Brazilian Dental Science
Ciência Odontológica Brasileira

unesp
UNIVERSIDADE ESTADUAL PAULISTA
"JÚLIO DE MESQUITA FILHO"
Instituto de Ciência e Tecnologia
Campus de São José dos Campos

LITERATURE REVIEW

doi: 10.14295/bds.2014.v17i3.1001

The finite element methods on oral rehabilitation: new trend for endodontically treated teeth

O método dos elementos finitos na reabilitação oral: novas perspectivas para dentes tratados endodonticamente

Juliana Santos OLIVEIRA[1], **Carolina de Andrade Lima CHAVES**[1], **Yara T. Corrêa SILVA-SOUSA**[1], **Érica Alves GOMES**[1]

1 – Area of Endodontic – School of Dentistry – University of Ribeirão Preto – Ribeirão Preto – SP – Brazil.

ABSTRACT

Objective: The aim of this study is to present the applicability of Finite Element Analysis (FEA) in oral rehabilitation, especially in endodontically treated teeth, based on its indication, methods, advantages and limitations through a literature review. **Material and Methods:** The search was conducted on National Library of Medicine's Pub Med, Google search and Science Direct databases including the keywords "finite element analysis", "oral rehabilitation" and "endodontics". **Results:** A total of 412 articles were found, 42 were carefully selected by two subject matter experts for discussion including 21 articles focusing on the applicability of FEA to endodontically treated teeth. **Conclusion:** The FEA is a versatile, low-cost and efficient approach for reliable evaluation of complex structures, as oral rehabilitation treatments.

KEYWORDS

Endodontics; Finite element analysis; Oral rehabilitation.

RESUMO

Objetivo: O objetivo deste estudo é apresentar a aplicabilidade do Método dos Elementos Finitos (MEF) na reabilitação oral, especialmente em dentes tratados endodonticamente, destacando sua indicação, métodos, vantagens e limitações por meio de uma revisão de literatura. **Material e Métodos:** A busca foi realizada na Biblioteca Nacional de Medicina do PubMed, pesquisa Google e Science Direct, incluindo as palavras-chave: "análise de elementos finitos", "reabilitação oral" e "endodontia". **Resultados:** Um total de 412 artigos foram encontrados, sendo que 42 foram cuidadosamente selecionados por dois especialistas no assunto para a discussão e incluídos 21 artigos com foco na aplicabilidade do MEF para dentes tratados endodonticamente. **Conclusão:** O MEF é uma abordagem versátil, de baixo custo e eficaz para avaliação de estruturas complexas, como tratamentos de reabilitação oral.

PALAVRAS-CHAVE

Endodontia; Análise de elementos finitos; Reabilitação oral.

CRITICAL REVIEW

In 1956, the finite element analysis was created based on a numerical method to evaluate complex structures through a virtual model made from a real model. Since then, the method overpassed the Engineering field due to its versatility and efficiency and became an important approach in health science, including dental research [1]. This method was considered innovative since the biological structures present variable geometry and properties as stress distribution that cannot be analyzed in such detail by other methods [2]. According with Marghalani et al. [2] finite element analysis (FEA) is an analytically powerful tool that provides detailed quantitative data at every location within a mathematical model that simulates the mechanical behavior of the system. Thus, FEA has become valuable in the assessment of various systems in dentistry.

This technique represents a computational analysis that creates a virtual model of the physical problem that will be solved by finite element software through mathematical equations. The virtual model created with multiple structures with specific properties (e.g. elasticity modulus and Poisson’s coefficient) is divided into minor pieces named finite element mesh. As a consequence, the researcher can operate the software to simulate the numerical performance of multiple materials, techniques and designs based on the stress distribution under loading [3]. To create the virtual teeth, there were two previous methods of manual description: 1) use of standard anatomical data in the literature, and 2) use of cross-sectional histological images of teeth [4]. Nowadays, the virtual model can be reproduced using computed tomography or microtomography (micro-CT) associated or not with CAD software, this technique save time, and provide a relevant results [5].

Endodontically treated teeth require proper rehabilitative treatment [3]. Due to the numerous treatments available, clinical trials are not feasible to test all these materials, which could result in ethical problems. Root fractures in the dental treatment remains high, it is still unclear how these root fractures occur after the endodontic treatment procedure and whether different root canal enlargement designs would affect the pattern of fracture risks along the root, or the combination of posts with various lengths, surface conditions and diameters [6]. Thus, it is crucial to use a valuation method that permits a satisfactory simulation of the oral environment.

Mechanical studies in dentistry have been used to determine the endodontic teeth fracture resistance. However, studies that analyzing fracture resistance is actually giving the results in Kgf or N, which is the fracture load, that is representative of the risk of fracture, and not the strength itself, calculated in MPa. For the strength, the bonded area is needed and therefore too difficult to be considered unless the studies including slices of the root are used. Other point of view is that they do present limitations when obtaining information on the internal behavior of the tooth-restoration complex [7] and cannot evaluate fatigue in other dental tissues according to masticator forces [6]. FEA is the best way to obtain the stress distribution on the canal wall, as well in complex hard tissue structures, such as bone and tooth. Since, according with Cheng et al. [6] analyzing whole tooth structures and surrounding tissues are essential to estimate the strength and fracture risk after endodontic treatments. So, the use of FEA is very powerful as using this method, not only the load, but the stress can be reported.

Thus, the aim of this study is to describe the applicability of the finite element method in Dentistry focusing on the rehabilitation of endodontically treated teeth. The literature review will describe the steps, advantages and limitations of this methodology.

The search was conducted in February of 2014 on National Library of Medicine’s Pub Med, Google search and Science Direct databases including the keywords “finite element analysis”, “oral rehabilitation” and “endodontics”. The criteria for selection were: (1) Published in the last 15 years; (2) published in high impact journals in English; and (3) have relevant information for this study.

A total of 412 articles were found and 42 were carefully selected by two subject matter experts for discussion, including 21 articles focusing on the applicability of FEA to endodontically treated teeth.

APPLICABILITY OF FINITE ELEMENT METHOD AND ORAL REHABILITATION

Considering that endodontically treated teeth exhibit dehydration and alteration of physical and mechanical properties, those structures are more prone to fracture mainly when intracanal posts and crowns are used [8].

Among all materials available in the market for oral rehabilitation, further analysis is required for better selection of the properties of the systems to provide appropriate results for each clinical scenario [9]. In addition, it is also difficult to evaluate the dental restoration since functional and parafunctional forces generate complex structural performances [10]. In this sense, the finite element method has been suggested as an important tool to evaluate the effect of several variables and predict the clinical performance of the treatment planning [11,12].

Current studies reported the importance of the finite element method for analysis of the stress distribution in teeth restored with different prefabricated posts [2,10-14], crowns [2], techniques for cementation of intracanal posts [2,10,14] and tooth length [12,13]. Furthermore, this method is also efficient to study the resistance and risk to fracture of endodontically treated teeth [11,15] and evaluate the stress distribution at post-resin-adhesive interface simulating a push-out-test [16]. The maxillary central incisor and canine is the most common tooth used in FEA researches [6,10,13] since it is frequently restored in clinical practice and the single loading point is easily reproduced to simulate mastication.

For analysis of restoration of endodontically treated teeth, the model represents the tooth and all the structures that can influence the response to the new material or technique to be tested: enamel, dentine, gutta percha, intracanal post, cement, periodontal ligament and bone tissue. Since the goal of the computational model is to represent the real model, it is of paramount importance to define all the structures that influence the final result of the material evaluated.

STEPS OF FEA

1- Virtual model generation

Initially, the researcher has to design the groups, materials and all structures that will be evaluated. Then, a 2D or 3D virtual model will be created to represent the real model using CAD software. Although some studies reported that 2D models are reliable and easily [10,17] it has been suggested that 3D models [3,6,9,13-16,18-22] provide more accurate information about stress distribution during the analysis, which generates results similar to the clinical reality.

As the oral cavity is very complex in nature, it is very difficult to represent this structure with high accuracy by means FE models. For this reason, current studies have been demonstrated new trends in the construction of 3D models. Actually, the virtual model can be reproduced using computed tomography or microtomography (micro-CT) associated or not with CAD software. This technology improves the model design, allowing acquisition of high resolution images and generates models with accurate geometry; which provides reproduction of anatomical details and decreases the errors during analysis [3,16,20-22] (Figure 1).

2 - Pre-processing

2.1 - Finite element mesh generation

After modeling, the virtual model is divided into several elements that are connected by nodes, generating the finite element mesh using specific software for finite element analysis. Each element represents a mathematical equation to be solved by the software. Additionally, those divisions allow detailed evaluation of the entire system since the performance of each element can be observed individually [22]. The most common geometries of the elements described in the literature are quadrilateral and tetrahedral and the size of each element depends on the accuracy of each structure to be evaluated. The more refined the elements mesh (i.e. the greater the number of elements in a given structure), the greater the accuracy and reliability of the results (Figure 2). The quality of mesh is an important step in model simulation because represent the reliability of the results. Pessoa et al. [24] recommended a mesh refinement based on the analysis convergence in 6%. A convergence analysis is an effective way to identify the balancing accuracy of the FE result and computing resources. In the convergence test, meshes with successively larger element numbers are used until further refinement of the model does not alter the maximum stress values [25].

2.2 - Definition of the materials' properties

Usually, the materials of oral rehabilitation are assumed as isotropic (i.e. material with the same property in the entire structure regardless the direction), elastic (i.e. material that returns to its original condition after deformation) and homogenous (i.e. material without empty spaces). So, the values of elasticity modulus and Poisson's coefficient are transferred to the finite element software [3,13]. These characteristics are related to rigidity and deformability of the object.

In this step, many authors have used values from the literature, in order to facilitate the search. In all articles analyzed, only one [26] proposes to seek the characteristics of each material evaluated citing its trademark. As is known the origin of the materials available in the literature (make and year) and how many of the references are old, there is discrepancy in the modulus of elasticity and Poisson's coefficient on the literature (Table 1).

Table 1 - Comparison of diferent material properties from the literature

Dentin		Post		Cement		Reference
Elastic properties (GPa)	Poisson's ratio (v)	Elastic properties (GPa)	Poisson's ratio (v)	Elastic properties (GPa)	Poisson's ratio (v)	
18.6	0.31	100 (gold) 210 (cercon)	0.31(gold) 0.30 (cercon)	2.6	0.36	[2]
18.6	0.32	130 (fiber carbon)	0.30	5.1	0.30	[3]
15.0	0.31	50	0.30	not applicable	not applicable	[4]
18.3	0.30	not applicable	not applicable	not applicable	not applicable	[5]
12.0	not applicable	not applicable	not applicable	not applicable	not applicable	[6]
18.0	0.31	Ex: 37, Ey: 9.5, Ez: 9.5 (glass fiber post)	vxy: 0.34, vyz: 0.27, vxz: 0.34 (glass fiber post)	not applicable	not applicable	[7]
18.6	0.30	116 (titanium) 200 (zirconia) 380 (alumina)	0.33 (titanium) 0.33 (zirconia) 0.25 (alumina)	not applicable	not applicable	[10]
18.6	0.31	112(titanium) Ex: 37, Ey: 9.5, Ez: 9.5 (glass fiber post)	0.33 (titanium) vxy: 0.27, vyz: 0.27, vxz: 0.34 (glass fiber post)	5.1	0.27	[12]
18.6	0.31	188 (cast NiCr) Ex: 37, Ey: 9.5, Ez: 9.5 (glass fiber post)	0.33 (cast NiCr) vxy: 0.34, vyz: 0.27, vxz:0.27 (glass fiber post)	8.3	0.35	[13]
18.6	0.31	120 (titanium)	0.30 (titanium)	not applicable	not applicable	[14]
18.6	0.31	200 (metal) Ex: 37, Ey: 9.5, Ez: 9.5 (glass fiber post)	0.30 (metal) vxy: 0.27, vyz: 3.4, vxz: 0.27 (glass fiber post)	7.0	0.27	[15]
18.3	0.3	Ex: 5.1, Ey: 5.1, Ez: 18 (fiber post)	vxy: 0.11, vyz: 0.11, vxz: 0.35 (fiber post)	9.5	0.3	[16]
18.6	0.32	89.5 (gold alloy) 200 (NiCr alloy) Ex: 37, Ey: 9.5, Ez: 9.5 (glass fiber post)	0.33 (gold alloy) 0.33 (NiCr alloy) vxy: 3.1, vyz: 3.5, vxz: 3.1 (glass fiber post)	18.6	0.28	[17]
18.6	0.31	93 (ILOR 56 gold alloy - post-and-core) E_L:125, E_T: 8.5 (carbon fibre post) E_L: 40, E_T: 11 (E-glass fibre post)	0.33(ILOR 56 gold alloy - post-and-core) V_{TL}: 0.25, V_{TL}: 0.017, V_{TT}: 0.32 (carbon fibre post) V_{TL} :0.26, V_{TL}: 0.07, V_{TT}: 0.32 (E-glass fibre post)	0.28 (for carbon fibre post) 0.26 (for E-glass fibre post)	0.33 (for carbon fibre post) 0.33 (for E-glass fibre post)	[18]
15.0	0.31	not applicable	not applicable	not applicable	not applicable	[21]
18.3	0.30	not applicable	not applicable	not applicable	not applicable	[22]
20.0	0.31	not applicable	not applicable	not applicable	not applicable	[23]
18.6	0.32	210 (steel post) Ex:118, Ey: 7.2, Ez: 7.2 (carbon post) Ex:37, Ey: 9.5, Ez: 9.5 (glass post)	0.30 (steel post) v_{XY}: 0.27, v_{ZY}: 0.34, v_{XZ}: 0.27 (carbono post) v_{XY}: 0.27, v_{ZY}: 0.34, v_{XZ}: 0.27 (glass post)	7.0 (C&B, Bisco, USA) 18.6 (Panavia, Kuraray, Japan)	0.28 (C&B, Bisco, USA) 0.28 (Panavia, Kuraray, Japan)	[26]

2.3 - Definition of loading and displacement boundary conditions

It is important to determine correctly the region that will be submitted to external loading, pressure, thermal cycling or other factors. The load can be applied in points that simulate the masticatory forces or adhesion tests to assess the stress distribution in the different materials used for dental restoration (e.g. posts and crowns) or as a pressure in push-out tests. Furthermore, the displacement boundary conditions have to be determined in the software to characterize the area under loading [3,23] and must be carefully treated with the use of computational modeling techniques (Figure 3). The boundary conditions are essential for the result accuracy [5] and influence the final result. Displacement conditions, for example contact or non-contact interfaces, or including or not the whole dental structure such as: with or without the bone and with or without periodontal ligament, as well as include or not the friction coefficient in the post at the dentin surface could modify the final results [2].

3 - Post-Processing

After determination of the previous steps, the software will solve all the equations with millions of variables to generate the results. Post-processing software allows evaluation of the stress by qualitative and quantitative analyses using graphs and numbers [23]. The qualitative analysis is based on color scales presented in graphs where hot colors (red) represent higher stress while cold colors (blue) indicate lower stress. For quantitative analysis, several analysis criteria are applied according to each material type. The von Mises criterion is a criterion of failures that associates tensile, compression and shearing stress to evaluate ductile materials based on the occurrence of damage [13,14,18,25] (Figure 4). According to Dejak and Mlotkowski [13], the criterion of maximum principal stress (σ_{max}) can be used to evaluate non-ductile materials (e.g. dentine) and predict failures at interfaces. The values of the structures can be also analyzed using the minimum principal stress (σ_{min}) and shearing stress (σ_{shear}).

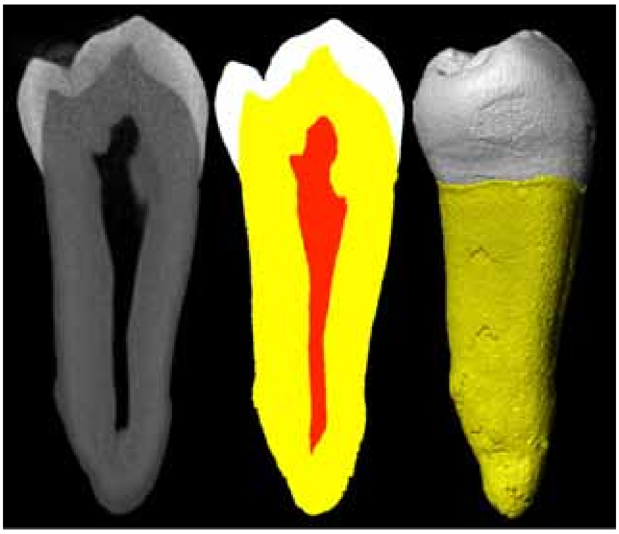

Figure 1 - Microtomograpy of a premolar; B – Reconstruction of a two-dimensional microtomographic image for fabrication of the virtual model; C – 3D virtual model based on the detailed microtomographic image.

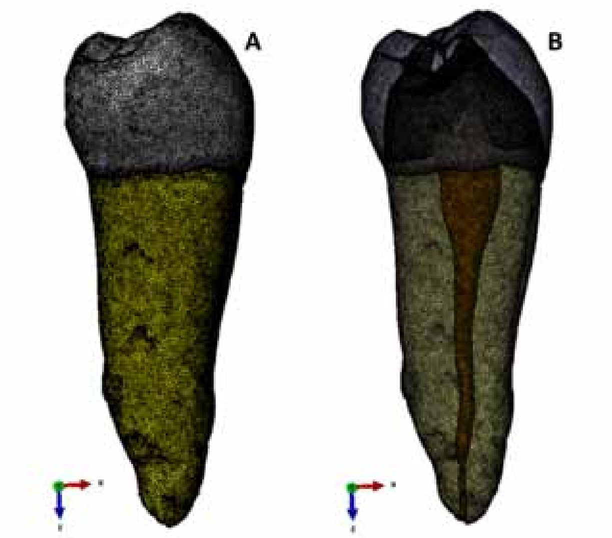

Figure 2 - Finite element mesh in tetrahedral elements with a total of 185,675 nodes and 950,703 elements. A – External visualization of the finite element mesh; B – Internal visualization of the finite element mesh.

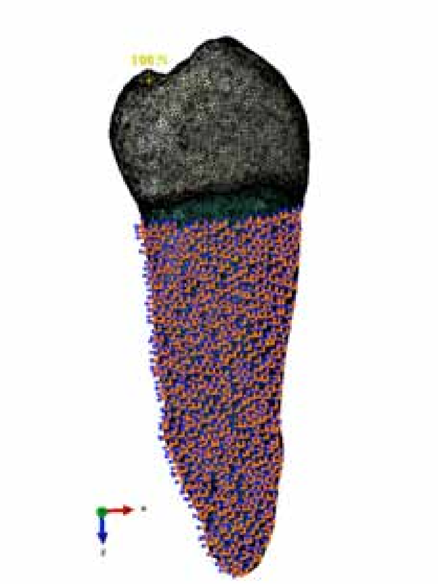

Figure 3 - Empirical definition of the boundary and loading conditions. The yellow arrow corresponds to the loading site and value while the blue and orange traingles show the boundary conditions of the model.

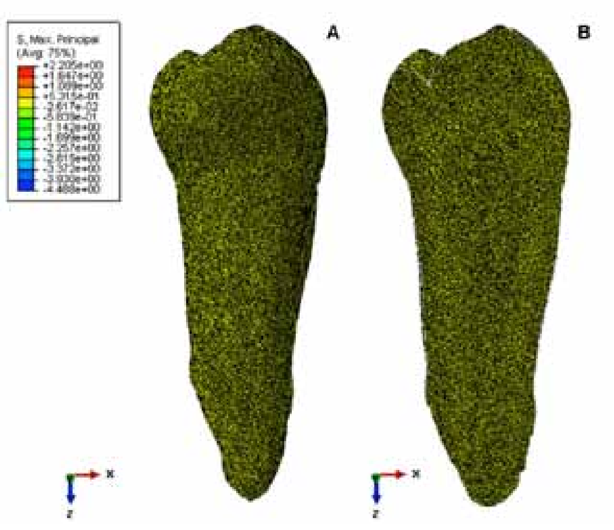

Figure 4 - Maximum principal stress (MPa) for premolar according to the simulation. A – external view of the maximum principal stress distribution (MPa); B – internal view of the maximum principal stress distribution (MPa).

ADVANTAGES

The finite element analysis is a relatively fast method, low-cost, nondestructive, and efficient approach to evaluate the stress distribution in complex structures [19,23]. This method provides detailed and accurate data based on mathematical analyses [2]. In addition, since it predicts the failure in some areas, it provides information about the regions exhibiting higher risk to fracture.

FEA studies are considered reliable when compared to laboratory, but small flaws in the methodology can compromise the entire result [22]. Therefore performed satisfactorily, FEA becomes a powerful evaluation tool, reducing the unnecessary use of samples for research.

Actually, there is the possibility of development of anatomical models for teeth using data retrieved from computed microtomography (micro-CT). The micro-CT models allows the use of only one human tooth, differently of in vitro studies that require a significantly larger sample size to find reliable results. According to Tajima et al. [22] the combination of scanned the teeth with micro-CT and used on FEA might result in a considerable reduction in the time consumed in creation of FE models, when compared with traditional methods of modeling from CT data and could be used as a valid model to estimate the actual strains with acceptable accuracy. Others studies have proved the new trend and confirm that the three-dimensional FEA based on micro-CT are valid and widely recognized in dental research [19,22]. Thus, using this method it is ease, speed and sophisticated the simulation tests with numerous rehabilitation materials in the same virtual model meaning save time, low cost, reliable and relevant results [15].

LIMITATIONS

Finite element analysis is an excellent tool to predict long-term failures in some regions. Using appropriate software, it is also a low cost and less time consuming approach compared to in vitro studies since it does not require several samples. On the other hand, the accuracy on modeling of the structures, mechanical properties of all materials, loading and boundary conditions influence the results.

Considering that the operator determines the design and properties of all materials as well as the direction and force of loading, errors can be induced in the experiment. Although static loading is usually reproduced, it does not simulate the real masticatory forces. Masticatory forces may vary in each individual according to gender, age, occlusion and oral health. In addition, the assumption of the materials as isotropic, elastic and homogeneous to the behavior for tooth structures ignoring their in homogeneities does not represent the real behavior [12]. It is necessary to highlight that biological tissue including enamel, dentin, periodontal ligament and bone show heterogeneous and anisotropic characteristic. According to O'Mahony et al. [27] the use of isotropic properties instead of anisotropic properties for bone tissue may affect the overall results of stress distribution. Additionally, the stress generated in bone under compression is different than those under tension [28]. Thus, this is a critical step considering that anatomical structures should be represented [19,26].

Most of the studies have assumed the materials as homogeneous and linearly elastic based on Young's modulus and Poisson's ratio [28]. However, some materials, such as glass fiber post, should be assumed as orthotropic since the mechanical properties vary along the fibers direction (x) and normal directions (y and z). Thus, the mechanical characteristics of the materials are represented by the elasticity modulus in the 3 directions (Ex, Ey, Ez) as well as Poisson's ratio (vxz, vxz, vyz) and shear modulus (Gxz, Gxz, Gyz) in the orthogonal planes (xy, xz and yz). In this sense, this assumption must be carefully analyzed to avoid mistakes in finite element analysis.

It should be highlighted that the values of materials properties, especially Young's modulus and Poisson's ratio, vary in the literature (Table 1) and can lead to wrong comparisons between results from different studies. Thus, any incorrect or inaccurate value can lead to different results and errors in research [29]. Ideally, the materials characteristics should be obtained and the brand of each material should be described in the FEA. Usually, commercial products update and modify its composition to achieve better clinical success. The lack of knowledge about the material origin and its Young's modulus and Poisson's ratio available in the literature may lead to wrong results. In this sense, Lanza et al. [26] mentioned the material used in the study, making the results more reliable. Most of the finite element analyses do not perform those tests in order to reduce cost and time.

Furthermore, when analyzing the risk of fractures in teeth, it is important to know that while the FEM can identify regions of stress concentrations that represent a high probability of failure, the method cannot accurately predict the fracture pattern and the progression of materials [26].

The method used for stress evaluation also influences the results. In the present study, von Mises stress was chosen for analysis. According to Eraslan et al. (17), von Mises stress (σvM) represents the association of tensile, compressive and yielding stress. The von Mises stress is usually used to indicate the occurrence of damage. Considering that the compressive strength of dentine is higher than tensile strength, the von Mises stress can be compared to the tensile strength of dentine to evaluate the risk to fracture.

Although the finite element analysis has been efficient to compare stress distribution between different groups, it does not represent the real scenario perfectly. According with Al-Sukhum et al. [19] the majority of experimental stress methods, its main disadvantage is that it is not appropriate for analyzing strain under in vivo conditions. However, the comparison between groups allows observation of the superiority of one material over the other. Thus, this methodology is essential to comparison groups, and has to be used by researchers with knowledge about the technique to obtain reliable results [28]. Based on these reports, it can be concluded that the finite element analysis is an efficient method to evaluate the stress distribution in oral rehabilitation, especially when involving endodontic treatment, since it has great recognized in dental research.

REFERENCES

1. Chandrupatla TR, Belegundu AD. Fundamental concepts: introduction to finite elements in engineering. Prentice hall of India private limited. New Delhi: Pearson Education; 2012.
2. Marghalani TY, Hamed MT, Awad MA, Naguib GH, Elragi AF. Three-dimension finite element analysis of custom-made ceramic dowel made using CAD/CAM technology. J Prosthodont. 2012;21(6):440-50.
3. Ausiello P, Franciosa P, Martorelli M, Watts DC. Mechanical behavior of post-restored upper canine teeth: a 3D FE analysis. Dent Mater. 2011;27(12):1285-94.
4. Li W, Swain MV, Li Q, Steven GP. Towards automated 3D finite element modeling of direct fiber reinforced composite dental bridge. J Biomed Mater Res B Appl Biomater. 2005;74(1):520-8.
5. Tajima K, Chen KK, Takahashi N, Noda N, Nagamatsu Y, Kakigawa H. Three-dimensional finite element modeling from CT images of tooth and its validation. Dent Mater J. 2009;28(2): 219–26.
6. Chen G, Fan W, Mishra S, El-Atern A, Schuetz MA, Xiao Y. Tooth fracture risk analysis based on a new finite element dental structure models using micro-CT data. Comput Biol Med. 2012;42(10):957–63.
7. Santos-Filho PCF, Verissimo C, Soares PV, Saltarelo RC, Soares CJ, Martins LRM. Influence of ferrule, post system, and lenght on biomechanical behavior of endodontically treated anterior teeth. J Endod. 2014;40(1):119–23.
8. Ree M, Schwartz RS. The endo-restorative interface: current concepts. Dent Clin North Am. 2010;54(2):345-74.
9. Cheung WA. Review of the management of endodontically treated teeth. Post, core and the final restoration. J Am Dent Assoc. 2005;136(5):611–9.
10. Abu Kasim NH, Madfa AA, Hamdi M, Rahbari GR. 3D-FE analysis of functionally graded structured dental posts. Dent Mater J. 2011 [Epud ahead of print].
11. Rosentritt M, Behr M, van der Zel JM, Feilzer AJ. Approach for evaluating the influence of laboratory simulation. Dent Mater. 2009;25(3):348–52.
12. Silva NR, Castro CG, Santos-Filho PC, Silva GR, Campos RE, Soares PV et al. Influence of different post design and composition on stress distribution in maxillary central incisor: Finite element analysis. Indian J Dent Res. 2009;20(2):153-8.
13. Dejak B, Młotkowski A. Finite element analysis of strength and adhesion of cast posts compared to glass fiber-reinforced composite resin posts in anterior teeth. J Prosthet Dent. 2011;105(2):115-26.
14. Garhnayak L, Parkash H, Sehgal DK, Jain V, Garhnayak M. A comparative study of the stress distribution in different endodontic post-retained teeth with and without ferrule design: a finite element analysis. ISRN Dent. 2011; 2011:102329. doi: 10.5402/2011/102329. Epub 2011 Jun 14.
15. Romeed SA, Dunne SM. Stress analysis of different post-luting systems: a three-dimensional finite element analysis. Aust Dent J. 2013;58(1):82-8.
16. Jongsma LA, Ir Nde J, Kleverlaan CJ, Feilzer AJ. Reduced contraction stress formation obtained by a two-step cementation procedure for fiber posts. Dent Mater. 2011;27(7):670-6.
17. Watanabe MU, Anchieta RB, Rocha EP, Kina S, Almeida EO, Freitas AC Jr et al. Influence of crown ferrule heights and dowel material selection on the mechanical behavior of root-filled teeth: a finite element analysis. J Prosthodont. 2012;21(4):304-11.
18. Pegoretti A, Fambri L, Zappini G, Bianchetti M. Finite element analysis of a glass fibre reinforced composite endodontic post. Biomaterials. 2002;23(13):2667-82.
19. Al-Sukhun J, Kelleway J, Helenius M. Development of a three dimensional finite element model of a human mandible containing endosseous dental implants. I. Mathematical validation and experimental verification. J Biomed Mater Res A. 2007;80(1):234–46.
20. Magne P, Oganesyan T. Premolar cuspal flexure as a function of restorative material and occlusal contact location. Quintessence Int. 2009;40(5):363–70.
21. Rodrigues FP, Li J, Silikas N, Ballester RY, Watts DC. Sequential software processing of micro-XCT dental-images for 3D-FE analysis. Dent Mater. 2009;25(6):47–55.
22. Tajima K, Chen KK, Takahashi N, Noda N, Nagamatsu Y, Kakigawa H. Three-dimensional finite element modeling from CT images of tooth and its validation. Dent Mater J. 2009;28(2):219–26.
23. Ricks-Williamson LJ, Fotos PG, Goel VK, Spivey JD, Rivera EM, Khera SC. A three- dimensional finite-element stress analysis of an endodontically prepared maxillary central incisor. J Endod. 1995;21(7):362-7.
24. Pessoa RS, Muraru L, Júnior EM, Vaz LG, Sloten JV, Duyck J, Jaecques SV. Influence of implant connection type on the biomechanical environment of immediately placed implants - CT-based nonlinear, three-dimensional finite element analysis. Clin Implant Dent Relat Res. 2010;12(3):219-34.
25. Bahrami B, Shahrbaf S, Mirzakouchaki B, Ghalichi F, Ashtiani M, Martin N. Effect of surface treatment on stress distribution in immediately loaded dental implants--a 3D finite element analysis. Dent Mater. 2014 Apr;30(4):89-97
26. Lanza A, Aversa R, Rengo S. 3D FEA of cemented steel, glass and carbon posts in a maxillary incisor. Dent Mat. 2005;21(8):709-15.
27. O'Mahony AM, Williams JL, Spencer P. Anisotropic elasticity of cortical and cancellous bone in the posterior mandible increases peri-implant stress and strain under oblique loading. Clin Oral Implants Res. 2001;12(6):648-57.
28. Geng JP, Tan KB, Liu GR. Application of finite element analysis in implant dentistry: a review of the literature. J Prosthet Dent. 2001;85(6):585-98
29. Aggarwal S, Garg V. Finite element analysis of stress concentration in three popular brands of fiber posts systems used for maxillary central incisor teeth. J Conserv Dent. 2011;14(3):293-6.

Érica Alves Gomes
(Corresponding address)
Curso de Odontologia - Universidade de Ribeirão Preto
Av. Costábile Romano, 2.200 - CEP 14096-900
Ribeirão Preto/SP, Brazil
e-mail: ericaagomes@yahoo.com.br

Date submitted: 2014 May 12
Accept submission: 2014 Aug 11