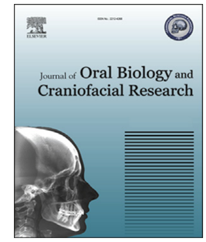
journal of oral biology and craniofacial research 4 (2014) 200–203

Available online at www.sciencedirect.com

ScienceDirect

journal homepage: www.elsevier.com/locate/jobcr



Review Article

Finite element analysis: A boon to dentistry

Shilpa Trivedi

Research Scholar, Department of Oral and Maxillofacial Surgery, FODS, King George's Medical University, Lucknow, Uttar Pradesh, India

ARTICLE INFO

Article history:
Received 15 November 2014
Accepted 20 November 2014
Available online 4 December 2014

Keywords:
Finite element analysis
Implants
Trauma

ABSTRACT

The finite element analysis (FEA) is an upcoming and significant research tool for biomechanical analyses in biological research. It is an ultimate method for modeling complex structures and analyzing their mechanical properties. In Implantology, FEA has been used to study the stress patterns in various implant components and also in the peri-implant bone. It is also useful for studying the biomechanical properties of implants as well as for predicting the success of implants in clinical condition. FEA of simulated traumatic loads can be used to understand the biomechanics of fracture. FEA has various advantages compared with studies on real models. The experiments are repeatable, there are no ethical considerations and the study designs may be modified and changed as per the requirement. There are certain limitations of FEA too. It is a computerized in vitro study in which clinical condition may not be completely replicated. So, further FEA research should be supplemented with clinical evaluation.

1. Introduction

The finite element analysis (FEA) is an upcoming and significant research tool for biomechanical analyses in biological research. It is an ultimate method for modeling complex structures and analyzing their mechanical properties. FEA has now become widely accepted as a non-invasive and excellent tool for studying the biomechanics and the influence of mechanical forces on the biological systems. It enables the visualization of superimposed structures, and the stipulation of the material properties of anatomic craniofacial structures.[1] It also allows to establish the location, magnitude, and direction of an applied force, as it may also assign stress points that can be theoretically measured.[2] Further, as it does not affect the physical properties of the analyzed materials it is easily repeatable.[2,3]

The finite element method (FEM) is basically a numerical method of analyzing stresses and deformations in the structures of any given geometry. The structure is discretized into the so called 'finite elements' connected through nodes. The type, arrangement and total number of elements impact the accuracy of the results.[4] 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. Various engineering software packages are available to model and simulate the structure of interest may be implants or jawbone.

Previously, when FEA was used in dentistry, various simplified assumptions were made regarding modeling geometry, load, boundaries and material properties.[5] Such assumptions inevitably affected the analytical results. In the human body, there are individual variations with respect to

E-mail address: shilpa.knp@gmail.com.
http://dx.doi.org/10.1016/j.jobcr.2014.11.008

bone quality, quantity and shape which have an important impact on the prognosis of the implant or regenerative treatment. Recently with the advances of digital imaging systems (CT and MRI), it has become possible to extrapolate the individual specific data of bone geometry and property to an FEA model.[6,7] CT and MRI image bone and implant structure at microlevel in three dimensions. These patient specific "biological data based FEA" are peculiar to that patient as bone morphology and quality vary among individuals. Thus, very accurate anatomical models can be created which in turn provide reliable results.

For FEA modeling, a series of patient CT image data is binarized to build FEA model geometry consisting of both cortical and cancellous bone. Then apparent density, porosity or apparent ash density is appraised using different correlations to model the heterogenous distribution of mechanical properties. Most models consider isotropic behavior, since it is not possible to quantify the whole anisotropic structure of a bone, organ with current techniques.[8] The load is applied either to the implant or to the bone as required. Although, the muscle activity and craniofacial morphology affect the occlusal load in actual clinical situation, it is presently difficult to simulate individual muscle forces to FEA modeling. So, usually vertical or oblique load on the teeth or implant is used as an input load in FEA.[9,10]

2. FEA in implants

In Implantology, FEA has been used to study the stress patterns in various implant components and also in the peri-implant bone. Several FEA studies postulated that the stress pattern on peri-implant bone is affected by the implant number, dia-meter, length, thread profile, material properties of implant components and also by the quality and quantity of surrounding bone.[11,12]

In FEA studies to assess mechanical stress in the peri-implant bone, stresses of various kinds which are used include von Mises stress, the maximum, the minimum principal stress and the maximum shear stress. The von Mises stress is the most frequently and mainly used scalar-valued stress invariant to appraise yielding/failure behavior of various materials. The maximum principal stress is appropriate for the observation of tensile stress and the minimum gives an idea about the compressive stress. The use of principle stress is considered good for such studies as the bone has both ductile and brittle properties.[13]

It has been suggested that mechanical stress has an eminent role in maintaining home-ostasis of the bone[14] and in animal models, occlusal overload was predicted to be a risk factor for peri-implant bone loss.[15]

In an FEA study, Demenko et al[16] suggested to select the implant size, giving importance to its load bearing capacity. The long term results of mandibular implant supported overdentures suggest that loss of osseointegration without signs of infection was more common than peri-implantitis.[17] These investigations highlight the role of biomechanical condition on the implant–bone interface for long term success of the implants.

Finite element methods substantiate that the higher crown to implant ratio increase the risk of mechanical failure. A recent study suggested that oblique loading induced higher stress to the fixation screw, chiefly when the crown: implant ratio was 1.5:1.[18] The results are in consensus with Urdaneta et al[19] who demonstrated a significant correlation between screw loosening, fracture of prosthetic abutments, and crown height.

The success of FE modeling depends on the accuracy in simulating the geometry and surface structure of the implant, the material characteristics of the implant and jawbone, the loading and support conditions as well as the biomechanical implant–jawbone interface. The assumptions made during modeling and limitations of the software invariably result in inaccuracies. FEA has been used to study the biomechanical properties of implants as well as for predicting the success of implants in clinical condition. The principal difficulty of simulating living bone tissue and its response to applied mechanical forces is also solved to some extent with the use of advanced imaging techniques.

FEA gives an in depth idea about the stress patterns in the implant and more importantly in the peri-implant bone. This knowledge about the stress patterns and distribution will lead to improvement in implant designs and placement techniques.

3. FEA for maxillary obturator

Biomechanics of implant supported maxillary obturator prosthesis can also be studied in simulated maxillary resection models by FEA. De Saussa and Mattos[20] concluded that the maximum dislodgment of the obturator prosthesis increased with a decrease in the area of bone support, the number of implants, and the number of clips. Also the gingival mucosa, cancellous and cortical bone was subjected to compressive stress, which increased with the decrease in the area of bone support, implant number and the number of clips in the bar retention system. So, FEA was found to be important for predicting the success of implant supported prosthetic rehabilitation of patients after maxillectomy.

4. FEA for post and core restorations

FEA numerical simulation has also been used to improve the mechanical stability and long-term success of post-and-core restorations. Liu et al[9] suggested that for teeth with limited coronal dentin at the loading location, as maxillary premolars with large-scale tissue loss it was crucial to lower the oblique forces by reducing the lateral occlusal contact area and by preventing contact on the top of the facial cusp, thus protecting the remaining dentin from fracture.

5. FEA and periodontal ligament

Periodontal ligament (PDL) is a soft highly specialized connective tissue which is located between the tooth root and alveolar bone.[21] Its primary function is tooth support and it is the most crucial component of periodontium in terms of deformation. Studies investigating dental biomechanics under masticatory and traumatic loads have included PDL in FEA model due to its effect on tooth mobility as well as stress and distribution on periodontium.[22,23] All of these studies consider PDL as a 3D body enclosing a tiny and relatively thin volume lying between the tooth and the alveolar bone.[22,23] Inclusion of PDL as a 3D model results in complexity of the model even though PDL is not the main concern in most of the studies. However, ignoring the ligament oversimplifies the models and results in inaccurate stress and strain distributions on periodontium along with improbable tooth movements.[24,25]

Recently, Tuna et al[26] simulated PDL as a contact model between the tooth and alveolar bone instead of a solid-meshed FE model with poor geometric morphology or very dense mesh. It was proposed that this model saves time and pre/post processing workforce, increases the accuracy and adds to the smoothness of interface stress distributions as well.

6. Role of FEA in trauma and fractures

Along with biological simulations, FEA can be used to estimate biomechanical responses of defined regions of tissues and organs to specific conditions. FEA of simulated traumatic loads can be used to understand the biomechanics of fracture. For investigating the biomechanics of facial trauma it is often challenging to have a model which is practically and ethically acceptable and also gives valid and reliable results. In the past, cadaveric studies were used, but in the present times ethical considerations don't allow such studies. Further, post mortem alterations and the age of death of cadaver usually don't match with that of a typical facial trauma patient. Most studies speculate that the average age of patients with maxillofacial trauma is 20–30 years.[27,28]

Considering these facts, FEA generates valid models for biomechanical analysis. Huempfner-Hierl et al[29] studied naso–orbitoethmoid fracture by this method. Finite element analysis showed a pattern of von Mises stresses beyond the yield criterion of bone that corresponded with fractures commonly seen clinically. It was concluded that finite element models can be used to simulate injuries to the human skull, and provide information about the pathogenesis of different types of fracture.

The stress distribution in the human mandible at three different life stages was estimated by Bujtár et al[30] by FEA. The authors observed highest stress levels in the mandibular neck in an edentulous mandible of a 67 year old patient which was attributed to bone stiffness.

The stress distributions from traumatic loads applied on the symphyseal, parasymphyseal and mandibular body regions in elderly edentulous mandible were analyzed using FEA in another study.[31] It was found that traumatic load on the symphyseal region generated higher stress levels than the traumatic load on the parasymphyseal region. These results indicated that stress levels depend on the site (symphyseal, parasymphyseal, and mandibular body regions) and intensity of the load. This type of analysis can also be a very useful instrument for the improvement of surgical techniques and the development of new biomaterials.

7. Conclusion

FEA has various advantages compared with studies on real models. The experiments are repeatable, there are no ethical considerations and the study designs may be modified and changed as per the requirement.[29] There are certain limitations of FEA too. It is a computerized in vitro study in which clinical condition may not be completely replicated. Stress analysis is usually conducted under static loading, and the mechanical properties of materials are set as isotropic and linearly elastic, although it is not so in reality. So, the results may only be acknowledged qualitatively. Szwedowski et al[32] compared FEA with real models and found that the model accorded well with strain gauge measurements. Keeping in mind these limitations, further FEA research should be supplemented with clinical evaluation.

Conflicts of interest

The author has none to declare.

REFERENCES

1. Sun J, Jiao T, Tie Y, Wang DM. Three dimensional finite element analysis of the application of attachment for obturator framework in unilateral maxillary defect. *J Oral Rehabil*. 2008;35:695–699.
2. Gao J, Xu W, Ding Z. 3D finite element mesh generation of complicated tooth model based on CT slices. *Comput Methods Programs Biomed*. 2006;28:916–924.
3. Viceconti M, Zannoni C, Testi D, et al. The multimod application framework: a rapid application development tool for computer aided medicine. *Comput Methods Programs Biomed*. 2007;85:138–151.
4. Van Staden RC, Guan H, Loo YC. Application of the finite element method in dental implant research. *Comput Methods Biomech Biomed Engin*. 2006;9:257–270.
5. Geng JP, Tan KB, Liu GR. Application of finite element analysis in implant dentistry: a review of the literature. *J Prosthet Dent*. 2001;85:585–598.
6. Lu S, Li T, Zhang Y, et al. Biomechanical optimization of the diameter of distraction screw in distraction implant by three-dimensional finite element analysis. *Comput Biol Med*. 2013;43:1949–1954.
7. Pessoa RS, Muraru L, Junior EM, et al. 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;12:219–234.
8. Doblare M, Garcıa JM, Gomez MJ. Modelling bone tissue fracture and healing: a review. *Eng Fract Mech*. 2004;71:1809–1840.
9. Liu S, Liu Y, Xu J, Rong Q, Pan S. Influence of occlusal contact and cusp inclination on the biomechanical character of a maxillary premolar: a finite element analysis. *J Prosthet Dent*. 2014;112:1238–1245.

10. Mahony AMO, 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*. 2000;9:648–657.
11. Baggi L, Cappelloni I, DiGirolamo M, Maceri F, Vairo G. The influence of implant diameter and length on stress distribution of osseointegrated implants related to crestal bone geometry: a three-dimensional finite element analysis. *J Prosthet Dent*. 2008;100:422–431.
12. Vairo G, Sannino G. Comparative evaluation of osseointegrated dental implants based on platform-switching concept: influence of diameter, length, thread shape, and in-bone positioning depth on stress-based performance. *Comput Math Methods Med*. 2013;2013:250929.
13. Shigemitsu R, Yoda N, Ogawa T, et al. Biological-data-based finite-element stress analysis of mandibular bone with implant supported overdenture. *Comput Biol Med*. 2014;54:44–52.
14. Robling AG, Castillo AB, Turner CH. Biomechanical and molecular regulation of bone remodeling. *Annu Rev Biomed Eng*. 2006;8:455–498.
15. Duyck J, Ronold HJ, VanOosterwyck H, Naert I, VanderSloten J, Ellingsen JE. The influence of static and dynamic loading on marginal bone reactions around osseointegrated implants: an animal experimental study. *Clin Oral Implants Res*. 2001;12:207–218.
16. Demenko V, Linetskiy I, Nesvit K, Shevchenko A. Ultimate masticatory force as a criterion in implant selection. *J Dent Res*. 2011;90:1211–1215.
17. Ueda T, Kremer U, Katsoulis J, Mericske-Stern R. Long-term results of mandibular implants supporting an overdenture: implant survival, failures, and crestal bone level changes. *Int J Oral Maxillofac Implants*. 2011;26:365–372.
18. Verri FR, de Souza Batista VE, Santiago Jr JF, de Faria Almeida DA, Pellizzer EP. Effect of crown-to-implant ratio on peri-implant stress: a finite element analysis. *Mater Sci Eng*. 2014;C45:234–240.
19. Urdaneta RA, Rodriguez S, McNeil DC, Weed M, Chuang SK. The effect of increased crown-to-implant ratio on single-tooth locking-taper implants. *Int J Oral Maxillofac Implants*. 2010;25:729–743.
20. de Sousa AA, Mattos BSC. Finite element analysis of stability and functional stress with implant-supported maxillary obturator prostheses. *J Prosthet Dent*. 2014. http://dx.doi.org/10.1016/j.prosdent.2014.06.020.
21. Berkovitz BKB. The structure of the periodontal ligament : an update. *Eur J Orthod*. 1990;12:51–76.
22. Qian L, Todo M, Morita Y, Matsushita Y, Koyano K. Deformation analysis of the periodontium considering the viscoelasticity of the periodontal ligament. *Dent Mater*. 2009;25:1285–1292.
23. Silva BR, Neto JJSM, Silva Jr FI, Aguiar ASW. Three-dimensional finite element analysis of the maxillary central incisor in two different situations of traumatic impact. *Comput Methods Biomech Biomed Eng*. 2011;16:158–164.
24. Su MZ, Chang HH, Chiang YC, et al. Modeling viscoelastic behavior of periodontal ligament with non linear fem analysis. *J Dent Sci*. 2013;8:121–128.
25. Aversa R, Apicella D, Perillo L, et al. Non-linear elastic three-dimensional finite element analysis on the effect of endo crown material rigidity on alveolar bone remodeling process. *Dent Mater*. 2009;25:678–690.
26. Tuna M, Sunbuloglu E, Bozdag E. Finite element simulation of the behavior of the periodontal ligament: a validated nonlinear contact model. *J Biomech*. 2014;47:2883–2890.
27. Roccia F, Diaspro A, Nasi A, Berrone S. Management of sport-related maxillofacial injuries. *J Craniofac Surg*. 2008;19:377–382.
28. Elhammali N, Bremerich A, Rustemeyer J. Demographical and clinical aspects of sports-related maxillofacial and skull base fractures in hospitalized patients. *Int J Oral Maxillofac Surg*. 2010;39:857–862.
29. Huempfner-Hierl H, Schaller A, Hemprich A, Hier T. Biomechanical investigation of naso-orbitoethmoid traumaby finite element analysis. *Brit J Oral Maxillofac Surg*. 2014;52:850–853.
30. Bujtár P, Sándor GKB, Bojtos A, Szucs A, Barabás J. Finite element analysis of the human mandible in 3 different stages of life. *Oral Surg Oral Med Oral Pathol Oral Radiol Endod*. 2010;110:301–309.
31. Santos LSdM, Rossi AC, Freire AR, Matoso RI, Ferreira Caria PH, Prado FB. Finite element analysis of three situations of trauma in human edentulous mandible. *J Oral Maxillofac Surg*. 2014. http://dx.doi.org/10.1016/j.joms.2014.10.014.
32. Szwedowski TD, Whyne CM, Fialkov JA. Toward characterization of craniofacial biomechanics. *J Craniofac Surg*. 2010;21:202–207.