EC DENTAL SCIENCE
Research Article

Application of Finite Element Model in Implant Dentistry

M Sesha Reddy[1]*, S Rajasekar[2] and Hossam Abdelatty Eid Abdemagyd[3]

[1]*Periodontics, College of Dentistry, Gulf Medical University, Ajman, UAE*

[2]*Professor of Periodontology, Rajah Muthiah Dental College and Hospital, Annamalai University, Tamil Nadu, India*

[3]*Associate Professor, Periodontics, College of Dentistry, Gulf Medical University, Ajman, UAE*

***Corresponding Author:** M Sesha Reddy, Periodontics, College of Dentistry, Gulf Medical University, Ajman, UAE.

Received: April 18, 2018; **Published:** October 31, 2018

Abstract

Initially, FEM was technologically innovated which aimed at answering structural analysis difficulties involving Mechanics, Civil and Aeronautical Engineering. FEM basically stands for a numerical model of analyzing stresses as well as distortions in the form of any agreed geometry. There for the shape is discretized into the so-called ‘finite elements’ coupled through nodes. Accuracy of the results is determined by type, planning and total number of elements used for a particular study model.

3-D FE model was designed for in-depth qualitative examination of the relations amongst implant, tooth, periodontal ligament, and bone. Scholarly work equating work reliability, validated with a 3-D modeling suggested that meticulous data can be acquired with respect to stress distribution in bone. Comparative results from 3-D FEA studies showed that 3D FEA, when matched with *in-vivo* strain gauge measurements were corresponding with clinical outcomes. The aim of this review of literature is to provide an overview to show the application of FEM in (Mini) implant dentistry.

Keywords: *FEM; Mini-Implants; Stress Analysis; Platform Switched-Implants*

Abbreviation

FEM: Finite Element Model Analysis

Introduction

Since the discovery of dental implants by Brainmark 1969 it has become a ground breaking reality, of the use of dental implants for replacing a missing teeth [1]. Dental implants have become an inseparable part of dental practice and its use in recent years has increased in leaps and bounds. Clinical success of dental implant mainly depends on its biomechanical behavior as the pattern of stress distribution in dental implants is completely different from that of a natural tooth [2]. Since the later has periodontal ligament which acts as a shock absorber to occlusal forces [3]. Success or failure of dental implant mainly depends on a key feature i.e. the manner in which stress is transferred from dental implant to the adjoining alveolar bone [4,5].

If the occlusal forces around a dental implant are distributed homogenously then the bone is maintained well. When we look into the literature several attempts to preserve the marginal bone around dental implants has been done [6]. Contributing factors for marginal bone loss that have been accepted to some degree are biological, clinical and mechanical factors [7]. It is vital to understand the biomechanical behavior of bony tissues and dental implants in order to prevent marginal bone loss and implant failure.

In order to prevent implant failures and complications due to mechanical and technical factors, these factors have to be evaluated in advance. As a result use of these essential steps could increase the survival rate of implant-supported restorations. Hence, there has been a dramatic increase in the number of biomechanical studies in the field of implant dentistry in an effort to decrease dental implant failure rates [8].

Research in different fields of Dentistry needs a methodology that is cost effective and reproducible. Such an approach may perhaps be situated to guide researchers in biomechanics structure in healthy and pathologic conditions.

In bioengineering field, the application of simulations introduced in recent years, certainly is a vital instrument to measure the best clinical option, only if that it is precisely sufficient in investigation particular physiological conditions. Oral environment in biomechanical research such as restorative dentistry, endodontics, orthodontics, prosthodontics, periodontics, and Implantology has been performed *in vitro* since the oral cavity is an intricate biomechanical system due to this complexity and limited access [9].

A non-invasive way to predict *in vivo* contact mechanics is done mainly by using computerized modeling. To investigate stress distribution around peri-implant bone various methods have been current explored. To name a few we have photo elastic model, strain gauge analysis, and 3-dimensional finite element model analysis (FEA) [10]. Due to availability of software and the ability to determine 3D stresses and strains Finite Element Modeling (FEM) is considered the most commonly used method [11,12].

Initially, FEM was technologically innovated which aimed at answering structural analysis difficulties involving Mechanics, Civil and Aeronautical Engineering. FEM basically stands for a numerical model of analyzing stresses as well as distortions in the form of any agreed geometry [13,14]. There for the shape is discretized into the so-called ‘finite elements’ coupled through nodes. Accuracy of the results is determined by type, planning and total number of elements used for a particular study model [15].

3-D FE model was designed for in-depth qualitative examination of the relations amongst implant, tooth, periodontal ligament, and bone. Scholarly work equating work reliability, validated with a 3-D modeling suggested that meticulous data can be acquired with respect to stress distribution in bone. Comparative results from 3-D FEA studies showed that 3D FEA, when matched with *in-vivo* strain gauge measurements were corresponding with clinical outcomes [16].

Objective

The aim of this study is to provide an overview to show the application of FEM in implant dentistry.

Materials and Methods

An internet search was done using PUBMED search engine using keywords FEM, 3-D finite element analysis, and dental implants. Only original studies and reviews were included for the year June 2017 to 2012 in this review, a full-text article in English was only considered for the review. Reports, commentaries, and letters to the editors were excluded. Articles in other languages and abstracts were not included.

Results and Discussion

S. No	Author	Year	Study Variables	Conclusion
1	Yashwant AV., *et al.* [17]	2017	Mini implants of five different designs in thread shape were used with 10 screws in each group. The mini implants were loaded on to the polyurethane foam perpendicular to the surface and the pull-out strength was tested using the Instrom testing machine. The control group consisted of mini implants with reverse buttress thread shape.	Trapezoidal fluted mini implants showed the highest pull out strength when compared to mini implants with other thread designs used in this study.
2	Küçükkurt S., *et al.* [18]	2017	Authors compared the success of sinus lifting and alternative treatment methods in applying dental implants in cases lacking adequate bone due to pneumatization of the maxillary sinus.	LSL method should be the first choice among treatment options. Considering its successful results under conditions of oblique forces, the SIP method may be preferable to the TIP method. In contrast, every effort should be made to avoid the use of DCs.
3	Yazicioglu D., *et al.* [19]	2016	Stress distribution of the short dental implants and bone-to-implant contact ratios in the posterior maxilla using 3D -FEM	The von Mises stress values on the implants and the cancellous bone around the implants of the 70% bone-to-implant contact group were almost 3 times higher compared with the values of the 100% bone-to-implant contact group.
4	Sotto-Maior BS., *et al.* [20]	2016	The aim was to simulate bone remodeling around single implants of different lengths using FEA and to validate the theoretical prediction with the clinical findings of crestal bone loss.	Results showed that the mechanoregulatory tissue model could be employed in monitoring the morphological changes in bone that is subjected to biomechanical loads. In addition, the implant length did not influence the bone remodeling around single dental implants during the first year of loading.
5	Kheiralla LS., *et al.* [21]	2014	This study compared the biomechanical responses of 3 single crowns supported by 3 different implants under axial and off-axial loading. Each implant supported a full metal crown made of Ni-Cr alloy with standardized dimensions. Strain gauges and FEA were used to measure the strain induced under axial and off-axial functional loads of 300 N.	Standard and short-wide implants proved to be preferable in supporting crowns, as the standard implant showed the lowest strains under axial and off-axial loading using FEA simulation, while the short-wide implant showed the lowest strains under nonaxial loading using strain gauge analysis.
6	Balkaya MC., *et al.* [22]	2014	The aim was to analyze the biomechanical behavior of implants with a varying number, inclination, and size, using 3-D FE analysis.	Decreasing cantilever length with distal implant inclination decreases the stress values in the implant, cortical bone, and framework.
7	Kang N., *et al.* [23]	2014	The study was done to evaluate the biomechanics of short dental implants.	Results revealed that implants with larger diameter (<5.5 mm) and bone quality enhancement may be preferable to get better clinical effects. Prospective clinical studies are required to confirm this.
8	Kim S., *et al.* [24]	2104	This study evaluated the biomechanical behavior of short dental implants with different heights of residual bone and compared it with that of standard dental implants in 13 mm or less of residual bone by means of FEM	This numeric simulation confirmed that, without maxillary sinus bone graft, more effective stress distribution could be obtained in 4, 5, 6, or 7 mm of residual bone with short dental implants than in 13 mm of residual bone with standard dental implants.
9	Baggi L., *et al.* [25]	2013	This study aimed to investigate the influence of implant design, in-bone positioning depth, and bone post healing crestal morphology on load transfer mechanisms of osseointegrated dental implants based on the platform-switching concept.	Proposed results contribute to identifying the mutual influence of a number of factors affecting the bone-implant loading transfer mechanisms, furnishing useful insights and indications for choosing and/or designing threaded osseointegrated implants.
10	Toniollo MB., *et al.* [26]	2013	FEA compared stress distribution on external surface of different morse taper implants, varying implant bodies length and dimensions of metal-ceramic crowns in order to maintain the occlusal alignment.	Moreover, these 5mm implants were positioned at the cortical bone level, which has higher elastic modulus and may have influenced at the stress distribution. However, despite the higher stresses, these implants were well able to withstand the applied forces.
11	Toniollo MB., *et al.* [27]	2012	FEA compared stress distribution on different bony ridges rehabilitated with different lengths of morse taper implants, varying dimensions of metal-ceramic crowns to maintain the occlusal alignment.	It was concluded that patients requiring short implants associated with increased proportions implant prostheses need careful evaluation and occlusal adjustment, as a possible overload in these short implants, and even in regular ones, can generate stress beyond the physiological threshold of the surrounding bone,
12	de Carvalho NA., *et al.* [28]	2012	Aim was to evaluate stress distribution on peri-implant bone simulating the influence of implants with different lengths on regular and switching platforms in the anterior maxilla by means of FEM	The influence of switching platform was more evident for the cortical bone in comparison with the trabecular bone for the short and long implants. The long implants showed lower stress values in comparison to the short implants, mainly when the switching platform was used.

Table 1: *Use of the FEM to evaluate the stress of Mini Implants.*

1	Markose J., *et al*. [29]	2017	Comparison of platform switched (PS), sloping shoulder, and regular implants on stress reduction in various bone densities with FEM.	Sloping shoulder implant in subcrestal position is much favorable for bone growth, stress distribution, and preservation of remaining bone.
2	Aradya A., *et al*. [30]	2016	The study was designed to evaluate and compare stress distribution in the transcortical section of bone with normal abutment and platform switched abutment under vertical and oblique forces in posterior mandible region.	Results from this study showed the platform switched abutment led to a relative decrease in von Mises stress in the transcortical section of bone compared to normal abutment under vertical and oblique forces in posterior mandible region.
3	Xia H., *et al*. [31]	2013	The aim of the present study was to investigate the stress distribution in the bone around a platform-switched implant with marginal bone loss.	Results suggest a biomechanical advantage for platform switching in a condition of marginal bone resorption, but this advantage may be weakened when bone resorption is dramatic.
4	Rismanchian M., *et al*. [32]	2013	The purpose of this study was to evaluate peri-implant bone stress distribution for platform-switched implants placed at different depths relative to the bone crest, maintaining the occlusal plane at the same level.	It can be concluded that shallow subcrestal placement of 2-stage platform-switched implants only slightly increases the stress within the cortical bone.
5	Vargas LC., *et al*. [33]	2013	The aim of this study was to evaluate stress distribution of the peri-implant bone by simulating the biomechanical influence of implants with different diameters of regular or platform switched connections by means of 3-D FEM	The influence of platform switching was more evident for cortical bone than for trabecular bone and was mainly seen in large platform diameter reduction.

Table 2: *3D FEM and stress distribution in platform switched-implants.*

Application of finite element analysis in dentistry

Meticulous quantifiable information on any place inside a mathematical model can be provided by Finite element analysis (FEA). As a result, FEA has to turn out to be a valued analytical instrument in the estimation of stress and strain in implant systems. One of the salient characteristics of FEM rests in its near physical similarity amongst the real structure as well as its FEM. However unnecessary simplification in geometry shall invariably lead to inconsistent results [34-37].

Rules general followed in FEM and implant-bone biomechanics

FEA model is able to apply to various physical problems and its power lies in its versatility. The structure that is to be analyzed may have random form, loads, and supporting conditions, in addition, the mesh be able to amalgamate features of diverse shapes, types, and physical properties.

FEA outcomes give: (1) complete geometry of the implant and surrounding bone to be modeled, (2) sboundary conditions, (3) material properties, (4) loading conditions, (5) interface between bone and implant, (6) convergence test, (7) validation [2].

Advantages of FEM

- It enables the visualization of superimposed structures
- Specification and the material properties of anatomic craniofacial structures can be evalauted1
- We can locate the magnitude and direction of an applied force.
- It provides stress points that can be measured theoretically
- Physical properties of the analyzed materials are not altered
- It is easy repeat
- It is a non-invasive technique
- Both Static and dynamic analysis can be carried out
- A reduced amount of time spent
- Study can be repeated as many times
- No need to sacrifice animals to evaluate stress and strain

Disadvantages of FEM

- Incorrect information, statistics, and interpretation will yield totally misguiding results.
- Need to have computer knowledge
- Need to have thorough information about their mechanical behaviors.
- Certain expectations are bound to be accepted. Hence outcomes will be determined by people associated in the study

Stress Values Evaluation and its Validity in FEM study

Stress distribution in FEM studies are generally interpreted as von Mises stress which could be maximum and minimum principal stress or it could be principal strains [38]. Von Mises stress are estimated in three plains i.e. X-axis, Y-axis and Z-axis using a formula [39]. Validation is done by comparing the current FEM results with that of the previous studies related to a particular topic. It provides insight knowledge weather precise models were designed for the study or not. Further, it would corroborate the results of previous studies and it may either support or refute with the literature. The best way to validate FEM results is to conduct in vitro and in vivo experimental studies simultaneously. If the results are good then it could be recommended for future studies [40].

Conclusion

For better understanding, the biomechanics of dental implants the use of computer technology alongside with more profound awareness about the concept, methodology, advantages, and limitations of FEA have to be assessed elaborately. As a result, clinicians can use this modern technology to enhance implant survival by well accepting the biomechanics of dental implantology.

In this article, authors had made an effort to address the basics of FEA in dental Implantology. The ingredients which create FEA a powerful tool sufficient to reliably mention on flexible stress states in a complex structure are known.

Similar to any other instrument used to resolve a problem, the explanation made can only be as robust as the suitable application of the instrument itself. Upcoming investigation ought to attempt to correlate results with clinical findings in doing so it escalates the validity of the models. In addition simulate the consequence of saliva, infection and fatigue failure under repetitive, realistic, cyclic loading conditions have to be evaluated.

Conflict of Interest

I hereby declare there is no conflict of interest.

Bibliography

1. Brånemark PI., *et al*. “Intra-osseous anchorage of dental prosthesis.1. Experimental studies”. *Scandinavian Journal of Plastic and Reconstructive Surgery and Hand Surgery* 3.2 (1969): 81-100.
2. Hsu ML and Chang CL. “Finite Element Analysis”. First Edition, Sciyo, InTech Europe, Rijeka, (2010).
3. Koosha S and Mirhashemi FS. “An Investigation of Three types of Tooth Implant Supported Fixed Prosthesis Designs with 3D Finite Element Analysis”. *Journal of Dentistry* 10.1 (2013): 51-63.
4. Glantz PO and Nilner K. “Biomechanical aspects of prosthetic implant-bone reconstructions”. *Periodontology 2000* 17.1 (1998): 119-124.
5. Akpinar I., *et al*. “A natural tooth’s stress distribution in occlusion with a dental implant”. *Journal of Oral Rehabilitation* 27.6 (2001): 538-545.
6. Berglundh T., *et al*. “A systematic review of the incidence of biological and technical complications in implant dentistry reported in prospective longitudinal studies of at least 5 years”. *Journal of Clinical Periodontology* 29.3 (2002): 197-212.
7. Rocha S., *et al*. “Effect of platform switching on crestal bone levels around implants in the posterior mandible: 3 years results from a multicentre randomized clinical trial”. *Journal of Clinical Periodontology* 43.4 (2016): 374-382.
8. Gaviria L., *et al*. “Current trends in dental implants”. *Journal of the Korean Association of Oral and Maxillofacial Surgeons* 40.2 (2014): 50-60.
9. Piccioni MRV., *et al*. “Application of the finite element method in Dentistry”. *RSBO Revista Sul-Brasileira de Odontologia* 10.4 (2013): 369-377.
10. Srirekha A and Bashetty K. “Infinite to finite: An overview of finite element analysis”. *Indian Journal of Dental Research* 21.3 (2010): 425-432.
11. Pesqueira AA., *et al*. “Use of stress analysis methods to evaluate the biomechanics of oral rehabilitation with implants”. *Journal of Oral Implantology* 40.2 (2014): 217-228.
12. Reddy K., *et al*. “Cervical stress due to normal occlusal loads is a cause for abfraction? - A finite element model study”. *Journal of Orofacial Sciences* 4.2 (2012): 120-123.
13. Bathe KJ. “Finite element procedures [M]”. Upper Saddle River (NJ): Prentice-Hall (1996): 148-377.
14. Gultekin BA., *et al*. “Finite Element Analysis - New Trends and Developments”. First Edition, Sciyo, InTech Europe, Rijeka (2012).
15. Chen L. “Finite Element Analysis - New Trends and Developments”. Book edited by Farzad Ebrahimi (2012).
16. Matteo Danza., *et al*. “3D finite element analysis to detect stress distribution: spiral family implants”. *Journal of Maxillofacial and Oral Surgery* 8.4 (2009): 334-339.
17. Yashwant AV., *et al*. “Does Change in Thread Shape Influence the Pull Out Strength of Mini Implants? An In vitro Study”. *Journal of Clinical and Diagnostic Research* 11.5 (2017): ZC17-ZC20.

18. Küçükkurt S., *et al*. “Biomechanical comparison of sinus floor elevation and alternative treatment methods for dental implant placement”. *Computer Methods in Biomechanics and Biomedical Engineering* 20.3 (2017): 284-293.

19. Yazicioglu D., *et al*. “Stress Distribution on Short Implants at Maxillary Posterior Alveolar Bone Model with Different Bone-to-Implant Contact Ratio: Finite Element Analysis”. *Journal of Oral Implantology* 42.1 (2016): 26-33.

20. 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* 19.7 (2016): 699-706.

21. Kheiralla LS and Younis JF. “Peri-implant biomechanical responses to standard, short-wide, and mini implants supporting single crowns under axial and off-axial loading (an in vitro study)”. *Journal of Oral Implantology* 40.1 (2014): 42-52.

22. Balkaya MC. “Investigation of influence of different implant size and placement on stress distribution with 3-dimensional finite element analysis”. *Implant Dentistry* 23.6 (2014): 716-722.

23. 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* 118.5 (2014): 519-523.

24. Kim S., *et al*. “A three-dimensional finite element analysis of short dental implants in the posterior maxilla”. *International Journal of Oral and Maxillofacial Implants* 29.2 (2014): e155-e164.

25. Baggi L., *et al*. “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”. *Computational and Mathematical Methods in Medicine* (2013): 250929.

26. Toniollo MB., *et al*. “Three-dimensional finite element analysis of the stress distribution on morse taper implants surface”. *Journal of Prosthodontic Research* 57.3 (2013): 206-212.

27. Toniollo MB., *et al*. “Three-dimensional finite element analysis of stress distribution on different bony ridges with different lengths of morse taper implants and prosthesis dimensions”. *Journal of Craniofacial Surgery* 23.6 (2012): 1888-1892.

28. de Carvalho NA., *et al*. “Short implant to support maxillary restorations: bone stress analysis using regular and switching platform”. *Journal of Craniofacial Surgery* 23.3 (2012): 678-681.

29. 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* 18.6 (2017): 510-515.

30. Aradya A., *et al*. “Influence of different abutment diameter of implants on the peri-implant stress in the crestal bone: A three-dimensional finite element analysis--In vitro study”. *Indian Journal of Dental Research* 27.1 (2016): 78-85.

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* 28.3 (2013): e122-e127.

32. Rismanchian M., *et al*. “The effect of placement depth of platform-switched implants on peri-implant cortical bone stress: a 3-dimensional finite element analysis”. *Implant Dentistry* 22.2 (2013): 165-169.

33. Vargas LC., *et al*. “Regular and switching platform: bone stress analysis with varying implant diameter”. *Journal of Oral Implantology* 39.3 (2013): 326-331.

34. Shilpa T. "Finite element analysis: A boon to dentistry". *Journal of Oral Biology and Craniofacial Research* 4.3 (2014): 200-204.

35. Mohamed I El-Anwar and Mohamed M El-Zawahry. "A three dimensional finite element study on dental implant design". *Journal of Genetic Engineering and Biotechnology* 9.1 (2011): 77-82.

36. DeTolla DH., *et al*. "Role of the finite element model in dental implants". *Journal of Oral Implantology* 26.2 (2016): 77-81.

37. Van Staden RC., *et al*. "Application of the finite element method in dental implant research". *Computer Methods in Biomechanics and Biomedical Engineering* 9.4 (2006): 257-270.

38. Bahuguna R., *et al*. "Evaluation of stress patterns in bone around dental implant for different abutment angulations under axial and oblique loading: A finite element analysis". *National Journal of Maxillofacial Surgery* 4.1 (2013): 46-51.

39. Akin JE. "Finite Element Analysis Concepts: Via SolidWorks". World scientific publishing 5 Ton Tuck Link Singapore (2010).

40. Omori M., *et al*. "A biomechanical investigation of mandibular molar implants: reproducibility and validity of a finite element analysis model". *International Journal of Implant Dentistry* 1.1 (2015): 10.

Volume 17 Issue 11 November 2018