ONESTI MP, PETERSSON TB, DERAND TB. Int J Prosthodont. 2007; 20(1):89-91.

10PESQUEIRA AA, GOIATO MC, FILHO HG, MONTEIRO DR, SANTOS DM, HADDAD MF, PELLIZZER EP. J Oral Implantol. 2014 Apr;40(2):217-28

11.VITALARIU A.M, COMANECI R, DUMITRAS C.J ournal Of Optoelectronics And Advanced Materials 2007; 9(11):3419-3422

12.LIU S., LIU Y., XU J., RONG Q., PAN S. J Prosthet Dent. 2014;112:1238–1245

13.MAHONY A.M.O., WILLIAMS J.L., SPENCER P. Clin Oral Implants Res. 2000;9:648–657.

14.FINKELSTEIN E, AMBROSIUS L. AutoCAD 2015 and AutoCAD LT, Bible 1st Edition 2015, Wiley

15.SHINYA A., D. YOKOYAMA, L. V. J. LASSILA, A. SHINYA, AND P. K. VALLITTU. Journal of Adhesive Dentistry, vol. 10, no. 5, pp. 365–371, 2008

16.DIACONU-POPA, D., VITALARIU, A., HOLBAN CIOLOCA, C., AUNGURENCEI, A., LUCA, O., TATARCIUC, M., Rev.Chim. (Bucharest), **68**, no.10, 2017,p.2382

17.KEULEMANS F,SHINYA A, LASSILA LV. J., VALLITTU P K., KLEVERLAAN CJ,. FEILZER A J, DE MOOR RJ.G. The Scientific World Journal 2015, Vol. 2015

18.EICK. D., KOTHA S. P,. CHAPPELOW C. C et al.,. Dental Materials, 2007, vol. 23, no. 8, 1011–1017

Manuscript received: 11.10.2017