DOI: 10.21276/sjds.2016.3.10.2

Scholars Journal of Dental Sciences (SJDS) **ISSN 2394-496X (Online)**
Sch. J. Dent. Sci., 2016; 3(10):278-283 **ISSN 2394-4951 (Print)**
©Scholars Academic and Scientific Publisher
(An International Publisher for Academic and Scientific Resources)
www.saspublishers.com DOI: 10.36347/sjds.2016.v03i10.002

Original Research Article

Finite Element Analysis (FEA) as a Decisive Tool for Study of Force Distribution in Dentistry

Dr Ajay Saxena*, Dr. Manoj Chandak
Department of Conservative Dentistry, Sharad Pawar Dental College and Hospital, Sawangi, Maharashtra, India

***Corresponding author**
Dr Ajay Saxena
Email: drajay.saxena@yahoo.in

Abstract: The FEM is basically a numerical method for solving a differential equation. The geometric model of the structure is built and subdivided into small elements. 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 equations are then solved using a computer. In endodontics, using the FEM, the parameters of the geometry of the structure such as the post design, material properties, along with the magnitude and direction of the load can be changed easily in simulation, which is a significant advantage over experimental methods.
Keywords: endodontics, files, finite element analysis (FEA)

INTRODUCTION

Finite element analysis (FEA)

The finite element analysis is one of the 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 establishing the location, magnitude, and direction of an applied force, as it may also assign stress points that can be theoretically measured. 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 described 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 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 micro level 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].

Basic steps involved in carrying out FEA are;

- Pre-processing.
  - Conversion of geometric model into finite element model.
  - Assembly/Material Property data representation.
  - Defining the boundary conditions.
  - Loading Configuration.
- Processing.
- Post-processing.

Pre processing

Construction of the Geometric model

The purpose of the geometric modeling phase is to represent geometry in terms of points, lines, areas and volume. Complicated or smooth objects can be represented by geometrically simple pieces (Elements) [2]. This can be achieved by:- 3D – CT scanner: Usually done for modeling complex structures or living tissues.

For example craniofacial skeleton, maxilla or mandible3D – Laser scanner: Usually done for modeling inanimate objects.

For example: modeling of brackets.

A. Conversion of Geometric model to Finite Element Model:

Discretization is the process of dividing problem into several small elements, connected with nodes. All elements and nodes must be numbered so that a setup of matrix connectivity is established. This greatly affects the computing time. The elements could be one, two or three-dimensional and in various shapes.

It is essential that the elements are not overlapping but are connected only at the key points, which are termed *nodes*. The joining of elements at the nodes and eliminating duplicate nodes is termed as '*Meshing*'.

B. Assembly / Material Property data representation:

Equations are developed for each element in the FEM mesh and assembled into a set of global equations that model the properties of the entire system. Minimum material properties required are *poisons ratio and young's modulus.*

Table 1:

Material	**Young's modulus(kg/mm2)**	**Poisons ratio**
Tooth	2.6 x 103	0.15
Pdl	6.8 x 10-2	0.49
Alveolar bone	1.40 x 103	0.15

C. Defining the Boundary Conditions:

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.

D. Loading configuration:

Application of force at various points of geometry and its configuration.

Processing:

Solve the system of linear algebraic equation. The stresses are determined from the strains by Hooke's law. Strains are derived from the displacement functions within the element Combined with Hooke's law.

Post-Processing:

The output from the Finite Element Analysis is primarily in the *numerical form*. It usually consists of nodal values of the field variables and its derivatives. For example in solid mechanical problems, the output is nodal displacement and element stresses.

Graphic outputs and displays are usually more informative. The curves and contours of the field variable can be plotted and displayed. Also deformed shapes can be displayed and superimposed on unreformed shapes. The output is primarily in the form of color-coded maps. The quantitative analysis is determined by interpreting these maps [10].

Available Commercial FEM Software Packages: [10]

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

Application in Endodontics

1. Comparison of different conventional preparation techniques

Different preparation techniques like step back, crown down technique induce various canal morphologies. To assess the stress and fracture possibilities, Finite Element Analysis (FEA) has been used to estimate these preparation techniques.

Cheng *et al*. [11, 12] studied the stress distribution on endodontically-treated teeth with curved canals under various loads and determined the differences among three preparation techniques. They pointed out that the three techniques (crown-down, step-back and reverse-flaring techniques) displayed a similar stress distribution at the lower part of FEA model when occlusal loads and condensation loads were applied.

2. **Root canal morphology after root preparation**

To discuss the relative contribution of geometrical parameters after root canal preparation to tooth fracture, FEA models have been constructed to analyze the stress distribution of teeth quantitatively after root canal preparation. By far, the most repeatedly discussed morphological parameters affecting stress distribution on modified FEA models are dentin thickness, radius of canal curvature, canal cross-sectional shapes, canal irregularities, and canal taper.

a) **Dentin thickness**

As reported in many clinical and experimental studies, the dentin thickness is in inverse proportion to the fracture susceptibility. Using FEA, Ricks-Williamson *et al*. [13] found that the magnitude of generated radicular stresses was directly correlated with the simulated canal diameters. Wilcox *et al*. [14] found that root surface craze lines formed on roots where greater percentages of the canal wall were removed.

b) Radius of canal curvature

Knowing the severity of root canal curvature is essential to selecting the instrument and instrumentation technique. What is more, root canal curvature is also a determinant of the prognosis of instrumented teeth, taking a variety of reported complications (apical foramen, Creation of ledges, elbows, zips, perforations, instrument fracture, and vertical root fracture) into consideration. Several FEA models have been constructed to discuss the relationship between the radius of canal curvature and stress distribution.

Lertchirakarn *et al.* [15] indicated that circumferential tensile stresses were concentrated on the buccal and lingual surfaces of the canal wall, corresponding to areas of greatest canal wall curvature, suggesting that the fracture initiates from the site of greatest curvature of the root canal wall and propagates to the outer root surface. FEA models demonstrated that changing the outer root shape from round to oval, with a round canal, resulted in a smaller increase in maximum tensile stress than changing the inner canal shape from round to oval, leading to the conclusion that canal curvature is more important than external root morphology, in terms of stress concentration .

Canal cross-sectional shapes and canal irregularities

Irregular canal cross-sectional shapes and canal irregularities are not only unfavorable factors in the operation of root canal therapy, but also compromise the prognosis of the treated teeth. Canal cross-sectional shapes include circular shape, oval shape, flat shape, and ribbon-shape canals. Canal irregularities include multiple foramina, additional canals, fins, deltas, intercanal connections, loops, 'C-shaped' canals and accessory canals. FEA models are constructed to simulate their influence on stress distribution.

Canal Taper

The prepared canal diameter has also long been proved to influence the propensity for vertical root fractures. Generally, taper should be sufficient to permit deep penetration of spreaders or pluggers during filling, but should not be excessive to the point where procedural errors occur, and the root is unnecessarily weakened. Holcomb *et al*. [16] remarked that there must be a point at which increased canal width and taper begin to weaken the root.

FEA of Nickel-Titanium Rotary Instruments

In root canal therapy, instrument fracture is a potential consequence of canal instrumentation, especially when the instrument is bound at the tip. FEA has been employed to compare the stresses in a number of Nickel-Titanium Rotary instruments.

FEA in investigation of the mechanical properties of different NiTi rotary instruments

With the increased use of nickel-titanium (NiTi) rotary instruments for root canal therapy in endodontics, instrument fracture has become more and more prevalent. Extensive research has been carried out on the physical properties and mechanical characteristics of NiTi rotary instruments. Kim *et al*. [17] estimated the residual stress thereafter for some nickel-titanium rotary instruments (Profile, ProTaper, and ProTaper Universal) using a3-dimensional finite-element package. The simulation in the ProTaper design revealed that there was the greatest pull in the apical direction and the highest reaction torque from the root canal wall, while the least stress in occurred in the Profile design. Stresses in ProTaper were concentrated at the cutting edge, and the residual stress reached a level that was close to the critical stress for phase transformation in the material. The residual stress was highest in ProTaper (see below for the ProTaper Universal and Profile design).

Flexibility and fracture properties are determinant for the performance of NiTi rotary instruments. Kim *et al.* [17] evaluated geometrical differences between three NiTi instruments which affect the deformation and stress distributions under bending and torsional conditions. Profile, with a U-shaped cross section, showed the highest flexibility among the three file models. The ProTaper, which has a convex triangular cross-section, was the stiffest file model. In the ProTaper, more force is required to reach the same deflection as the other models, and more torque is needed than other models to achieve the same amount of rotation. Under torsion, all NiTi files showed the highest stress at their groove area. The Profile showed the highest von Mises stress value under the same torsional moment whereas the ProTaper Universal showed the highest value under the same rotational angle. Additionally, the assessment of the stress distributions of three NiTi instruments with various cross-sectional configurations under bending or torsional conditional showed that ProTaper has the lowest flexural rigidity of all if a U-shaped groove is incorporated in the middle of each side [18].

FEA in investigation of the parameters contributing to instrument failure

The radius and the position of the canal curvature

By FEA, Necchi [19] investigated rotary endodontic instruments and demonstrated the usefulness of the finite element method in simulating the mechanical behavior of these instruments during root canal preparation. The results indicated that the radius and the position of canal curvature are the most crucial parameters that determined the stress in the instrument, in that higher stress levels are produced by decreasing the radius and moving from the apical to the mid root position. The most demanding working conditions were observed in canals with sharp curves, especially in areas in which the instruments had larger diameters. To prevent the possible damage to instruments and fracture, it is suggested that the instruments should be discarded following their use in such canals.

Cross-sectional design of Nickel-titanium instruments

As NiTi instruments are generally perceived to have high fracture risk during use, new designs with lower fracture risks have been marketed. However, these design variations may also alter the forces distribution on a root during instrumentation and increase the potential dentinal defects that predispose to fracture. Previous study Kim *et al.* [18] has indicated that, in Nickel-titanium instruments with rectangle-based cross-section, higher stress differentials are created during the simulated canal shaping, and higher residual stress and plastic deformation occurs than in instruments with triangle-based cross sections. It has also been shown that different cross-sectional designs affect stress distribution in NiTi instruments during the bending, torsion and simulated shaping of a curved canal.

For three NiTi file designs ProFile (U-shaped cross-section and constant 6% tapered shaft), ProTaper Universal (convex triangular cross-section with notch and progressive taper shaft),and LightSpeed LSX (non cutting round shaft), the stress conditions during the rotary instrumentation in a curved root were also estimated [20]. ProTaper Universal introduced the highest stress concentration in the root dentin and also had the highest tensile and compressive principal strain components in the external root surface. ProTaper Universal had the biggest taper shaft and the calculated stress values from ProTaper Universal approached the strength properties of the dentin. Light Speed generated the lowest stresses. It can be concluded that the stiffer file designs created higher stress concentrations in the apical root dentin during the shaping of the curved canal, which increases the risk of dentinal defects that may lead to the apical root cracking.

Finite element analysis of the thermal distribution

Nowadays, warm vertical compaction is a widely used technique. However, the use of the technique may lead to an unconscious transmission of excessive heat to the surrounding tissues, which may cause irreversible injury to tissues. The use of peak temperature should be well defined.

FEA is the right choice for thermal distribution evaluation, on account of its detection not only for root surface temperature, but also internal distribution of heat. Er O *et al* [21] established a model of maxillary canine to determine the distribution and level of temperature .

When used with a 200°C initial setting, simulated in a process involving seven stages and lasting for 34 seconds, the maximum temperature lying in gutta-percha is 56.6°C and the periodontal ligament temperature is between 37.3°C and 39.7°C. And the maximum temperature rise was observed at the apical tip of the simulated heat source.

FEA and fracture strength test in root canal therapy

As there is a high occurrence of VRF in endodontically treated teeth, endodontic procedures have been considered as a frequent cause of VRF. One study, involving combined fracture strength testing and FEA to compare the preparation techniques of hand files and rotary Ni-Ti, demonstrated that the fracture load was almost identical, but the fracture pattern differed, and the FEA models correlated very well with the observed fracture pattern, demonstrating a reliable predictability for VRF [22].

Besides having an application in endodontic procedures, FEA and fracture strength testing have also been used to evaluate different kinds of posts and crowns. For endodontically-treated teeth, it is pivotal to strengthen the fracture resistance. One study revealed that endodontically treated premolars whose coronal hard tissue were severely damaged, obtained higher fracture resistance with the computer-aided design/computer-aided manufacturing ceramic endocrown restoration compared with classical crown configuration [23]. Another study showed that the combination of a fiber post and composite resin core with a full cast crown is most beneficial for the remaining tooth structure under the conditions of vertical and oblique loadings [24].

Application of Finite element analysis in other branches of dentistry

It is useful in dental implants; jaw bone surrounding the implant and biomechanical implant and jawbone interactions [25, 26].

1. In prosthodontics, FEM is useful for dental prosthetic designing. Designs for fiber framework for FPD have been extensively investigated using FE method and FE analysis [27]. FEM is also used to find out Stress distributions in adhesively cemented ceramic and resin-composite class II inlay restorations [28].

2. In Periodontics, this application is used to evaluate the stress distribution in periodontium based on the length, diameter, and geometry of dental implants. FEM can also be applied for prediction of face soft tissue deformations resulting from bone repositioning in maxillofacial surgery [29].

3. In Orthodontics, FEM is one of the very important applications in the field of bio-mechanics and cranio-facial biology. The orthodontist can precisely determine the effect of various biomechanical materials involved in tooth movement and can simulate the same in treatment planning [10].

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. 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. Keeping in mind these limitations, further FEA research should be supplemented with clinical evaluation.

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;35:695e699.
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;28:916e924.
3. Viceconti M, Zannoni C, Testi D. The multimod application framework: a rapid application development tool for computer aided medicine. Comput Methods Programs Biomed. 2007;85:138e151.
4. Van Staden RC, Guan H, Loo YC. Application of the finite element method in dental implant research. Comput Methods Biomech Biomed Engin. 2006;9:257e270.
5. Geng JP, Tan KB, Liu GR. Application of finite element analysis in implant dentistry: a review of the literature. J Prosthet Dent. 2001;85:585e598.
6. Lu S, Li T, Zhang Y. Biomechanical optimization of the diameter of distraction screw in distraction implant by three dimensional finite element analysis. Comput Biol Med. 2013;43:1949e1954.
7. Pessoa RS, Muraru L, Junior EM. 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;12:219e234.
8. Doblare M, Garcıa JM, Gomez MJ. Modelling bone tissue fracture and healing: a review. Eng Fract Mech. 2004;71:1809e1840.
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;112:1238e1245.
10. Konda P, Tarannum SA. Journal of Pharmaceutical and Biomedical Sciences (JPBMS). 2009;16:16.
11. Cheng R, Zhou XD, Liu Z, Hu T. Development of a Finite Element Analysis Model With Curved Canal and Stress Analysis. Journal of Endodontics. 2007;33:727-731.
12. Cheng R, Zhou XD, Liu Z, Yang H, Gao QH, Hu T. Finite element analysis of the effects of three preparation techniques on stresses within roots having curved canals. International Endodontic Journal. 2009;42:220-226.
13. Ricks-Williamson LJ, Fotos PG, Goel VK, Spivey JD, Rivera EM, Khera SC. A three-dimensional finite-element stress analysis of an endodontically prepared maxillary central incisor. Journal of Endodontics. 1995;21:362-327.
14. Wilcox LR, Roskelley C, Sutton T. The relationship of root canal enlargement to finger-spreader induced vertical root fracture. Journal of Endodontics. 1997;23:533-534.
15. Lertchirakarn V, Palamara JE, Messer HH. Patterns of vertical root fracture: factors affecting stress distribution in the root canal. Journal of Endodontics. 2003;29:523-528
16. Honlcomb JQ, Pitts DL, Nicholls JL (1987) Further investigation of spreader loads required to cause vertical root fracture during lateral condensation. Journal of endodontics 13:277-284
17. Kim HC, Cheung GS, Lee CJ, Kim BM, Park JK, Kang SI. Comparison of Forces Generated During Root Canal Shaping and Residual Stresses of Three Nickel–Titanium Rotary Files by Using a Three-

Dimensional Finite-element Analysis. Journal of Endodontics. 2008;34:743-747.
18. Kim HC, Kim HJ, Lee CJ, Kim BM, Park JK, Versluis A. Mechanical response of nickel-titanium instruments with different cross-sectional designs during shaping of simulated curved canals. International Endodontic Journal. 2009;42:593-602.
19. Necchi S, Taschieri S, Petrini L, Migliavacca F. Mechanical behavior of nickel-titanium rotary endodontic instruments in simulated clinical conditions: a computational study. International Endodontic Journal. 2008;41:939-949.
20. Kim HC, Lee MH, Yum J, Versluis A, Lee CJ, Kim BM. Potential Relationship between Design of Nickel-Titanium Rotary Instruments and Vertical Root Fracture. Journal of Endodontics (ARTICLE IN PRESS). 2010.
21. Er O, Yaman SD, Hasan M Finite element analysis of the effects of thermal obturation in maxillary canine teeth. Oral Medicine, Oral Patholology, Oral Radiology and Endodontics, 2007;104:277-286.
22. Sathorn C, Palamara JEA, Messer HH. A comparison of the effects of two canal preparation techniques on root fracture susceptibility and fracture pattern. Journal of Endodontics. 2005;31:283-287
23. Lin CL, Chang YH, Chang CY, Pai CA, Huang SF. Finite element and Weibull analyses to estimate failure risks in the ceramic endocrown and classical crown for endodontically treated maxillary premolar. European Journal of Oral Sciences. 2010;118:87-93.
24. Hayashi M, Takahashi Y, Imazato S, Ebisu S. Fracture resistance of pulpless teeth restored with post-cores and crowns. Dental Materials. 2006;22:477-485.
25. Vanstaden RC, Guan H, Loo YC. Application of finite element method in dental implant research. Compute Methods Biomech Biomed Engin. 2006;9(4):257- 70.
26. Ruppin J, Popovic A, Strauss M, Spüntrup E, Steiner A, Stoll C. Evaluation of the accuracy of 3 different Computer Aided surgery systems in dental implantology. Clinical oral implants research. 2008;19(7):709-16.
27. Shinya A, Yokoyama D. Finite element analysis for dental prosthetic design. Accessed on 30-01-2012 Available from URL: http://www.intechopen.com/articles/show/title/finite-element-analysis-for-dental-prosthetic-design.
28. Stress distributions in adhesively cemented ceramic and resin-composite class II inlay restorations: a 3DFEA study. Dent Mater. 2004;(20): 862-872.
29. Anitua E, Tapia R, Luzuriaga F, Orive G. Influence of implant length, diameter and geometry on stress distribution: a finite element analysis. Int J Periodontics Restorative Dent. 2010;(30):89-95.