Finite Element Method – An Effective Research Tool for Dentistry

Preeth Shetty * / Amitha M Hegde** / Kavita Rai***

Stress analysis of dental structures has been a topic of interest in recent years with the objective of determination of stresses in the tooth and it's supporting structures and improvement of the mechanical strength of these structures. The purpose of this article is to give an insight of the finite element analysis which has totally overtaken other experimental analysis due to its ability to model even the most complex of geometries with is immensely flexible and adaptable nature. Finite Element Analysis (FEA) is a computer-based numerical technique for calculating the strength and behavior of structures. It can be used to analyze either small or large-scale deflection under loading or applied displacement. However it is extremely expensive and can be used only with the help of an expert engineer who has mastered this technique. Still this methodology of stress analysis has become extremely popular in dentistry as various properties of dental tissues and materials can be just fed into it and with the ease and accuracy the analysis is done is just remarkable.
Keywords: *Finite element method (FEM), Stress distribution, Finite element analysis (FEA)*
J Clin Pediatr Dent 34(3): 281–286, 2010

INTRODUCTION

Dentistry has gone through some fundamental changes over the last 50 years. Just think what the possibilities are for the next 50! If you look back on the tools of our trade over the last 20 years, do you remember the speed that once—familiar techniques and materials have appeared and then been superseded? Now try to imagine what we are to face in the next 10! For some it will be the most daunting prospect of their careers, whilst for a proportion of dentists throughout the world; it will be the most exciting time of all. The emerging fields of nanoscale science, engineering and technology—the ability to work at the molecular level, atom by atom to create large structures with fundamentally new properties and functions – are leading to unprecedented understanding and control over the basic building blocks and properties of natural and manmade things.

One such field in science which has got immense popularity is Finite element method (FEM) or Finite element analysis (FEA). FEA has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material and allowing designers to see all of the theoretical stresses within. This method of product design and testing is far superior to the manufacturing costs which would accrue if each sample was actually built and tested. Finite element analysis has been slowly but steadily found wide spread popularity in the fields of medicine and dentistry. Especially in dentistry; where this tool of research methodology has been used to understand the behaviour of various materials.

WHAT IS FINITE ELEMENT ANALYSIS?

It is a numerical technique to obtain approximate solutions to a wide variety of engineering problems where the variables are related by means of algebraic, differential and integral equations. The concept of FEA originated during 1940s with the advances in aeronautical engineering. It was introduced to study stresses in complex airframe structures. The mathematical foundation for this method was laid down during 1940s and 1950s. FEA was first developed in 1943 by R. Courant. Clough coined the term 'Finite element' in 1960. FEA consists of a computer model of a material or design that is stressed and analyzed for specific results. It is used in new product design, and existing product refinement.[1]

There are generally two types of analysis that are used in industry: 2-D modeling, and 3-D modeling. While 2-D modeling shows simplicity and allows the analysis to be run on a

* Preeth Shetty ,Reader, Department of Pedodontics and Preventive Children Dentistry, A.B. Shetty Memorial Institute of Dental Sciences
** Amitha M Hegde, Professor and Head of the Department, Department of Pedodontics and Preventive Children Dentistry, A.B. Shetty Memorial Institute of Dental Sciences
*** Kavita Rai, professor, Department Of Pedodontics and Preventive Children Dentistry, A B. Shetty Memorial Institute Of Dental Sciences

Send all correspondence to: Dr. Preeth Shetty, Reader, Department of Pedodontics and Preventive Children Dentistry, A.B. Shetty Memorial Institute of Dental Sciences, Derlakatte, Mangalore-575018, Karnataka, India.

Fax no: 0824-2204572.

Email: amipedo@yahoo.co.in
preeth_s@rediffmail.com

relatively normal computer, it tends to yield less accurate results. 3-D modeling, however, produces more accurate results. Linear systems are far less complex and generally do not take into account plastic deformation. Non-linear systems do account for plastic deformation, and many also are capable of testing a material all the way to fracture. Its applicability increased as the computers became readily available for making complex computations.

Out of various FEA applications, the analysis most relevant to us is the structural stress analysis – e.g. temperature distribution in structures and fluid flow analysis e.g. velocity, pressure and concentration in fluid flows. As the name indicates, this method schematically divides an object under study called a continuum or a domain, into finite number of smaller sub domains called elements. This process is called as discretization.

Elements could be one (straight lines), two (triangles, quadrilaterals) or three dimensional (pyramid or a brick like shape) and in various shapes. Elements are not overlapping, but are connected only at the key points which are termed nodes. Joining of elements at the nodes and eliminating duplicate nodes is termed as 'Meshing'. Thus a mesh of all the elements connected at the nodes represents the object under study. The sum of deformation of all the elements is the deformation of the entire structure. Thus stress strain behavior of even a geometrically irregular shape can be calculated.[2]

COMPONENTS OF THE FINITE ELEMENT METHOD

Broadly stated, the components of the finite element method are:

1. Finite Element Modelling
2. Finite Element Analysis
   Pre-processing stage:
   Solution stage
   Post-processing stage.

PRE-PROCESSOR**:** The preprocessor stage involves the following sections:

- *Specifying the title*, i.e. the name of the problem.
- *Setting the preferences*, this is the type of filtering to be used, e.g. structural, fluid, thermal or electromagnetic
- *Defining the element type*, this may be 2d or 3d in the structural element types, (Figure No. 1) and there are

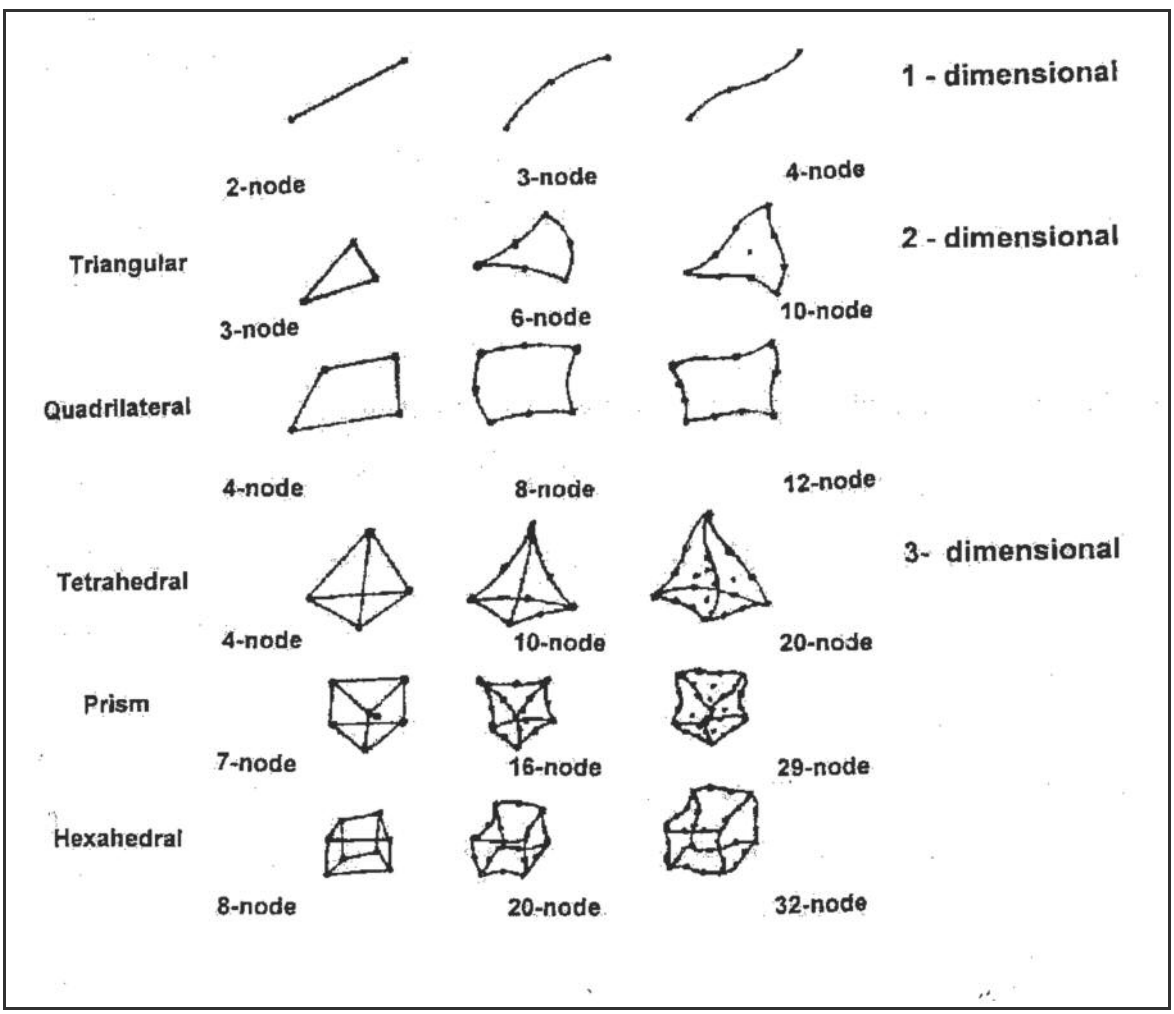

Figure No.1. Different types of elements & the no. of nodes they could have.

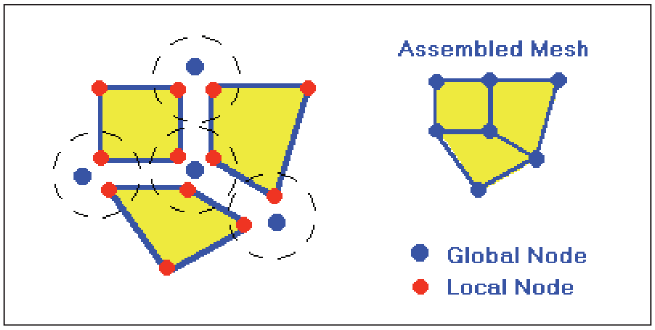

Figure No. 2. Assembly of elements

many types then to choose from as mentioned in the introductory paragraph to the FEM method. This is possibly the most crucial part of an analysis if a highly accurate set of results is required (Figure No. 2).

- *Defining the material properties*, i.e. the Young's modulus, Poisson's ratio, the density, and if applicable, the coefficients of expansion, friction, thermal conductivity, damping effect, specific heat etc.
- *Creating the model in appropriate dimensions*. This is where the actual model is drawn in 2D or 3D space in the appropriate units (M, mm, in, etc.).
- *Defining the mesh density*. This may be done by manually defining the number of elements along the lines of the model, thus customizing the number of elements. In complex cases, the mesh density may be generated by specifying the element edge length, and hence the mesher, meshes the model automatically on the command using the edge length specified.
- *Loading and boundary conditions* are applied to the model. The boundary conditions are the second most critical stage of the analysis (element type is first).

Once the geometry is created it is transferred into a finite element model by the processor. Mesh generation is used to describe this procedure. Developing the mesh is usually the most time-consuming task in FEA.

SOLUTION**:** The solution of the problem is done automatically by executing the command. The package then proceeds to form the element-stiffness matrix for the problem, followed by solving for the matrix and then updating the displacement value for each node within the component or continuum.

POST-PROCESSOR**:** After a finite element model has been prepared and checked, boundary conditions have been applied, and the model has been solved, it is time to investigate the results of the analysis. This activity is known as the post-processing phase of the finite element method.[1-4]

REQUIREMENTS OF FINITE ELEMENT ANALYSIS

Finite Element Analysis is done principally with commercially purchased softwares like NISA, ANSYS and NASTRAN-PATRAN to name a few. These commercial software programs are expensive depending on their extensive capabilities—plastic deformation, and specialized work

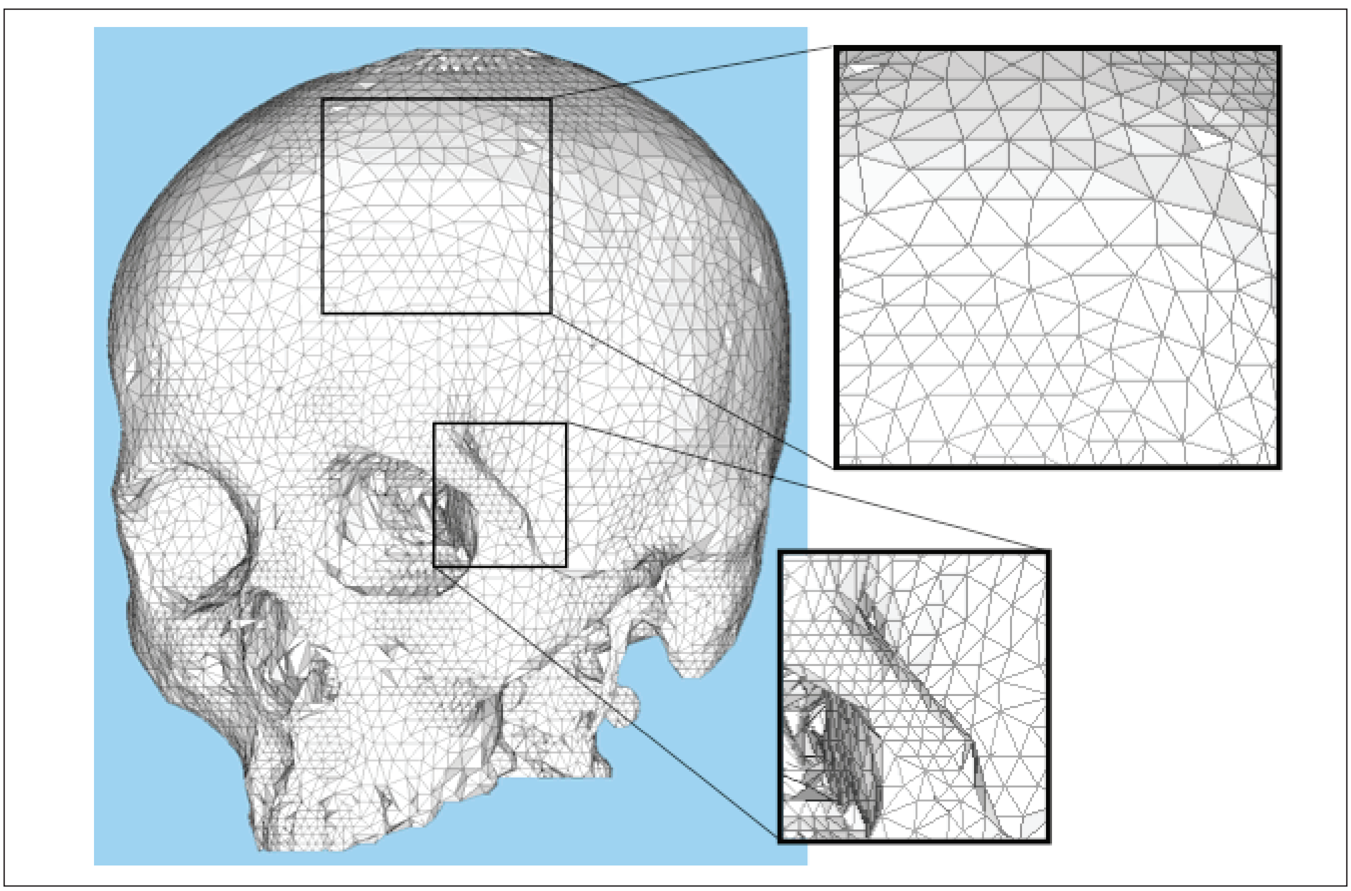

Figure no. 3. Finite element model of a human skull

Figure No. 4. Finite element analysis for crash testing

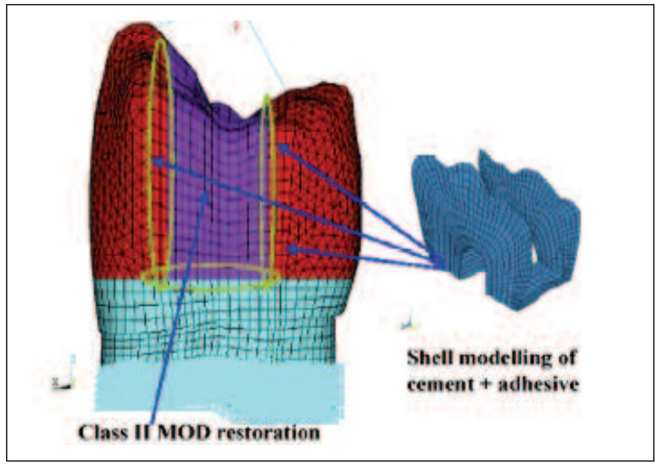

Figure No. 5. FE model of Class II MOD of upper premolar

such as metal forming or crash and impact analysis. Finite element packages may include pre-processors that can be used to create the geometry of the structure, or to import it from CAD files generated by other software. The FEA software includes modules to create the element mesh, to analyze the defined problem, and to review the results of the analysis. Output can be in printed form, and plotted results such as contour maps of stress, deflection plots, and graphs of output parameters

APPLICATION OF FINITE ELEMENT ANALYSIS

The finite element method is being used in virtually every engineering discipline. Aerospace, automotive, biomedical, geotechnical, electrical, hydraulic, and nuclear engineering applications have become standard objects for finite element analysis.[5] In addition, it is not only used for analyzing classical static structural problems, but also for such diverse areas as mass transport, heat transfer, dynamics, stability, radiation problems and crash testing by various automobile company[6] (figure no. 4).

Experimental stress analysis of dental structures has been a topic of interest during the later half of this century. The object of such research was the determination of stress distribution and improvement of the mechanical strength of these structures.[7, 8] Stresses in dental structures have been studied by various techniques e.g. brittle coatings analysis, strain gauges, holography, two and three dimensional photoelasticity, finite element analysis and other numerical methods. Stress analysis studies of inlays, crowns, bases[9] supporting restorations, fixed bridges, complete dentures, partial dentures,[10] endodontic posts[11] and implants[12] have been reported, as well as studies of teeth, bone, and oral tissues (figure no. 5). Most of the stress analysis of dental structures was carried out using the photoelastic technique. The advantages of using photoelastic study are that it can quantify stresses throughout a three-dimensional structure and determine stress gradients. However, it requires a bifringent material is more difficult with complex geometries.[8]

The use of this method in dental structures was started in 1968 when Ledley and Huang developed a linear model of the tooth based on experimental data and on linear displacement force analysis. The one shortcoming of their study was that they considered the tooth to be homogeneous structure. In reality the human tooth is highly inhomogeneous since the elastic modulus of the enamel outer surface of the tooth is about three times that of the inner dentin material.[13] The major contribution was made by W. Farah (1972), Thresher R.W (1973) and Yettram A.L (1976) who modeled a tooth and studied the stresses in a tooth structure using a finite element method. After this there have been numerous studies on assessing the stresses of various restorative materials, endodontic & surgical techniques, implants or orthodontic forces on tooth and its supporting structures.[14-20]

The FEM in dentistry recently focused on simulation of realistic intra-oral conditions such as the nonlinear stress–strain relationship in the periodontal tissues and the contact phenomena in teeth, which could hardly be solved by the linear static model.[21] The nonlinear FE analysis has become an increasingly powerful approach to predict stress and strain within structures in a realistic situation that cannot be solved by conventional linear static models. The nonlinear simulation of the PDL properties enhances a precise estimation of the stress and strain with wide range of tooth movement. The determination of the elastic, plastic, and viscoelastic material properties of a target material often requires mechanical testing prior to FEM analyses. Once the material properties are known, simulation of the material and prostheses of complex geometries and varied dimensions are feasible. To predict the failure risk of a bonded tooth– restoration interface, it is essential to assess the normal and shear interfacial stresses.[22, 23] The use of nonlinear FEM in dental applications was reported in literature recently; however, the validity and reliability have not been sufficiently established. Further development of the nonlinear FEM solutions is encouraged to gain a wide range of mechanical solutions that would be beneficial for dental and oral health science.

CONCLUSION

From the point of view of the numerous researches taking place in the field of dentistry in the recent times especially in the field of biomechanics and bioengineering, finite element analysis has proved to be the most adaptable, accurate, easy and less time consuming process as compared to the other experimental analysis. Inspite of certain limitations out of which the most important is the cost factor and requirement of an expert to operate the analysis, this technique has taken the field of dentistry to great heights and provided results which couldn't have been possible with any other technique. It has provided clinicians with useful information to achieve higher degree of success and satisfaction to the patients.

REFERENCES

1. J.N.Reddy. An introduction to the Finite Element method. 3rd Ed, Mc Graw Hill Education, 2005
2. Huebner K.H., Thornton E.A. The Finite Element Method for Engineers, Second Edition, John Wiley and Sons, 1982.
3. J.H. Keyak, M.G. Fourkas, J.M. Meager and H.B. Skinner: Validation of an automated method of three-dimensional finite element modelling of bone, J Biomed *Eng,* 15; 505–509, 1993.
4. Erik G Thompson. Introduction to the Finite Element Method: Theory, Programming and Applications. Wiley & sons, 2005.
5. Chandrupatla T.R., Belegundu A.D. Introduction to finite element engineering. 2nd Ed. Prentice Hall of India Pvt., Ltd., New Delhi: 1–7, 209–245 & 412–426, 1997.
6. S.W. Kirkpatrick, J.W. Simons, and T.H. Antoun, "Development and Validation of High Fidelity Vehicle Crash Simulation Models," International Journal of Crashworthiness, 602–611, 1998.
7. Caputo A.A, Standlee. Biomechanics in clinical dentistry. Chicago, Quintessence 19–23, 1987.
8. Craig R.G, Powers J.M. Restorative dental materials. 11th ed. St. Louis, Missouri. Mosby, Inc. 110–111, 2002.
9. Pietro Ausielloa,, Sandro Rengoa, Carel L. Davidsonb, David C.Watts, Stress distributions in adhesively cemented ceramic and resin-composite Class II inlay restorations: a 3D-FEA study. Dental Materials, 20: 862–872, 2004.
10. Eto M, Wakabayashi N, Ohyama T. Finite element analysis of deflections in major connectors for maxillary RPDs Int J Prosthodont, 15(5): 433–8, 2002.
11. Eskitascioglu G, Belli S, Kalkan M. Evaluation of two different post core systems using two different methods (fracture strength test and a finite element stress analysis. Journal of Endodontics, 28(9): 629–33, 2002.
12. Kohal RJ, Papavasiliou G, Kamposiora P, Tripodakis A, Strub JR. Three – Dimensional computerized stress analysis of commercially pure Titanium and Yttrium – partially stabilized Zirconia implants International Journal of Prosthodontics, 15(2): 189–94, 2002.
13. Ledley R.H, Hung H.K. "Linear model of tooth displacement by applied forces." J Dent Res, 48: 32–37, 1968.
14. Rees J.S, Jacobsen P.H "Modeling the effects enamel anisotropy with the finite element method" J Oral Rehabil, 22: 451–454, 1995.
15. Toparli M, Gokay N, Aksoy T. "Analysis of a restored maxillary second premolar tooth by three dimensional finite element method. J Oral Rehab, 26: 157–164, 1999.
16. Toparli M, Gokay N, Aksoy T. "An investigation of temperature and stress distribution on a restored maxillary second premolar tooth using a three- dimensional finite element method" J Oral Rehabil, 27: 1077–1081, 2000.
17. Zhang L, Zhou Y, Meng W "A three-dimensional finite element analysis of the correlation between lengths and diameters of the implants of fixed bridges with proper stress distribution. Hua Xi Kou Qiang Yi Xue Za Zhi, 18(4): 229–31, 2000.
18. N.Inou, S.Suzuki, K.Maki and S.Ujihashi: An Automated Modeling Method of a Bone Based on the Xray CT Data(Generation of a finite element model by use of Delaunay triangulation) J. Japan Soc. Mech. Eng. SeriesC, 68: 669, 1481–1486, 2002.
19. Ukon S, Moroi H, Okimoto K, Fujita M, Ishikawa M, Terada Y, Satoh H. Influence of different elastic moduli of dowel and core on stress distribution in root. Dent Mater J, 19(1): 50–64, 2000.
20. Cheng R, Zhou XD, Liu Z, Hu T. Development of a finite element analysis model with curved canal and stress analysis. J Endod, 33(6): 727–31, 2007.
21. N. Wakabayashi, M. Ona, T. Suzuki, Y. Igarashi. Nonlinear finite element analyses: Advances and challenges in dental applications. Journal of Dentistry, 36: 463–471, 2008.
22. Ichim I, Li Q, Loughran J, Swain MV, Kieser J. Restoration of non-carious cervical lesions. Part I. Modelling of restorative fracture. Dental Materials, 23: 1553–61, 2007.
23. Ichim IP, Schmidlin PR, Li Q, Kieser JA, Swain MV. Restoration of non-carious cervical lesions. Part II. Restorative material selection to minimize fracture. Dental Materials, 23: 1562–9, 2007.