Finite element analysis: A Boon To Dental Research

G Vasudeva

Citation

G Vasudeva. *Finite element analysis: A Boon To Dental Research*. The Internet Journal of Dental Science. 2008 Volume 6 Number 2.

Abstract

Stress analysis of dental structures has been a topic of interest in recent years with the objective of determination of stresses in the dental 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 overshadowed 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 is (FEA) is a computer-based numerical technique for calculating the strength and behavior of structures. It can be used to calculate deflection, stress, vibration, buckling behavior and many other phenomena. 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.

INTRODUCTION

For a successful restoration the mechanical properties of dental materials must withstand the stresses and strains caused by the repetitive forces of mastication. Along with this the design of dental restorations is particularly important if the best advantage of a material is to be taken. The necessary designs are those that do not result in stresses or strains that exceed the strength properties of a material under clinical conditions [1] .

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 [1,2] .

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 supporting restorations, fixed bridges, complete dentures, partial dentures, endodontic posts and implants have been reported, as well as studies of teeth, bone, and oral tissues. 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 birefringent material is more difficult with complex geometries [2] .

A more recent method of stress analysis, generally developed in 1956 in the aircraft industry was the finite element method. This technique was used widely only in aerospace engineering at first but slowly due to the flexibility of the method to model any complex geometries and provide instant results it made its presence felt in dentistry in early 1970's [3] .

WHAT IS FINITE ELEMENT ANALYSIS?

Finite Element Analysis is (FEA) is a computer-based numerical technique for calculating the strength and behavior of engineering structures. It can be used to calculate deflection, stress, vibration, buckling behavior and many other phenomena. It can be used to analyze either small or large-scale deflection under loading or applied displacement. It can analyze elastic deformation, or “permanently bent out of shape” plastic deformation. In the finite element method, a structure is broken down into many small simple blocks or elements [4] . The behavior of an individual element can be described with a relatively simple set of equations. Just as the set of elements would be joined together to build the whole structure, the equations describing the behaviors of the individual elements are joined into an extremely large set of equations that describe the behavior of the whole structure. The stresses will be compared to allowed values of stress for the materials to be used, to see if the structure is strong enough. The information needed to calculate the stresses and displacements in finite element model is (1) the total number of nodal points and elements, (2) a numbering system for identifying each nodal point and element, (3) the elastic moduli and poisson's ratio for the materials as associated with each element, (4) the coordinates of each nodal point, (5) the type of boundary strains and (6) the evaluation of the forces applied to the external nodes [4] .

The term “finite element” distinguishes the technique from the use of infinitesimal “differential elements” used in calculus, differential equations, and partial differential equations. The method is also distinguished from finite difference equations, for which although the steps into which space is divided are finite in size, there is little freedom in the shapes that the discreet steps can take. Finite element analysis is a way to deal with structures that are more complex than can be dealt with analytically using partial differential equations. FEA deals with complex boundaries better than finite difference equations will, and gives answers to “real world” structural problems. Finite Element Analysis makes it possible to evaluate a detailed and complex structure, in a computer, during the planning of the structure. The demonstration in the computer of the adequate strength of the structure and the possibility of improving the design during planning can justify the cost of this analysis work. [4,5]

In the absence of Finite Element Analysis (or other numerical analysis), development of structures must be based on hand calculations only. For complex structures, the simplifying assumptions required to make any calculations possible can lead to a conservative and heavy design. A considerable factor of ignorance can remain as to whether the structure will be adequate for all design loads. Significant changes in designs involve risk. Designs will require prototypes to be built and field tested. The field tests may involve expensive strain gauging to evaluate strength and deformation. With Finite Element Analysis, the weight of a design can be minimized, and there can be a reduction in the number of prototypes built. Field testing will be used to establish loading on structures, which can be used to do future design improvements via Finite Element Analysis. [5,6]

REQUIREMENTS FOR DOING FINITE ELEMENT ANALYSIS

Finite Element Analysis is done principally with commercially purchased software. These commercial software programs can cost roughly $1,000 to $50,000 or more. Software at the high end of the price scale features extensive capabilities -- plastic deformation, and specialized work 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. [4,5]

The choice of a computer is based principally on the kind of structure to be analyzed, the detail required of the model, the type of analysis (e.g. linear versus nonlinear), the economics of the value of timely analysis, and the analyst's salary and overhead. An analysis can take minutes, hours, or days. Extremely complex models will be run on supercomputers. Usually, “Faster and Bigger is Better!” “The fundamental theorem of finite element analysis.” Color is usually desirable, although you can sometimes live without it by doing gray-scale plots where the palette for contour maps has been changed to a progressive gray-scale. Higher prices will let you consider computers with more memory, larger hard drives, and one or more high-speed processors. Depending on the complexity of the structures to be studied and the volume of manufacturing, the expense for FEA hardware can be small in comparison with the savings in weight and construction cost that can result from design improvements, and speed of analysis. The expense can be very small in comparison to the cost of a failure. [5]

The background of a finite element analyst includes an understanding of engineering mechanics (strength of materials & solid mechanics) as well as the fundamentals of the theory underlying the finite element method. The analyst must appreciate the basics of numerical methods. An engineering degree is typical, though not an absolute requirement. Use of a particular finite element program requires familiarity with the interface of the program in order to create and load the models, and to review the results [6] .

APPLICATION OF FINITE ELEMENT ANALYSIS IN DENTISTRY

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. [7]

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.

Farah and Craig (1973) [8] analyzed a restored axisymmetric first molar and compared different types of finish lines and their effect on the marginal configuration with the help of a finite element method. They found that the chamfer geometry exhibit exhibited the most uniform stress distribution. Thresher R.W and Saito (1973) [3] also did a stress analysis of a homogeneous and a non-homogeneous human tooth using a finite element method. They concluded that the enamel portion of the tooth carries the major portion of the load and the load is transferred into surrounding bone structure remarkably high on the tooth root.

Yettram A.L (1978) [9] studied a class I amalgam restoration subjected to setting and thermal expansion. In 1983 Vree et al compared the photoelastic and finite element stress analysis in restored tooth structures. They found that the sensitivity of the finite element method towards variation of a number of relevant parameters was more but the results depend on the description of the outer surface of the tooth structure, the amount friction coefficient between various parts and the amount of Poisson's ratio.

Rubin C et al (1983) [10] developed a three dimensional model for the purpose of analyzing the stress distribution in a human mandibular right first molar. The model took in account the non-symmetric geometry and the material unhomogeneities of the tooth. They compared it with the existing two-dimensional analyses. They found that there was no effect of pulp on the stress analysis thus it could be modeled as a void.

The advantages of using a three dimensional analysis over a two dimensional analysis are: The human tooth is neither planner nor symmetrical. It is highly irregular in shape; the loading on the tooth is neither in a state of plane stress nor is symmetrical. Such assumptions would be expected to produce considerable errors in the calculations of the stress distribution within the tooth and the distribution of the various materials of which tooth is composed does not blend itself to two-dimensional modeling [10] .

Vijay K Goel et al (1991) [11] analyzed the stresses at the dentinoenamel junction of human teeth using a finite element method. They observed that normal and shear stresses were markedly affected by the contour of dentinoenamel junction and concluded that mechanical interlocking between the enamel and dentin in the cervical region were weaker than in other regions of dentinoenamel junction.

Rees JS and Jacobsen PH (1995) [12] used a finite element method to model an in vitro tooth loading system. They modeled dentine as a linear elastic and isotropic material, while the enamel was modeled as an anisotropic material to take account of the biological variation in prism angulations and found that a dentine modulus of 15 Gpa and an enamel modulus of 80 Gpa in the principal prism direction. Again in (1998) Rees JS [13] investigated the effect that an occlusal restoration has on the stress profile in the cervical region of a lower second premolar using two-dimensional finite element stress analysis and concluded that the weakening effect of cavity preparation may contribute to the development of non-carious cervical tooth loss.

Toparli M et al (1999) [14] investigated the stress distribution which was the resultant of the stresses which come from the mastication force and those resulting from the contraction and expansion of restorative materials. They carried out calculations using the finite element method and found that the modulus of elasticity was four times that of composite resin and the tensile stresses were larger in enamel than those when composite resin had been used.

Recently polymerization shrinkage of composites, its effect and various temperature changes can be incorporated if the coefficient of thermal expansion of the material is known and can be evaluated with the advancements in the finite element methodology. This eliminated the use of tedious thermocycling as done in the experimental analysis. Some studies were done in relation to this. Rees JS and Jacobsen PH (2000) [15] studied the partial failure around the tooth-composite interface of a class V restoration due to the effects of polymerization shrinkage. They observed that partial failure resulted in a 4-6-fold increase in peak tensile and shear forces compared to a tooth with a fully intact cavity wall interface also the peak stresses were greater than the known bond strengths of composite to dentine. Toparli M et al (2000) [16] investigated the effect of temperature and stress distribution on a restored maxillary second premolar using a three dimensional finite element method.

Ausiello P et al (2001) [17] studied the cusp movements in a human upper premolar, restored with adhesive resin-based composites using a 3D finite element analysis and concluded that premature failure due to stress arising from polymerization shrinkage and occlusal loading can be prevented by proper selection and combination of materials.

Due to the flexibility and adaptability to model even the most complex of geometries it has been used in almost all the fields of dentistry and quiet expensively in prosthodontics and endodontics and implantology. Sato Y et al (2003) [18] studied the effect occlusal rest size and shape on yield strength using the finite element analysis and concluded that the basic principles for optimizing the size and shape of occlusal rests should be followed. Han P et al (2001) [19] did the 3D finite element analysis of stress distributions in supporting tissues of clasp-type partial dentures of transferring occlusion force and found that the mastication function can be satisfactorily recovered, and the reasonable rest recess angle for the clasp-type partial denture of transferring occlusion force was about 75 degrees.

Zhang L et al (2000) [20] did a three-dimensional finite element analysis of the correlation between lengths and diameters of the implants of fixed bridges with proper stress distribution and found that the proper stress distribution of implants with different implant sizes can be achieved by adjusting diameters or/and lengths of implants.

Pierrisnard et al (2002) [21] compare the effect of different corono-radicular reconstruction methods on stress transmission to dental tissues with the help of FEA. It was confirmed that all simulated reconstructed teeth were more subject to stress in the cervical region. The absence of a cervical ferrule was found to be a determining negative factor, giving rise to considerably higher stress levels. Ukon S et al (2000) [22] investigated the influences of elastic moduli of the dowel-core combination on the stress distribution in the root by the use of 2-dimensional finite element analysis and suggested that at the marginal region the effect of the core material contributed more than 86% to the peak stress value for both loadings, and the post material affected at most about 11% of the bending resistance.

Recently complex root canals with roots have been modeled to study the stress pattern for posts, Obturation stresses and stress induced by nickel titanium instruments. Cheng et al (2007) [23] established a model of a curved canal by finite element analysis (FEA). To develop a repeatable and comparable model, simulated curved canals with uniform shape were selected as prototype canals, and a suitable extracted single-root tooth was chosen as the outline. The study has established a standardized FEA model with a curved canal. Consequently, this model may be applied for future estimations in endodontic procedures for a curved canal.

The most important conclusions of various researches that compared the photoelastic study with finite element analysis show that (1) the numerical results of the FEM stress research tally with those of the photoelastic study (2) calculations of stresses in restored tooth structures with the help of the FEM are rather insensitive to inaccuracies in the description of the outer surface of the tooth structure, the amount of friction coefficient between various parts and the amount of Poisson's ratio.

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 on 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.

CORRESPONDENCE TO

Dr. Gaurav Vasudeva 59, Sector – 8, Panchkula – 134109, INDIA e-mail – dr_vasu@hotmail.com Phone - +91-172-2562569 (O) +91-9915025809 (M)

References

1. Caputo A.A, Standlee. Biomechanics in clinical dentistry. Chicago, Quintessence 1987: 19-23.
2. Craig R.G, Powers J.M. Restorative dental materials. 11th ed. St. Louis, Missouri. Mosby, Inc.2002: 110-111.
3. Thresher R.W, Saito G.E. "The stress analysis of human teeth" J Biomech,1973 ;6:443-449
4. NISA-II User's Manual; Feb. 1997, Version 7.0;

Engineering Mechanics Research Corporation, Troy, Michigan.
5. Display-IV User's Manual; Mar. 2001, Version 10.5; Engineering
Mechanics Research Corporation, Troy, Michigan.
6. Aggarwal S.K, Nayak C, Shankar Lal. Prediction of Stresses by Finite Element Method; 1977 Edition; Mechanical & Industrial Department, University of Roorkee, Roorkee.
7. Ledley R.H, Hung H.K. "Linear model of tooth displacement by applied forces." J Dent Res,1968 ; 48:32-37.
8. Farah J.W, CRAIG R.G. "Element Stress Analysis of a Axisymmeric Molar" J Dent Res, 1973: 53:859-864
9. Yettram A.L, Wright K.W.J, Pickard H.M. "Finite Element Stress Analysis of the crowns of normal and restored teeth" J Dent Res, 1976; 55(6):1004-1010
10. Rubin C, Krishanamurity N, Capilouto E "Stress Analysis of the Human Tooth Using Three- dimensional Finite Element Model" J Dent Res, 1983;62(2):82-86
11. Goel V.K, Satish C. Kehra, Ralston J.L, Chang K.H "Stresses at the dentinoenamel junction of human teeth - A finite element investigation." J Prosthet Dent, 1991; 66:451-459
12. Rees J.S, Jacobsen P.H "Modeling the effects enamel anisotropy with the finite element method" J Oral Rehabil, 1995; 22:451-454
13. Rees J.S "The role of cuspal flexure in the development of abfraction lesions: a finite element study" Eur J Oral Sci, 1998; 106:1028-1032
14. Toparli M, Gokay N, Aksoy T. "Analysis of a restored maxillary second premolar tooth by three dimensional finite element method" J Oral Rehabil, 1999;26:157-164
15. Rees J.S, Jacobsen P.H "The effect of interfacial failure around a class V composite restoration analyzed the finite by the finite element method" J Oral Rehabil, 2000; 27:111-116
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, 2000; 27:1077-1081
17. Ausiello P, Apicella A, Davidson C.L, Rengo S "3D-finite element analysis of cusp movements in a human upper premolar, restored with adhesive resin based composites". J Biomech, 2001; 34:1269-77
18. Sato Y, Shindoi N, Koretake K, Hosokawa R. "The effect of occlusal rest size and shape on yield strength". J Prosthet Dent. 2003 May; 89(5):503-7
19. Han P, Dia K, Jia H "3D finite element analysis of stress distributions in supporting tissues of clasp-type partial dentures of transferring occlusion force" Hua Xi Kou Qiang Yi Xue Za Zhi. 2000 Aug;18(4):262-5
20. 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. 2000 Aug; 18(4): 229-31
21. Pierrisnard L, Bohin F, Renault P, Barquins M "Corono-radicular reconstruction of pulpless teeth: a mechanical study using finite element analysis" J Prosthet Dent. 2002 Oct; 88(4): 442-8
22. 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. 2000 Mar ;19 (1):50-64
23. Cheng R, Zhou XD, Liu Z, Hu T. Development of a finite element analysis model with curved canal and stress analysis. J Endod. 2007 Jun; 33(6): 727-31

Author Information

Gaurav Vasudeva, MDS
Reader, Department of Conservative Dentistry and Endodontics, Gian Sagar Dental College and Hospital