ORIGINAL RESEARCH PAPER

Periodontics

"FINITE WITH INFINITE POSSIBILITIES"- "A BOON TO IMPLANT DENTISTRY" - A BRIEF OVERVIEW ON FINITE ELEMENT ANALYSIS

KEY WORDS: Dental implants, stress distribution, simulations, finite element analysis

Dr. Sangita Show*
MDS (Periodontics), (Ex-PGT) Dr. R. Ahmed Dental College & Hospital, Kolkata; Dental Surgeon, Chief Consultant, Trinity Dental & Maxillofacial Multispeciality Clinic, Budge Budge, Kolkata, West Bengal, India - 700137. *Corresponding Author

Dr. Arka Kanti Dey
MDS (Oral & Maxillofacial Surgery), Dental Surgeon cum Clinical Tutor / Demonstrator ,Diamond Harbour Government Medical College and Hospital, South 24 pgs,West Bengal,India.Pin Code-743331.

ABSTRACT

Numerous techniques to determine stress distribution around the peri-implant bone, for instance photoelastic model, strain gauge analysis, and three-dimensional (3D) finite element analysis have been extensively studied, however finite element analysis still remains the most widely utilized technique. This paper briefly reviews the fundamental concepts, applications pertaining to dental implants, various advantages as well as limitations of finite element analysis. The finite element method not only serves as a significant tool for biomechanical analysis, it also enables to reveal stress within complex structures and analyzing their mechanical properties. Keeping in mind the various limitations of the method further improvements might be made which would help to widen its range of applications in various domains of dental sciences.

INTRODUCTION

In the contemporary era of dentistry, the use of dental implants for restoring missing teeth has increased manifold. In case of implant-supported prosthesis the occlusal load is transferred to the implants and subsequently to the bone, in such a scenerio the biomechanical behaviour of the implant-supported prosthesis is believed to play a significant role in the longevity of the bone encompassing the dental implants.[1] The stress distribution pattern in case of dental implants differs completely from that of a natural tooth,[2] since the latter has a periodontal ligament that acts as a shock absorber to occlusal forces,[3] thus success or failure of dental implant primarily depends on the manner in which stress is transferred from dental implant to the adjoining alveolar bone.[4,5] Under the influence of stress, bone tissue undergoes a remodeling process, which influences the long-term function of a dental implant system.[6] Numerous researchers have extensively studied the biomechanical effect of force directions, force magnitudes, prosthesis type, prosthesis material, implant design, number and distribution of supporting implants, bone density, and the mechanical properties of the bone- implant interface in order to comprehend the steps of force transfer.[7] The resultant stress must be kept below the failure stress of the materials involved.[1,8]

In recent years the application of simulations have served as a vital tool to compute the best clinical options. Computerized modeling is a popularly utilized non-invasive way to predict *in vivo* contact mechanics. Various techniques to investigate stress distribution around the peri-implant bone, have been explored, for instance photoelastic model, strain gauge analysis, and three-dimensional (3D) finite element analysis (FEA).[9] Amongst these finite element analysis is the most commonly used method due to the ease of availability of software and the ability to determine 3D stresses and strains.[10] The finite element analysis (FEA) serves as a significant research tool for biomechanical analysis, aids in modeling complex structures and analyzing their mechanical properties.

Being a numerical method, it analyzes stresses and deformations in the structures of a said geometry. The structure is quantized into the so called "finite elements" which are in turn are interconnected through "nodes". The accuracy of the analysis is determined by the type, arrangement and total number of elements present.[11] The general steps involved in this analysis are constructing a finite element model, following which appropriate material properties, loading and boundary conditions are specified, so that the desired settings can be accurately simulated. Various software packages are widely available to model and simulate the structure of interest may be implants or jawbone. During the initial phases, finite element model was technologically innovated with the aim of analyzing structural difficulties involving the fields of Mechanics, Civil, and Aeronautical Engineering,[12] however eventually finite element analysis has proved its calibre as a widely utilized non- invasive and excellent tool for studying the biomechanics and the influence of mechanical forces on biological systems.

The 3D finite element model enables in-depth qualitative examination of the harmony amongst implant, tooth, periodontal ligament, and bone. Apart from the estimation of location, magnitude, and direction of an applied force by assigning of stress points that can be measured theoretically,[13] it also enables proper visualization of superimposed structures, and also allows the stipulation of the material properties of anatomic craniofacial structures.[14]

For the purpose of finite element analysis modeling, a finite element model geometry is constructed of both cortical and cancellous bone which is build via a series of binarized patient's CT image data. Following this the apparent density, porosity or apparent ash density assessment is carried out using different correlations to model the heterogenous distribution of mechanical properties. Isotropic behaviour is considered by most of the models, as because quantifying the whole anisotropic structure of a bone, organ is not possible with the currently utilized techniques.[15] Next step involves the application of load either to the implant or to the bone according to the requirement. Despite the fact that the muscle activity and craniofacial morphology affects the occlusal load in actual clinical situation, with the currently available models it is difficult to simulate individual muscle forces to finite element analysis modeling. For the consideration of input load in finite element analysis usually vertical or oblique load on the teeth or implant is utilized.[16] Literature suggests scholarly works where diligent data was acquired with respect to stress distribution in bone with a 3D finite element modeling, and the comparative analysis between 3D finite element analysis studies and *in vivo* strain gauge measurements depicted 3D finite element analysis results matched with those of *in vivo* strain gauge measurements, and were corresponding to clinical outcomes. [17]

Finite element analysis and dental implants

Homogenously distributed occlusal forces around a dental implant results in a well maintained osseous contour.[18] Finite element analysis provides an end to end knowledge about the stress patterns and their distribution in the implants as well as the peri-implant bone which would lead to an enhancement in implant designs and placement techniques. The technical difficulties of simulating living bone tissue and as well as its response to applied mechanical forces is solved to some extent with the use of advanced imaging techniques. For the purpose of detailed study of biomechanical properties of implants, finite element analysis has been utilized, also it has helped to predict the long term clinical success of the implants. The accuracy in simulating every minute details of geometry and surface structure of the implants, the loading and support conditions, the material properties of the implant and jawbone, as well as exact simulation of the biomechanical implant jawbone interface determines the success of finite element modelling. Any assumptions made during modeling or limitations of the software used might lead to errors in the analysis.[19]

In the finite element analysis studies, for the purpose of assessment of mechanical stress in the peri-implant bone, von Mises stress, the maximum principal stress, the minimum principal stress and the maximum shear stress are utilized. Amongst these the most frequently and primarily utilized scalar-valued stress invariant is the von Mises stress. This stress mainly appraises yielding/failure behaviour of various materials. On the other hand the maximum principal stress is used for the observation of tensile stress and the minimum gives an idea about the compressive stress. Bone has both ductile and brittle properties hence principle stress might be considered for such studies.[20] Mechanical stress might play a significant role in maintaining homeostasis of the bone [21] however occlusal overload was predicted to be a risk factor for peri-implant bone loss in studies conducted on animal models.[22] The outcome of finite element analysis are : (1) comprehensive and detailed geometry of the implant and peri-implant bone to be modeled, (2) material properties (3) boundary conditions (4) interface between bone and implant, (5) loading conditions (6) validation and (7) convergence test.[23]

Demenko et al.[24] in a finite element analysis study, suggested that the implant size should be selected keeping in mind its load bearing capacity. Finite element methods have further validated the fact that the higher crown to implant ratio increases the risk of mechanical failure. Similarly another study depicted that oblique loading induced higher stress to the fixation screw, chiefly when the crown: implant ratio was 1.5:1.[25] This result corroborates with the work of Urdaneta et al. [26] who demonstrated a significant correlation between screw loosening, fracture of prosthetic abutments, and crown height. The long term consequences of mandibular implant supported overdentures suggestive of loss of osseointegration without overt signs of infection was far more common than peri-implantitis.[27] Hence overall implications of biomechanical condition of the implant- bone interface for long term success of the implants cannot be denied.

In the past numerous studies have utilized finite element analysis to assess stress distribution related to dental implants. Scholarly work by Sotto-Maior BS., *et al.* [28] showed that the mechanoregulatory tissue model could be employed in monitoring the morphological changes in bone that is subjected to biomechanical loads. Also they found that the implant length did not have a contributory effect on the bone remodeling around single dental implants during the first year of loading. Balkaya MC., *et al.* [29] analyzed the biomechanical behaviour of implants with a varying number, inclination, and size, using 3-D finite element analysis. They observed that on decreasing the cantilever length with distal implant inclination decreases the stress values in the implant, cortical bone, and framework. Evaluation of biomechanics of short dental implants was conducted by Kang N., *et al.* [30] which depicted that implants with larger diameter (<5.5 mm) and bone quality enhancement may be preferable to get better clinical effects, however further prospective clinical studies are required to confirm this.

Xia H., *et al.* [31] investigated the stress distribution in the peri-implant bone around a platform-switched implant with marginal bone loss by using 3D- finite element analysis model and results of the study suggested a biomechanical advantage for platform switching in clinical scenerio of marginal bone resorption, however they also pointed out that this advantage was limited in clinical situations of extensive bone resorption. Another study conducted a comparative analysis amongst platform switched, sloping shoulder, and regular implants and their consecutive effects on stress reduction in various bone densities with the help of a finite element analysis model. They noted that sloping shoulder implants in subcrestal position is much favorable for bone growth, uniform stress distribution and preservation of remaining bone.[32] In scholarly work of Rubo and Souza [33] observations made were : (1) The increase in cantilever length is proportional to the increase in stress concentration (2) Increasing the abutment length (increases the lever arm) resulted in a decrease of the stress on implants and framework (3) The more rigid the framework, the better is the distribution of stress among the abutments/ implants, and less stress is seen in the framework (4) The increase of the elastic modulus of cancellous bone had little effect in the stress distribution in abutments/ implants and was inversely proportional to the stress in the framework (5) Increasing implant length from 10 to 13 mm resulted in less stress in the framework, but the same has not been demonstrated at the 13 to 15 mm change, and (6) The relative physical properties of the materials substantially affect the way stresses are distributed.

Advantages of finite element analysis:[19,23]

- Apart from the advantage of being a non-invasive technique, it enables proper visualization of superimposed structures.
- Specification and the material properties of anatomic craniofacial structures can be assessed.
- Offers maximum standardization.
- No ethical considerations and modifications in study designs can be done according to requirement.
- It not only helps in locating the magnitude and direction of an applied force, but at the same time it also provides stress points that can be measured theoretically.
- FEM can minimize laboratory testing requirements.
- Physical properties of the analyzed materials remain unaltered.
- It is easy to repeat as well as time saving.
- It is possible to conduct both static and dynamic analysis.
- No animal sacrifices are required to evaluate stress and strain.

Limitations of finite element analysis:[23]

- Despite the fact that finite element model helps in visualizing points of maximum
- stress and displacement, but it is comparatively difficult to predict failure in materials with complex geometric shapes composed of various materials, complex loading varying with relation to time and point of application further complicated by residual stresses.
- Requirement of through computer knowledge.
- Inaccuracy in information, statistics or interpretation will yield totally misguiding results.
- Inability of finite element analysis to simulate accurately

the biological dynamics of the tooth and its supporting structure.

CONCLUSION

In the recent past finite element analysis has emerged as a powerful research tool for biomechanical analysis and its use has increased manifold. Just as a coin has two sides, similarly finite element analysis has various advantages as well as there are certain limitations too. Being computerized *in vitro* studies, it might not be possible to completely replicate the clinical situations. Moreover the results are more of qualitative type since stress analysis is usually conducted under static loading. Unlike the real clinical scenario, mechanical properties of materials are set as isotropic and linearly elastic. In order to overcome such limitations to some extent, supplementary clinical evaluation might be conducted along side finite element analysis *in-vitro* studies. Research scholars and clinicians can utilize this modern technology to enhance prognosis as well as implant survival. This would be possible with elaborate assessment of the biomechanics of dental implants , use of advanced softwares, possessing profound knowledge of the methodology, advantages, and shortcomings of finite element analysis.[19]

REFERENCES

1. Skalak R. Aspects of biomechanical considerations.n In:Branemark P-I, Zarb GA, Albreksson T, eds. Tissueintegrated prostheses – Osseointegration in clinical dentistry. Chicago, IL: Quintessence, 1985:117–128.
2. Hsu ML, Chang CL, Finite element analysis. 1st ed. Rijeka: Sciyo, InTech Europe; 2010.
3. Koosha S, Mirhashemi FS. An investigation of three types of tooth implant supported fixed prosthesis designs with 3D finite element analysis. J Dent (Tehran) 2013;10:51-63.
4. Glantz PO, Nilner K. Biomechanical aspects of prosthetic implant-borne reconstructions. Periodontol 2000. 1998;17:119-24.
5. Akpinar I, Anil N, Parnas L. A natural tooth's stress distribution in occlusion with a dental implant. J Oral Rehabil 2000;27:538-545.
6. Meijer HJA, Starmans FJM, Bosman F, Steen WHA. A comparison of finite element models of an edentulous mandible provided with implants. J Oral Rehabil 1993; 20:147–157.
7. Sahin S, Çehreli MC, Yalçin E. The influence of functional forces on the biomechanics of implant-supported prostheses – a review. J Dent 2002; 30:271–282.
8. Frost HM. Wolff's Law and bone's structural adaptation to mechanical use: an overview for clinicians. Angle Orthod 1994; 64:175–188.
9. Srirekha A, Bashetty K. Infinite to finite: An overview of finite element analysis. Indian J Dent Res 2010;21:425-32.
10. Pesqueira AA, Goiato MC, Filho HG, Monteiro DR, Santos DM, Haddad MF, et al. Use of stress analysis methods to evaluate the biomechanics of oral rehabilitation with implants. J Oral Implantol 2014;40:217-228.
11. 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.
12. Bathe KJ. Finite element procedures. Upper Saddle River (NJ): Prentice-Hall; 1996. Chapter 4, p. 148-377.
13. 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.
14. 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.
15. Doblare M, Garcıa JM, Gomez MJ. Modelling bone tissue fracture and healing: a review. Eng Fract Mech. 2004;71:1809-1840.
16. 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.
17. Danza M, Zollino I, Paracchini L, Riccardo G, Fanali S, Carinci F. 3D finite element analysis to detect stress distribution: Spiral family implants. Maxillofac Oral Surg 2009;8:334-339.
18. Berglundh T, Persson L, Klinge B. A systematic review of the incidence of biological and technical complications in implant dentistry reported in prospective longitudinal studies of at least 5 years. J Clin Periodontol 2002;29:197-212.
19. Trivedi S. Finite element analysis: A boon to dentistry journal of oral biology and cranio facial research 2014 ;4:200-203.
20. 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.
21. Robling AG, Castillo AB, Turner CH. Biomechanical and molecular regulation of bone remodeling. Annu Rev Biomed Eng. 2006;8:455-498.
22. 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.
23. M Sesha Reddy., et al. "Application of Finite Element Model in Implant Dentistry". EC Dental Science 17.11 (2018): 1950-1957.
24. Demenko V, Linetskiy I, Nesvit K, Shevchenko A. Ultimate masticatory force as a criterion in implant selection. J Dent Res. 2011;90:1211-1215.
25. 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;45:234-240.
26. 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.
27. 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.
28. Sotto-Maior BS., et al. "Evaluation of bone remodelling around single dental implants of different lengths: a mechanobiological numerical simulation and validation using clinical data". Computer Methods in Biomechanics and Biomedical Engineering 2016 ;7: 699-706.
29. Balkaya MC. "Investigation of influence of different implant size and placement on stress distribution with 3-dimensional finite element analysis". Implant Dentistry 2014;6:716-722.
30. Kang N., et al. "A study of force distribution of loading stresses on implant-bone interface on short implant length using 3-dimensional finite element analysis". Oral Surgery, Oral Medicine, Oral Pathology, Oral Radiology, and Endodontology 2014; 5:519-523.
31. Xia H., et al. "The effect of platform switching on stress in peri-implant bone in a condition of marginal bone resorption: a three dimensional finite element analysis". International Journal of Oral and Maxillofacial Implants 2013; 3:122-127.
32. Markose J., et al. "Comparison of Platform Switched and Sloping Shoulder Implants on Stress Reduction in various Bone Densities: Finite Element Analysis". Journal of Contemporary Dental Practice 2017; 6: 510-515.
33. Rubo HJ. and Edson ACS. Finite-Element Analysis of Stress on Dental Implant Prosthesis, Clinical Implant Dentistry and Related Research 2010; 2: 105-113.