

Basic Concepts of Finite Element Analysis and its Applications in Dentistry: An Overview

Mohammed SD^{1*} and Desai H²

¹Senior Lecturer, Manubhai Patel Dental College and Oral Research Institute, Orthodontics and Dentofacial Orthopaedics, India

²Professor and Head, Department of Orthodontics and Dentofacial Orthopedics, Manubhai Patel Dental College and Hospital, India

Abstract

In attempt to improve mechanical properties of various dental structures, analysis of stress and strain of the same under various loading circumstances has become an integral part of research in recent era. As we all know, oral cavity consists of various complex structures with a very limited accessibility. Due to this, most biomechanical research of the oral environment has been performed *in vitro*. Finite Element Analysis (FEA) is a modern tool for numerical stress analysis, with an advantage of being applicable to solids of irregular geometry that contain heterogeneous material properties. The history of Finite Element Analysis (FEA) dates back to 1943 when R. Courant first developed this technique. This article provides a review of the achievements and advancements in dental technology brought about by all powerful finite element method of analysis. The scope of the review covers various steps of finite element analysis, its applications in context to orthodontics, restorative dentistry and endodontics as well as to the field of implantology. Advantages and limitations are discussed in some detail which helps identify the gaps in research as well as future research direction.

Keywords: Finite element analysis; 3 Dimensional; Orthodontics; Implants; Restorative dentistry

Introduction

Stress analysis of dental structures has been a topic of interest in recent years with an objective of determining stresses in the dental structures and improvement of the mechanical strength of these structures. As we all know, oral cavity is a complex biomechanical system with limited access. Due to this, most biomechanical research of the oral environment such as in orthodontics, implantology, restorative dentistry, endodontics, prosthodontics etc. have been performed *in vitro*. Finite Element Analysis (FEA) is a modern tool for numerical stress analysis, with an advantage of being applicable to solids of irregular geometry that contain heterogeneous material properties. Such numerical techniques may yield an improved understanding of the reactions and interactions of individual tissues [1,2]. The science of finite element analysis (FEA) is purely a mathematical way of solving complex problems in the universe, as it gives easier mathematical solution to biological problems. Usually Many people get confused between the terminologies FEM and FEA. In reality, both are one and the same. FEA is more popular in industries and FEM at universities. It was introduced originally as a method for solving structural mechanical problems, which was later recognized as a general procedure for numerical approximation to all physical problems that can be modelled by a differential equation description. It involves a series of computational procedures to calculate the stress and strain in each element. This makes it possible to adequately model the tooth and periodontal structure by dividing the problem domain into a collection of much smaller and simpler domains. The field variables can be interpolated with the use of shape functions for scientific checking and validating the clinical assumptions [3]. The structure is discretized into so called “elements” connected through nodes. When choosing the appropriate mathematical model, element type and degree of discretization are important to obtain accurate as well as time and cost effective solutions.

Historical Perspective

R. Courant was a first researcher who developed this technique [4]. His main goal was to minimize the various calculative procedures to

gain absolute solution to bio-mechanical system. He used ritz method to solve such numeric equations. Later in Turner et al. attempted to describe this method by developing broader definition of these numeric analyses [5]. Weinstein in 1976 used this technique in implant dentistry to evaluate various loads of occlusion on implant and adjacent bone. Since then evolution of this technique has been observed in a very rapid and sophisticated scale in micro-computer as well as analysis of large scale structural system [6].

Advantages of FEM

- FEM can be applicable to linear and non-linear as well as solid and fluid structural interactions.
- Any problems can be split into smaller number of Problems.
- It is a non-invasive technique.
- By using FEA, It's very easy to simulate any biological condition in pre, intra and post-operative stages to achieve more accurate and reliable results.
- Reproducibility does not affect the physical properties involved.
- FEA techniques can replace stereo lithographic models for presurgical planning, thus provides an economical solution for the same.
- Static and dynamic analysis can be done.
- This technique is less time consuming, so that the complicated

***Corresponding author:** Mohammed SD, Senior Lecturer, Manubhai Patel Dental College and Oral Research Institute, Orthodontics and Dentofacial Orthopaedics, India, Tel: 919725656250; E-mail: shahid_4404@yahoo.co.in

Received June 21, 2014; **Accepted** August 08, 2014; **Published** August 14, 2014

Citation: Mohammed SD, Desai H (2014) Basic Concepts of Finite Element Analysis and its Applications in Dentistry: An Overview. Oral Hyg Health 2: 156. doi: [10.4172/2332-0702.1000156](http://dx.doi.org/10.4172/2332-0702.1000156)

Copyright: © 2014 Mohammed SD et al. This is an open-access article distributed under the terms of the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original author and source are credited.

studies which would take a very long duration to finish can now be evaluated in a lesser time frame.

- It does not require extensive instrumentation.
- The study can be repeated as many times as the operator wants.

Shortcomings of FEM

- Inaccurate data, information, and interpretation will yield totally misleading results.
- Modelling human structures is extremely difficult because of their complex anatomy and lack of complete knowledge about their mechanical behaviours. Certain assumptions are bound to be accepted. Hence results will depend on the personnel involved in the process.
- The progress in the FEA will be limited until better defined physical properties for enamel, dentin and periodontal ligament and cancellous and cortical bone are available.
- The tooth is treated as pinned to the supporting bone, which is considered to be rigid and the nodes connecting the tooth to the bone are considered fixed. This assumption will introduce some error.

Process of FEA

The process of finite element analysis can be divided into three phases:

- Pre-processing phase
- Solution phase
- Post-processing phase

i) The pre-processing phase is comprised of following steps

- Defining the type of element :
 - It's usually directly proportional to accuracy of model.
- Defining property of given material :
 - Physical properties are assigned to the selected material.
 - Young's modulus and poisson's ratio are minimal physical properties to be assigned.
 - These properties will decide the behaviour of materials after specific load application.
 - Table 1 showing ideal physical properties of dental structures and materials [7].
- Model creation :
 - 3-D CT scan and 3-D laser scan is used to achieve it.
 - For living structure 3-D scanner is preferable method while for non living structures 3-D laser scan is used.
- Defining the mesh density :
 - As the number of elements increases, the finite element model will become more accurate.
 - Create more number of element is time consuming procedure, So in order to improve accuracy in a less time area of interest are modelled with more number of element than other areas.

Materials	Young's Modulus	Poisson's Ratio
Teeth	2.60E+04	0.30
Periodontal Ligament	6.80E-01	0.49
Bone	1.40E+04	0.30
Stainless Steel	2.00E+05	0.31
Orthodontic Bracket	2.14E+05	0.30

Table 1: Ideal physical property of dental structures and materials.

ii) Solution phase:

- Boundary condition is defined in this phase.
 - Boundary conditions means that suppose an element is constructed on the computer and a force is applied to it, it will act like a free-floating rigid body and will undergo a translatory or rotatory motion or a combination of the two without experiencing deformation. To study its deformation, some degrees of freedom must be restricted (movement of the node in each direction x, y, and z) for some of the nodes. Such constraints are termed boundary conditions.

iii) Post processing phase :

- It includes result output obtained via the processing phase.
 - Output can be achieved via following three different manners.
 - Graphical output
 - Graphic outputs are usually more informative.
 - The output is primarily in the form of colour-coded maps. The quantitative analysis is determined by interpreting these maps.
 - The colours vary from red to blue. Red represents the area of maximum tensile stress and blue represents the area of maximum compressive stress.
 - Numeric output
 - This kind of output displays amount of principle stress/strain into the given material.
 - Animated output
 - In such kind of output manner, results are shown as animation for better visualization.

Flow chart 1 represent summery of phases of finite element analysis.

iv) Commercial FEM Software

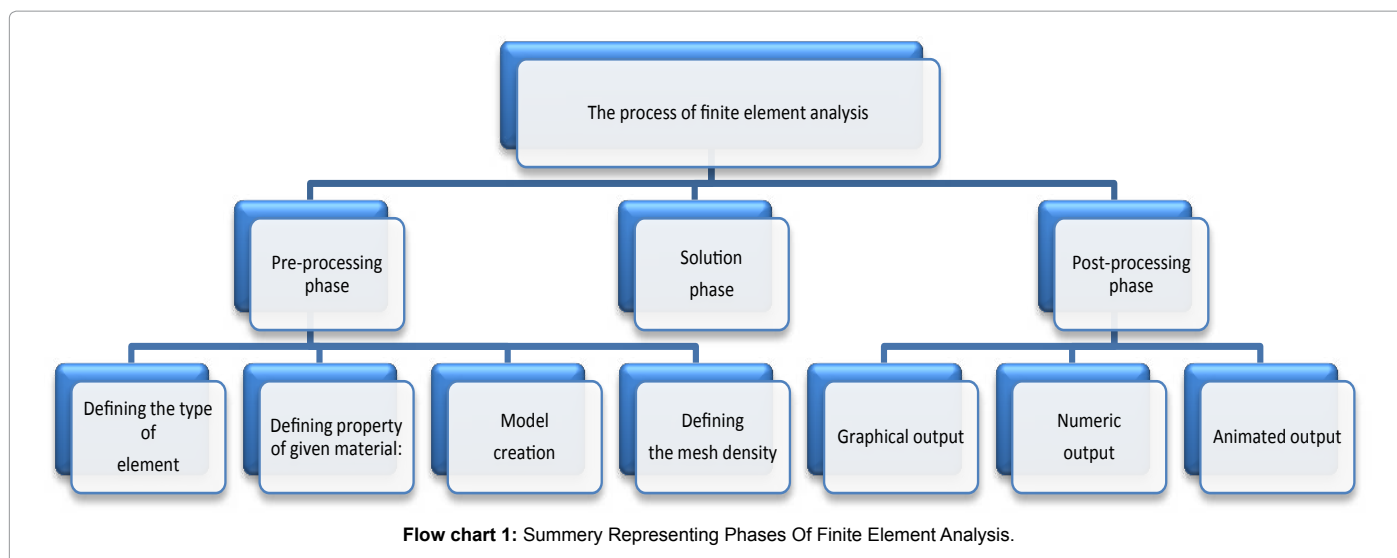
- ABAQUS (Non linear and dynamic)
- ANSYS
- HYPER MESH (Pre/Post processor)

Review of Literature

Numerous studies have been done over the years by dental researchers using the finite element analysis. It is beyond the scope of this paper to review all the published finite element studies in different specialities of dentistry, but a few of them are cited below with a title of their respective speciality.

Orthodontics and Dentofacial Orthopaedics

Liang et al. [8] generated 3-D finite element models of maxilla and maxillary incisors to evaluate the torque control of maxillary incisors during retraction in labial and lingual orthodontic technique. 3D force system including retraction, intrusion and moment of the force was



applied to predetermined points to simulate retraction in both the techniques. Distribution of stress and strain on PDL and adjacent bone as well as resultant movements of incisors were evaluated and compared. Results showed same amount of forces results in bodily movement and uncontrolled tipping in labial and lingual techniques respectively.

Sung et al. [9] generated a 3D finite element models of mandible and mandibular incisors to evaluate the anti tip and anti rotation effect of reverse curve of spee in labial and lingual orthodontics. 150 gm of retraction force was applied to canine in both the techniques. Every time amount of spee was increased from 0 mm to 4 mm. Resultant stress and strain on PDL was observed and compared. The comparison showed, reverse curve of spee works well with labial technique in comparison to lingual technique.

A study [10] was done to investigate the location of the centre of resistance (CRe) for the nasomaxillary complex by the use of finite element analysis. A three-dimensional finite element model of the craniofacial complex was used. 9.8 N of force was applied in anterior and inferior direction at pre determined points which were parallel and perpendicular to occlusal plane. After force application 3D displacement of various anatomic points on nasomaxillary complex as well as maxillary dentition was observed. When load was applied in horizontal plane passing from superior ridge of the pterygomaxillary fissure, resultant movement was observed to be translation. This suggested that CRe of the nasomaxillary complex is located on the posterosuperior ridge of the pterygomaxillary fissure, registered on the median sagittal plane.

A study [11] was done to evaluate level of stress in single and multi tooth system using 3D FEM technique. Two different models were made which consisted single mandibular canine and a mandibular incisors, canine and premolars respectively. Loads were applied on both models to simulate tipping orthodontic movement. The observed stress distribution stated that there was a elevated distortion strain at crest of alveolar ridge, while the tensile and compressive stresses coincide at tooth apex which resulted in root resorption. Multi tooth system consisted more stress levels in comparison to single tooth system.

A study [12] was undertaken to evaluate stress generation at root apex during orthodontic tooth movements. A 3-dimensional finite

element model of a maxillary central incisor, periodontal ligament (PDL), and alveolar bone was constructed on the basis of average anatomic morphology. The material properties of enamel, dentin, PDL, and bone were compared under various possible tooth movements. Results showed that higher stresses were found concentrated at apex during intrusive, extrusive, and rotational movements. The principal stress from a tipping force was located at the alveolar crest. For bodily movement, stress was distributed throughout the PDL.

Operative Dentistry and Endodontics

Ichim et al. [13] did a 3D FEM study to investigate co-relation of cavity shape, depth and occlusal forces with durability of GIC restoration. They have concluded that depth and shape have no significant effect on restoration. They advised to re-adjust the inter-occlusal contacts for better retention of restoration. They suggested occlusal readjustment of tooth contacts. Another study [14] by the same authors using nonlinear technique for crack propagation showed the mechanical failure of biomaterials in clinical loading conditions. Further studies by these authors on elastic modulus of materials stated that more flexible materials with elastic modulus of 1 GPa should be used for cervical restorations for the better results [15].

Asmussen et al. [16] did a study to evaluate influence of modulus of elasticity of resin composite in order to minimize marginal failure by using 3D finite element analysis. They have taken class-1 and 2 restorations in consideration and various occlusal loading were applied on prepared models. Stress distribution was observed which concluded that restorations should have a high modulus of elasticity in order to reduce the risk of marginal deterioration.

Subramaniam et al. [17,18] did a 3D FEM study to analyse bending and torsional stresses in simulated models of Protaper and Profile nickel-titanium rotary instruments. They stated that when loads are equal, the Protaper model showed uniform distribution of stress and less elasticity compared with Profile model. Few shortcomings of their study were that they ignored the variations in taper of the Protaper and Profile instruments and also nonlinear mechanical behaviour of the Ni-Ti material was not considered.

Hong et al. [19] undertook a 3D FEM study to find out better method of condensation in root canal to prevent vertical root fracture. They found that vertical condensation technique generate high stress

on root canal walls than lateral condensation. But over forced lateral condensation with improper technique can be a reason for vertical root fracture.

Several studies [20,21] have suggested that the complex interaction caused by a non-functional distribution of occlusal loads, combined with poorly developed enamel and the demineralising and weakening effects of erosive acids may operate to produce non-carious cervical tooth loss.

Prosthodontia and Implantology

Siegele and Soltesz [22] did a 3D FEM study to evaluate stress generation within the jawbone after insertion of implants of various shapes. They concluded that different implant shapes creates different stress distributions in the jawbone. They also stated that conical implant imply distinctly higher stresses than cylindrical and screw-shaped implants.

Genj et al. [1] reviewed the present application of FEA in implant dentistry. They concluded that, to prepare models with high accuracy, advanced digital imaging techniques should be used. The anisotropic and non-homogenous nature of the material should be considered; and boundary conditions must be refined to the top most level. In addition, to prevent prosthesis failure change in implant design was suggested for proper stress distribution.

Himmlova et al. [23] undertake a 3D FEM study to evaluate stress values produced at the implant-bone interface. They have taken implants of various lengths and diameter. They found Maximum stress areas at the neck of implant. The maximum decrease in stress was found with increasing the diameter of implant up to 4.5 mm. Stress observed to decrease with increase in length of implant. However, length was not as influencing as that of implant diameter.

Mailath et al. [24] did a 3D FEM study to evaluate the stress values at the level of bone while placing implants having different shapes and designs. They have compared cylindrical and conical implant shapes exposed to physiological stresses. They concluded that cylindrical implants produced a more desirable stress profile than the conical shaped. The need for tapered implants have been raised due to the fact that the survival rate of oral implants in soft quality bone were demonstrated to be inferior to that of implants inserted in good-quality bone.

Dos Santos et al. [25] did a 3D FEM study to evaluate and compare stress distribution between submerged and non-submerged implants and concluded that the simulations with non-submerged implants showed higher values of stress concentration than those that were submerged. It was also stated that soft liner materials gave better results. They also stated that use of soft liners with submerged implants seems to be the most suitable method to be used during the Osseo integration phase. The data were evaluated using Maximum Principal Stress.

Haraldson et al. [26] evaluated the mastication forces and chewing efficiency in nine patients, whom were treated with over dentures on Osseo integrated implants in the mandible. The bite force was measured during gentle biting, biting as when chewing and biting with maximal effort. All subjects improved subjectively as well as clinically after treatment. The bite force was increased from 17.3 to 24 N and from 24.0 N to 38.7 N during gentle biting and during chewing respectively within 1 year of post treatment duration.

Conclusion

Finite element analysis has proven itself an established numerical analysis with a paramount importance in not only aerospace, civil engineering and the automotive industry, but also in health care. The modelling and simulation steps save time and money for conducting the live experiment or clinical trial. Although finite element analysis is an accurate tool in assessing stress distribution, it is effective only for a given set of values or situation. But situation and biomechanical properties of living structures could vary from person to person. Hence, the obvious shortcomings should be kept in mind before any decision making procedure in experimental as well as clinical dentistry.

References

1. Geng JP, Tan KB, Liu GR (2001) Application of finite element analysis in implant dentistry: a review of the literature. *J Prosthet Dent* 85: 585-598.
2. Kazuo Tanne, Mamoru Sakuda, Charles J (1987) Three dimensional finite analyses for stress distribution in the periodontal tissue by orthodontic forces. *Am J Orthod Dentofacial Orthop* 92: 499-505.
3. Gallagher RH (1975) Finite element analysis: fundamentals. (4th Edn) Englewood Cliffs: Prentice-Hall.
4. Yettram AL, Wright KW, Pickard HM (1972) Finite element stress analysis of crowns of normal and restored teeth. *Acta Orthop Scand* 3:304-4.
5. Turner MJ, Clough RW, MactinandL HC (1956) Stiffness and deflection analysis of complex structure. *J aeronaut science* 23:805-823.
6. Yijunliu (2003) Introduction to FEM –CAE research laboratory university of Cincinnati U.S.A. Accessed.
7. Tanne K, Sakuda M, Burstone CJ (1987) Three-dimensional finite element analysis for stress in the periodontal tissue by orthodontic forces. *Am J Orthod Dentofacial Orthop* 92: 499-505.
8. Liang W, Rong Q, Lin J, Xu B (2009) Torque control of the maxillary incisors in lingual and labial orthodontics: a 3-dimensional finite element analysis. *Am J Orthod Dentofacial Orthop* 135: 316-322.
9. Sung SJ, Baik HS, Moon YS, Yu HS, Cho YS (2003) A comparative evaluation of different compensating curves in the lingual and labial techniques using 3D FEM. *Am J Orthod Dentofacial Orthop* 123: 441-450.
10. Tanne K, Matsubara S, Sakuda M (1995) Location of the centre of resistance for the nasomaxillary complex studied in a three-dimensional finite element model. *Br J Orthod* 22: 227-232.
11. Clarice Field, Ionut Ichim, Michael V Swain, Eugene Chan, M Ali Darendeliler, et al. (2009) Mechanical responses to orthodontic loading: a 3-dimensional finite element multi-tooth model. *Am J Orthod Dentofacial Orthop* 135:174-181.
12. Rudolph DJ, Willes PMG, Sameshima GT (2001) A finite element model of apical force distribution from orthodontic tooth movement. *Angle Orthod* 71: 127-131.
13. Ichim I, Schmidlin PR, Kieser JA, Swain MV (2007) Mechanical evaluation of cervical glass-ionomer restorations: 3D finite element study. *J Dent* 35: 28-35.
14. Ichim I, Li Q, Loughran J, Swain MV, Kieser J (2007) Restoration of non-carious cervical lesions Part I. Modelling of restorative fracture. *Dent Mater* 23: 1553-1561.
15. Ichim IP, Schmidlin PR, Li Q, Kieser JA, Swain MV (2007) Restoration of non-carious cervical lesions Part II. Restorative material selection to minimise fracture. *Dent Mater* 23: 1562-1569.
16. Asmussen E, Peutzfeldt A (2008) Class I and Class II restorations of resin composite: an FE analysis of the influence of modulus of elasticity on stresses generated by occlusal loading. *Dent Mater* 24: 600-605.
17. Subramaniam V, Indira R, Srinivasan MR, Shankar P (2007) Stress distribution in rotary nickel titanium instruments: A finite element analysis. *J Conserv Dent* 10:112-118.
18. Magne P (2007) Efficient 3D finite element analysis of dental restorative procedures using micro-CT data. *Dent Mater* 23: 539-548.
19. Hong J, Xia WW, Xiong HG (2003) Analysis of the effect on the stress of root

- canal wall by vertical and lateral condensation procedures. Shanghai Kou Qiang Yi Xue 12:359-361.
20. Rees JS (2002) The effect of variation in occlusal loading on the development of abfraction lesions: a finite element study. J Oral Rehabil 29: 188-193.
21. Rees JS, Hammadeh M (2004) Undermining of enamel as a mechanism of abfraction lesion formation: a finite element study. Eur J Oral Sci 112: 347-352.
22. Siegele D, Soltesz U (1989) Numerical investigations of the influence of implant shape on stress distribution in the jaw bone. Int J Oral Maxillofac Implants 4: 333-340.
23. Himmlová L, Dostálová T, Káčovský A, Konvicková S (2004) Influence of implant length and diameter on stress distribution: a finite element analysis. J Prosthet Dent 91: 20-25.
24. Mailath G, Stoiber B, Watzek G, Matejka M (1989) [Bone resorption at the entry of osseointegrated implants--a biomechanical phenomenon. Finite element study]. Z Stomatol 86: 207-216.
25. Dos Santos MB, Da Silva Neto JP, Consani RL, Mesquita MF (2011) Three-dimensional finite element analysis of stress distribution in peri-implant bone with relined dentures and different heights of healing caps. J Oral Rehabil 38: 691-696.
26. Haraldson T, Jemt T, Ståhlblad PA, Lekholm U (1988) Oral function in subjects with overdentures supported by osseointegrated implants. Scand J Dent Res 96: 235-242.

Citation: Mohammed SD, Desai H (2014) Basic Concepts of Finite Element Analysis and its Applications in Dentistry: An Overview. Oral Hyg Health 2: 156. doi: [10.4172/2332-0702.1000156](https://doi.org/10.4172/2332-0702.1000156)

Submit your next manuscript and get advantages of OMICS Group submissions

Unique features:

- User friendly/feasible website-translation of your paper to 50 world's leading languages
- Audio Version of published paper
- Digital articles to share and explore

Special features:

- 350 Open Access Journals
- 30,000 editorial team
- 21 days rapid review process
- Quality and quick editorial, review and publication processing
- Indexing at PubMed (partial), Scopus, EBSCO, Index Copernicus and Google Scholar etc
- Sharing Option: Social Networking Enabled
- Authors, Reviewers and Editors rewarded with online Scientific Credits
- Better discount for your subsequent articles

Submit your manuscript at: <http://www.omicsonline.org/submission/>