ECRONICON OPEN ACCESS

EC DENTAL SCIENCE

Review Article

Finite Element Analysis (FEA) With ANSYS - Tool of Computer Aided Engineering (CAE): Basics and its Applications in Dentistry

Jasjit kaur[1]*, Munish Goel[2], Navneet Sharma[3], Mamta Mahajan[4] and PR Verma[5]

[1]PhD Scholar in Dental Sciences, Department of Prosthodontics, Pacific University, Udaipur, Rajasthan, India

[2]Professor and HOD, Department of Conservative Dentistry and Endodontics, Himachal Dental College, HP, India

[3]Professor, Department of Oral Medicine and Radiology, Himachal Dental College, HP, India

[4]Senior lecturer, Himachal Dental College, HP, India

[5]PhD Guide, Pacific University, Udaipur, Rajasthan, India

***Corresponding author :**Dr. Jasjit Kaur, PhD Scholar in Dental Sciences, Department of prosthodontics, Pacific University, Udaipur, Rajasthan,India

Received: March 13, 2016; **Published:** April 16, 2016.

Abstract

Finite element analysis is the backbone of computer aided engineering. Today the finite element method (FEM) is considered as a well established and convenient technique for the computerized solutions of complex problems in different fields of engineering and biomedical engineering. ANSYS workbench developed NY ANSYS Inc; USA, it is a computer aided Finite element analysis (FEA) tool. It has a graphical user interface that can be used to generate 3D and FEA models and analyze the stresses and strains in all materials including living tissues. This article comprehensively reviews the finite element analysis and its software, its methodology and its applications in various specialties in dentistry.

Keywords: *Computer Aided Engineering; FEA; FEM; ANSYS; Dentistry*

Abbreviations: FEA: Finite Element Analysis; FEM: Finite Element Method; CAE: Computer Aided Engineering

Introduction

CAE (computer aided engineering) is the use of computer software to improve the product's quality and durability.It is the study of behavior of components in real time conditions. Here, designs are evaluated and refined using computer simulations rather than physical prototype testing thus saving time, efforts and money. CAE can also evaluate the performance in order to improve the product design. FEA is the backbone of CAE (computer aided engineering). FEA is a computing technique that is used to construct computer model of design that is loaded and analyzed for specific results. ANSYS is that computer software used in FEA to design model in computer so that the model can be simulated and tested in real time conditions. This article addresses the basics of FEA, its software, its methodology and its applications in various specialties in dentistry.

Material and Methods

An exhaustive search was undertaken to identify published literature related to computer aided engineering. A systematic review of PubMed/MEDLINE, databases was undertaken from publications between the years 2000 to 2014. Material related to generation of 3D and FEA models in ANSYS workbench programme in the graphical user interface and for analysis of stresses and strains in all materials including living tissues was collected. Literature related to the various applications of computer aided engineering which are helpful in dentistry was gathered. Single or combined key words were used to obtain the most possible comprehensive list of articles. Studies and investigations related to the topic were included. Selected articles were then obtained, studied and reviewed.

Basics of ANSYS work bench

ANSYS workbench developed NY ANSYS Inc; USA is a computer aided finite element modeling (FEM) and Finite element analysis (FEA) tool. In the graphical user interface of ANSYS workbench the user can generate the 3D and FEA models as shown in figure 1. It performs analysis and generates results. It is the framework upon which advanced simulation technology is built.

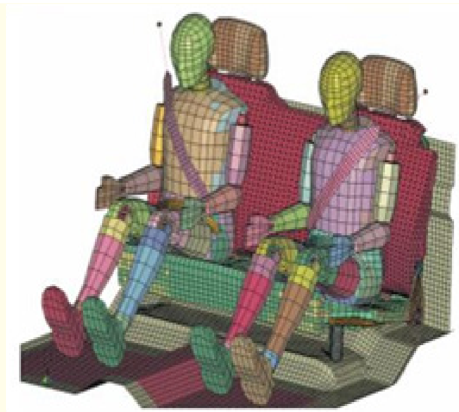


***Figure 1**: 3D Images generated with FEA.*

General process of ANSYS is divided into three main phases as shown in flow chart 1

a. Preprocessing

b. Processing/ Solution

c. Post processing

Physical problem

FEA through ANSYS software

Preprocessor — **FEM -** (Generates nodes, elements, boundary conditions, material properties, load and data file)

Solution — **FEA -** (Generates element, matrices, compute nodal value, derivatives and store results)

Post processor — **FEA -** Analyze results **-** (Display curves, counters, deformed shapes)

Preprocessing -Sketching and part modeling

The part model is created in ANSYS workbench are parametric and feature based. "Parametric" is defined as the ability to use the standard parameters in determining the size and shape of the geometry. "Feature" is defined as the smallest building block of the model that can be modified individually. 3D model is created using ANSYS workbench are a combination of sketched and placed features. ANSYS designer technology also provides powerful tool for construction of geometry from ground up. Two dimensional sketches can be extruded into 3-D solids and then modified with Boolean operations. A construction history is recorded during the creation of geometry allowing the user to make changes and update the design. "Design modeler" is a full featured system that provides importation of models from other CAD systems and allows user to export models to other CAD system.

Defining material properties: Material selection is very important and it should be guided by the way the material behaves in the real time condition according to use. Material response to the environmental stimulus is called "property". Based on its application and usage of material its properties become significant.

Meshing: A mesh is discretisation of components into a number of small elements of defined size. "Element" is an entity into which the system under study is divided. Type of elements in various dimensions is shown in Figure 2. These elements are connected to each other at points called nodes. Each node has two or more than two elements connected to it. The shape of an element depends upon the nodes with which it is made is shown in Figure 3. There are many types of element shapes that further divided into various classes depending on their uses e.g. Point element - A point element is in the form of a point and therefore it has only one node. Line element - It has shape of line or curve. Therefore minimum of two nodes are required to define the material. The element does not have a node in between the edges is called a linear element. The element that has nodes in between the edges is called quadratic or second order elements. Area element - An area element has the shape of a triangle or a quadrilateral so it requires a minimum of three or four nodes to define it.

Volume element: A volume element has the shape of a hexadron (8 nodes), wedges (6 nodes), tetradron (4 nodes) or a pyramid (5 nodes) (Figure 4). A collection of these elements is called a mesh. (Figure 5) Meshing is an extremely important part of preprocessing. ANSYS workbench creates mesh which requires less computational time and gives maximum accuracy. Meshing is also most important part of simulation.

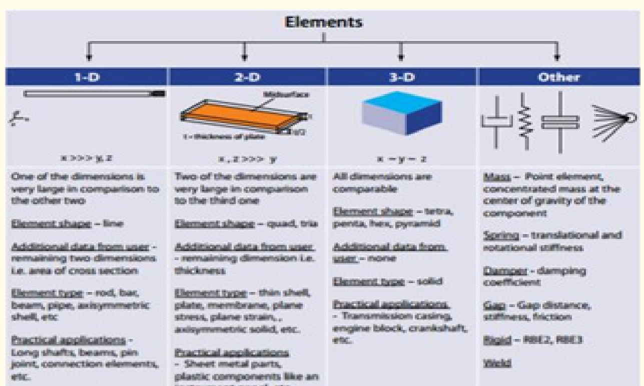


Figure 2: *Different Types of elements in various dimensions*

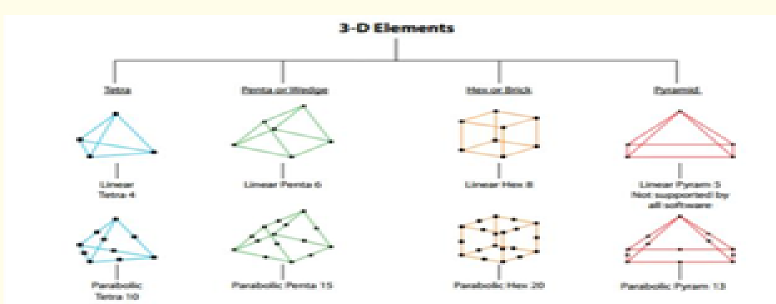


Figure 3*: Shape of 3D elements with different nodes.*

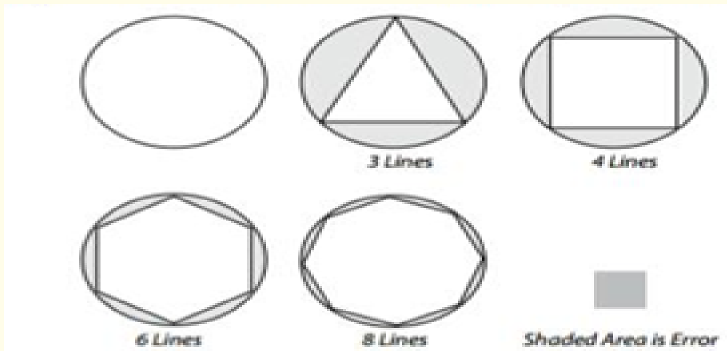


Figure 4*: Line, Point, Area And Volume Elements.*

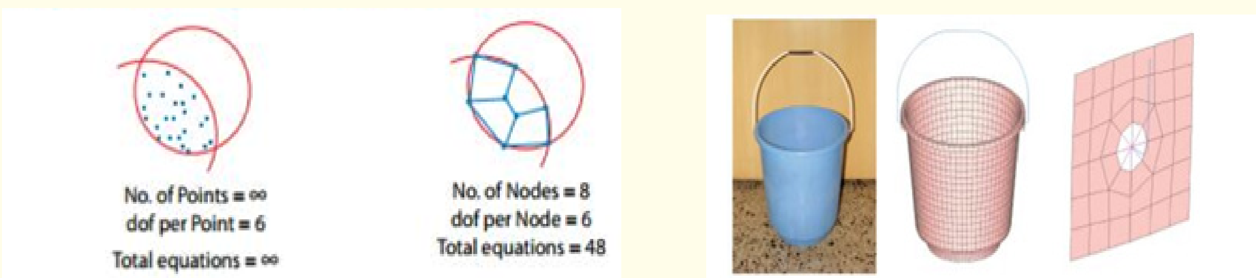


Figure 5*: Meshing with nodes.*

Loading and boundary conditions: Application of force at various points of geometry can also be evaluated.

Processing

It is completely automatic. Solution software generates element matrices, computer nodal value and stores result data in files. There are two approaches to solve any problem.

- Continuous approach
- Discrete approach

Discretization of problem

All real life objects are continuous. This means there is no gap between two consecutive particles. As per material science, any object is made up of small particles, particles of molecules, molecules of atoms and so on and they are bonded together by the force of attraction. Solving a real life problem with continuous approach is difficult. The basis of numerical method is to simplify a problem by discretizing it. In other words nodes work like atoms and results are interpolated for the elements. As shown in figure 6.

These files are further used in subsequent phase to review and analyze the results through the graphic display and tabular listings.

Post processing effects

It processes the data. This phase is also automatic and generates graphical output in the specified form to check or analyze the results.

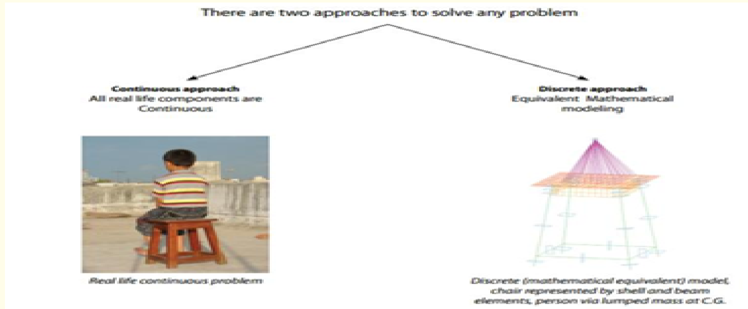
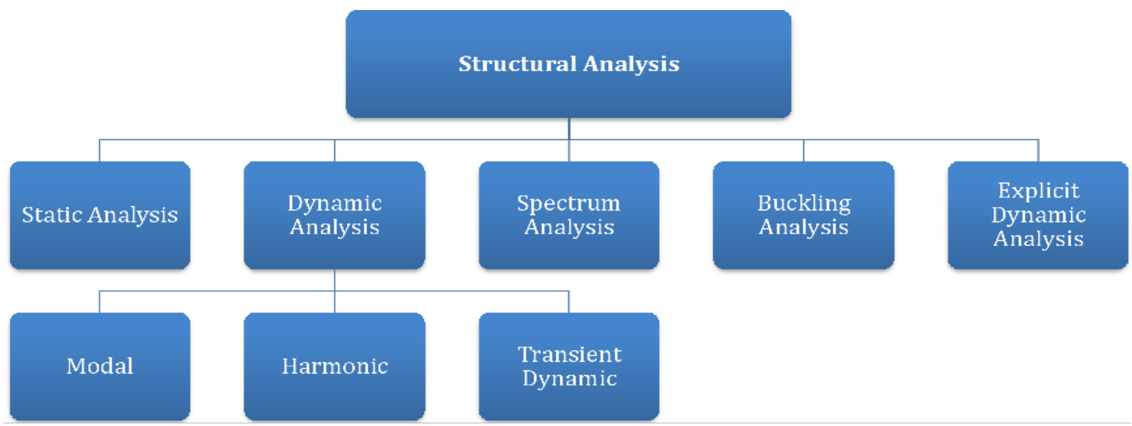


Figure 6: *Discretization approach in processing.*

Following types of analysis performed through ANSYS



Features of ANSYS

- Drag drop simplicity with bidirectional connectivity,
- Powerful and automated meshing can simulate modal with more accuracy.
- Project level updates mechanism- capable of solving models with several million degrees of freedom and above.
- Solver improvements and increased computer resources have enabled modal solution of the full model improving accuracy and robustness.
- Solver improvements in ANSYS extend the capabilities for larger simulations.
- Refinement of the solver interfaces reduced the minimum memory requirements.

Advantages of ANSYS in Dentistry

FEA is a non invasive technique for obtaining a solution to complex mechanical problems by dividing the problem's domain into collection of much smaller domains in which field variables can be interpolated with the use of shape function. In FEA the actual amount of stress experienced at given point can be theoretically measured by simulating the oral conditions. In this displacement of tooth can be visualized graphically, the point of application, magnitude, direction of force may easily varied to simulate the clinical situations, reproducibility does not affect the physical properties and the study can be repeated as many times as the operator wishes [1].

Clinical Applications of FEA

1. FEM is used as tool to study orthodontic tooth movement [2].
2. It is a useful tool in morphometric analysis in craniofacial biology [3].
3. The programme uses the color graphics display of common personal computers to show the size change, shape change, and angle of maximum change [4].
4. CAD CAM template gives orthodontics software to place mini screws [5].
5. It helps in studying the effect of altering the geometry of the bracket base mesh on the quality of orthodontic attachment by employing a 3D model [6].
6. It is useful in studying the biomechanical interactions between dental implants and jaw bones surrounding the implants. FEA predicts stress distribution in the contact area of the implants with cortical bone and around apex of the implants in trabecular bone [7,8].
7. In Prosthodontics, it is extremely useful in dental prosthetic designing. It helps in studying the stress distribution in supporting structures in relation to different designs of removable and fixed prosthesis. Designs for fiber framework for FPD are extensively investigated with FEA.
8. It is extremely useful in find out the stress distribution in adhesively cemented ceramics and resin composite class II inlay restorations [9].
9. FEM can also be applied for prediction of face soft tissue deformations resulting from bone repositioning in maxillofacial surgery [10].
10. In Periodontology, it is used to evaluate the stress distribution in peridontium under different loading conditions.
11. FEA is used to study the pattern of stress distribution in various post and core restorations [11-15].

Role of FEA in Previous Researches

1. Weinstein *et al*. were the first to use FEA in dentistry [16].
2. Atmaram and Mohamed [17-19] analyzed the stress distribution of a single tooth implant to understand the effect of elastic parameters and geometry of the implants, implant length variations and pseudo periodontal ligament incorporation.
3. Borchers and Reichart [20] performed 3-dimensional FEA of an implant at different stages of bone interface development.
4. Cook et al. [21] applied FEA to porous dental implants.
5. Williams *et al*. [22] carried out FEA on cantilever prosthesis in implants.
6. Akpiner [23] used FEA to simulate combinations of a natural tooth to implants.
7. Davy *et al*. [12] analyzed two dimensional plane strain finite element models to determine stresses in post restored central incisor models.
8. Eskitascioglu G., *et al*. [13] studied stress distribution in post and core through FEA.
9. Barjau-Escribano A., *et al*. [14] studied stress distribution with FEA in prefabricated glass fibre and stainless steel post and core.
10. Ausiello P., *et al*. [15] studied the mechanical behavior in post restored teeth with FEA.

11. Chen D., *et al*. [16] studied the behavior of maxiallry canine restored with zirconia post .

12. Chuang SF., *et al*. [17] studied the effect of post length on stress distribution in various post and core system.

13. Dejak B and Mlotkowski A [18] compared the strength of cast post and fiber post with FEA.

Key assumptions in FEA

There are following types of key assumptions that must be considered while performing the FEA.

Assumptions related to geometry

- Displacement value will be small so that linear solution is valid.
- Stress behavior outside the area of interest is not important. Local behavior at the corners, joints and intersection of geometry is important and of primary interest.
- Decorative external features should be omitted

Assumptions related to material properties

- Material properties should remain in linear region. It should not be affected by load rate.
- The components should be free from surface imperfections. All simulation will assume room temperature. Water sorption and humidity considered to be negligible.

Assumptions related to boundary conditions

- Displacement value will be small so that linear solution is valid.
- Stress behavior outside the area of interest is not important. Local behavior at the corners, joints and intersection of geometry is important and of primary interest.
- Decorative external features should be omitted

Limitations of FEA

- The roots, PDL, and teeth are represented in idealized geometric forms and physical properties seem to be isotropic, homogenous and linear.
- The most significant limitation is the accuracy of results which depends upon the mesh resolution. Areas of concentrated stress should be carefully evaluated with help of refined mesh.

Future applications

In dentistry FEA was mainly used to analyze the structural components for displacements, stresses, strain and force under loading conditions. It has provided information on the structural behavior of components under a specific boundary and loading conditions. The use of Buckling analysis and thermal analysis in dentistry are the recent topic of interest for researchers. The researchers should also take benefit from other types of structural analysis like Dynamic analysis, Spectrum analysis, and Explicit Dynamic analysis.

Conclusions

CEA was limited to different branches of engineering but now can be used in biomedical engineering especially in various fields of dentistry to study the stress patterns related to structural behavior of oral tissues. FEA is recent discipline that has quickly become a method of choice in structural analysis. It is a numerical method which reduces the infinite degree of freedom to finite with help of discretization or meshing. ANSYS is used to model, analyze and simulate systems in structural, thermal, mechanical, computational fluid dynamics, electromagnetics, and other applications. Here, solution of complex mechanical problems can be obtained by dividing the problem domain into a collection of much smaller and simpler domain and in which the field variables can be interpolated with the use of shape function. It is a method whereby, instead of seeking a solution function for entire domain, one formulates the solution functions for each finite element and combines them properly to obtain the solution for whole body.

The FEA was first developed to be used in the aerospace and nuclear industries where the safety of structure is critical. Today, even the simplest of products rely on FEA for design evaluation. As ANSYS is fully capable of delivering higher quality products in shorter design cycle with fewer chances of field failures.

Conflict of Interest

The authors declared that this case has no personal and financial relationship with any corporate affiliations or any other organizations.

Disclosures

The grants, patent licensing arrangements, consultancies, stock or other equity ownership, advisory board memberships or payments were not related for conducting and publicizing this case.

Bibliography

1. Alirezajafari., *et al.* "Study of stress distribution and displacement of various craniofacial structures following applications of transverse orthopaedic forces-A three dimensional FEM study". *The angle orthodontist* 73.1 (2003): 12-20.

2. PM Cattaneo., *et al.* "The finite element method: A tool to study orthodontic tooth movement". *JDR* 84.5 (2005): 428-433.

3. K Prasad and SA Tarannum. "Basic principles of FEM and its applications in orthodontics". *Journal of Pharmaceuticals and Biomedical Sciences* 16.16 (2012): 1-4.

4. Glenn T Sameshima and Michael Melnick. "FEA based cephalometric analysis". *The Angle Orthodontist* 64.5 (1994): 343-350.

5. Hong Liu., *et al.* "Accuracy of surgical positioning of orthodontic mini screw with a computer aided design and manufacturing template". *American Journal of Orthodontics and Dentofacial Orthopedics* 137.6 (2010): 728e1-728 e10.

6. Jeremy Knox., *et al.* "An evaluation of the quality of orthodontic attachment offered by single and double mesh bracket bases using the fem of stress analysis". *Angle Orthodontist* 71.2 (2001): 149-155.

7. RC Vanstaden., *et al.* "Application of FEM in dental implant research". *Computer Methods in Biomechanics and Biomedical Engineering* 9.4 (2006): 257-270.

8. Ruppin J., *et al.* "Evaluation of accuracy of 3 different computer aided surgery system in dental implantology". *Clinic Oral Implants Research* 19.7 (2008): 709-716.

9. Stress distribution in adhesively cemented ceramic and resin composite class ii inlay restorations: A 3d FEA study. *Dental Materials* 20 (2004): 862-872.

10. Anitua E., *et al.* "Influence of implant length, diameter and geometry on stress distribution: A FEA". *The International Journal of Periodontics and Restorative Dentistry* 30 (2010): 89-95.

11. Davy DT., *et al.* "Determination of stress patterns in root filled teeth incorporating various dowel designs". *Journal of Dental Research* 60 (1981):1301-1310.

12. Eskitascioglu G., *et al.* "Evaluation of two post core systems using two different methods (fracture strength test and a finite elemental stress analysis)". *Journal of Endodontics* 28.9 (2002): 629-633.

13. Ausiello P., *et al.* "Mechanical behaviour of post-restored upper canine teeth: a 3D FE analysis". *Dental Material* 27.12 (2011): 1285-1294.

14. Chuang SF., *et al.* "Influence of post material and length on endodontically treated incisors: an *in vitro* and finite element study". *Journal of Prosthetic Dentistry* 104.6 (2010): 379-388.

15. Dejak B and Mlotkowski A. "Finite element analysis of strength and adhesion of cast posts compared to glass fiber-reinforced composite resin posts in anterior teeth". *Journal of Prosthetic Dentistry* 105.2 (2011): 115-126.

16. Weinstein AM., *et al.* "Stress analysis of porous rooted dental implants". Journal of Dental Research 55 (1976): 772-777.

17. Atmaram GH and Mohammed H. "Stress analysis of single-tooth implants. "Effect of elastic parameters and geometry of implant". *Implantologist* 3 (1983-84): 24-29.

18. Atmaram GH and Mohammed H. "Stress analysis of single-tooth implants. II. Effect of implant root-length variation and pseudo periodontal ligament incorporation". *Implantologist* 3 (1983-84): 58-62.

19. Mohammed H., *et al.* "Dental implant design: a critical review". *Journal of Oral Implantology* 8 (1979): 393-410.

20. Borchers L and Reichart P. "Three-dimensional stress distribution around a dental implant at different stages of interface development". *Journal of Dental Research* 62 (1983): 155-159.

21. Cook SD., *et al.* "A three-dimensional finite element analysis of a porous rooted Co-Cr-Mo alloy dental implant". *Journal of Dental Research* 61 (1982): 25-29.

22. Williams KR., *et al.* "Finite element analysis of fixed prostheses attached to osseointegrated implants. *Quintessence International* 21 (1990): 563-570.

23. Akpinar I., *et al.* "A comparison of stress and strain distribution characteristics of two different rigid implant designs for distal- extension fixed prostheses". *Quintessence International* 27 (1996): 11-17.

24. Practical finite element analysis. Nitin S Gokhale., *et al.* 1st edn (2008).

25. Figure 1 courtesy: Tata Johnson Controls Automotive limited, India.

Volume 4 Issue 2 April 2016