Impact Factor (SJIF): 3.632

International Journal of Advance Research in Engineering, Science & Technology

e-ISSN: 2393-9877, p-ISSN: 2394-2444

Volume 3, Issue 7, July-2016

CAD MODELING AND STRESS ANALYSIS OF HUMAN TOOTH USING FINITE ELEMENT METHOD

Vinayak R.Tayade

P.G.Student, Department of Mechanical Engineering, S S G B COE&T, Bhusawal, Maharashtra, (India)

Abstract

In the present modern era, every human is concerned about dental health and look. It represents ones personality in the social field. Various functional loads create stresses inside the tooth which governs failure of tooth. To study the stress of critical part of human body, the field of dentistry popularly applies important branch of engineering "Biomechanics". Traditional dentistry have many restrictions for image analysis these methods are confined to only 2D , on the other front traditional stress analysis confined to only methods of strain gauge, Photo elasticity and 2D Finite Element analysis. These traditional methods resulted in inability of finding solution for complex behavior of human tooth. These methods are unable to carry stress analysis of human's real tooth in 3D as well these methods are time consuming and not accurate. Present work finds scope to reproduce CAD model and carry stress analysis of human's real tooth. The scope covers critical factors of tooth like irregular geometry, complex material properties and complicated loading conditions in 3D to bring accurate results in short time. There is a scope to perform analytical stress calculation of complex geometry of tooth. Present work overcomes past restrictions by image processing and reproducing actual 3D tooth model of human's real Molar tooth by the use of CT scan and suitable biomedical imaging software 3D Doctor. The reproduced 3D model is used for FE analysis with the use of FEA software ANSYS. As the Analytical approaches are not available for complex dental structures the stress analysis is done by analytical method too. Present study revealed that, the use of CT scan and 3D doctor software enabled to generate accurate 3D CAD model of actual humans tooth. This obtained exact solution, better visualization for actual tooth subjected to loads of mastication. Stress analysis results quantified the stress value and confirmed that stress due to bending load is higher compared to axial load. Use of FEA tool for such complex geometry and complex loading conditions overcomes the limitations of experimental and analytical approaches used for stress analysis. The methodology adopted in this study adds value to investigate optimal designs for dental implant in shortest time. This also helpful for treatments & complicated surgeries for Bio medical, Dentistry field & Skeletal Anchorage System (SAS) of human body in shortest time.

Keyword- *Normal Human tooth, cad model, finite element analysis, stress analysis.*

I. INTRODUCTION

Dental science, like much of the evolution of human civilization, progresses in the steps that are often the result of the complex relationship between science, empirical knowledge, and advances in technology. Computing has modified our perception, sense, use and interpretation of time and enabled scientists to perform existing procedures far faster and more accurately than ever. It has allowed them to make a reality of things they had only dreamed of before and perhaps of greater consequence and more excitingly. It has often stimulated them to perceive and focus on their subject with new eyes to see it on a different scale from a completely different perspective. Human tooth is a critical and most important part of human to survive. It is a part of body which develops in early trimester during pregnancy of mother. During its life cycle, human tooth has to fight with different kinds of chemical and mechanical effect by the action of chewing of different kind of food, accidents, progress in age. In the fashionable era every human is concerned about dental health and look .To maintain its health, preventive or predictive clinical operations are required. Clinical operations on tooth are always painful and fearful. For gaining accurate solution, traditional practices often required multiple visits and multiple repetitive operations , however it doesn't guarantee for the optimum solution. Every human wish to have technology which should be painless, optimum in cost and in shortest time. One would have never thought of going to dentist where doctors can make use of latest technology like CT scan, 3D doctor and FEA which gives the best clinical solution within shortest time. Present work is golden mean of science and latest software tool FEA for finding cost effective and reliable clinical solutions for betterment of most important part for facial expression and digestive media i.e. human molar tooth.

II. FE ANALYSIS OF MOLAR TOOTH

2.1 Introduction

Many problems in engineering are governed by differential or integral equations. The solution to these equations would provide an exact, closed-form solution to these equations for the particular problem being studied. However, complexities in geometry & in boundary conditions that are seen in most real world problems usually means that an exact solution cannot be obtained or obtained in real time. But current product design cycle times imply that engineers must obtain design solutions in a short amount of time. They are content to obtain approximate solutions that can be readily obtained in a reasonable time frame with reasonable effort. The FEM is one such approximate solution technique.

The FEM is a numerical procedure for obtaining approximate solutions to many problems encountered in engineering analysis. Mathematically, the structure to be analyzed is subdivided into a mesh of finite sized elements of simple shape. Within each element, the variation of displacement is assumed to be determined by simple linear or quadratic shape function & nodal displacements. For this the equations of equilibrium are assembled in a matrix form which can be easily be programmed & solved on a computer. After applying approximate boundary conditions, nodal displacements are found by solving the matrix stiffness equation. Once the nodal displacements are known, element stresses & strains can be calculated.

Fig. 2.1 below shows typical FE procedure using commercial tools.

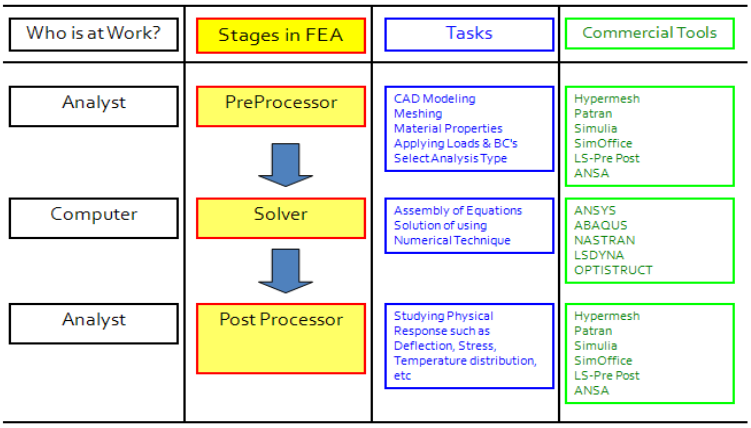

Fig. 2.1. Typical FE Procedure using Commercial Tools

The FEA is mainly divided into three following stages:

A. Preprocessing

A1. Creating the model – It is the basic step to prepare model for analysis.

A2. Defining the element type - For an accurate analysis in FEA, selection of the proper elements is very important. The selected elements must represent the engineering structure as close to the original structure as possible. In FEA, an engineering structure is divided into small elements. These elements coincide with the geometry of the structure and represent the geometry and the mechanical properties in the regions. FEA need elements that have geometric shape similar to the real structure or region of the structure that is being modeled. The elements further divided into line element, 2-D element & 3-D element.

A3. Defining material properties – Material properties are needed as a input for analyzing the behavior of model for specified applicable material properties. The only material properties that are generally required by an isotropic, linear static FEA are Young's modulus (E), Poisson's ratio (v), shear modulus (G), and yield strength (or ultimate strength).

A4. Meshing - The process of dividing the model into small pieces is called meshing. The structure represented by nodes and elements is called "mesh". The behavior of each element is well-known under all possible support and load scenarios. Accuracy of FEA analysis is depend on quality optimization of mesh. This is done by process called convergence [Von-Mises Stress (VMS) convergence, Strain Energy Convergence, Deflection Convergence].

A5. Applying loads - Loads are used to represent inputs to the system. They can be in the forms of forces, moments (torque), pressures, temperature, or accelerations.

A6. Applying boundary conditions - In FEA, the terminology "Boundary condition" is used for calculating the load and figuring out constraints that each component experiences in its working environment. The choice of boundary conditions has a direct impact on the overall accuracy of the model. The boundary conditions relates to the degree of freedom.

B. Solution: Assembly of equations and obtaining solution.

C. Post processing: Review of results such as deformation plot, stress plot, etc.

ANSYS is widely used FE analysis software both in academics as well as in industry [56]. Using ANSYS one can perform various tasks of FE analysis to meet the user requirements such as composite material definition, optimization and probabilistic design. There are two different user interfaces available i.e. ANSYS classic and ANSYS workbench. Traditionally, ANSYS classic is choice of analysts. Hence, present work uses ANSYS classic version 13. Figure 5.2 shows user interface of ANSYS classic version 13. Using this user interface various inputs are provided to the ANSYS required for [51]. There are two distinct modes of ANSYS usage, i.e. the Graphical User Interface (GUT) and Batch Mode.

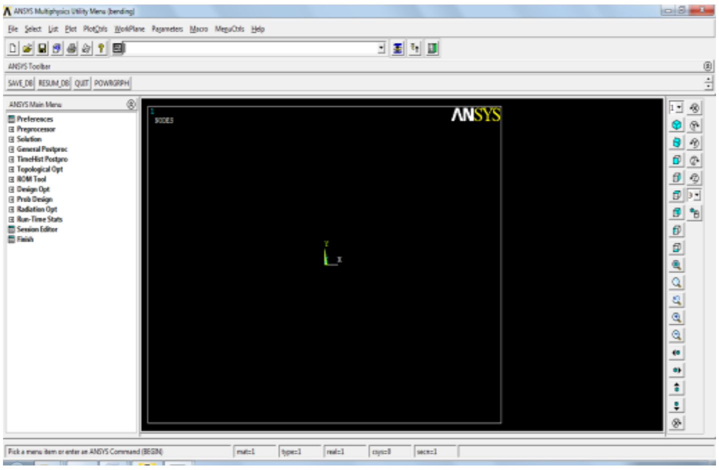

Fig. 2.2. ANSYS Classic User Interface

During the last two decades considerable advances have been made in the applications of FEM techniques to the dentistry. FE analysis of molar tooth can be carried out using 3D elements.

2.2 FE Analysis of Molar Tooth

Finite element method is an excellent tool to analyze complex structures like molar tooth by using computer which can help to reduce time and cost required for prototyping during design.

2.2.1 Importing of CAD Model

CAD model of molar tooth is shown in Figure 4.9 and is created using 3D Doctor as discussed in chapter 4. 3D model is exported in STereoLithography (STL) format. STL is a file format native to the stereolithography CAD software created by 3D Systems. STL is also known as Standard Tessellation Language . This file format is supported by many other software packages. It is widely used for rapid prototyping and computer-aided manufacturing. STL files describe only the surface geometry of a three dimensional object without any representation of color, texture or other common CAD model attributes. The STL format specifies both ASCII and binary representations [58]. An STL file describes a raw unstructured triangulated surface by the unit normal and vertices (ordered by the right-hand rule) of the triangles using a three-dimensional Cartesian coordinate system. STL coordinates must be positive numbers, there is no scale information, and the units are arbitrary. Figure 2.3 shows imported CAD model in ANSYS with 3D view. Model in STL format is imported in ANSYS-ICEM.

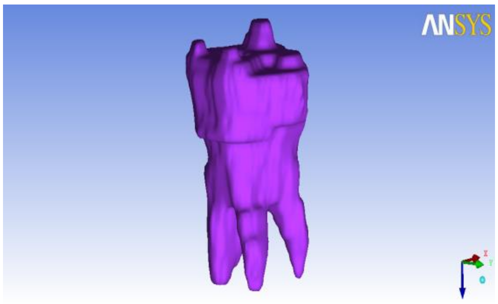

Fig.2.3. 3D CAD Model of Molar Tooth in ANSYS-ICEM

2.2.2 Mesh Generation

Present work uses ANSYS-ICEM for meshing of molar tooth. ANSYS-ICEM provides advanced geometry acquisition, mesh generation and mesh optimization tools for today's sophisticated FE analysis. Maintaining a close relationship with the geometry during mesh generation ANSYS-ICEM is used especially in engineering applications such as computational fluid dynamics and structural analysis. ANSYS ICEM provides a direct link between geometry and analysis. In ANSYS ICEM, geometry can be input from just about any format, whether from a commercial CAD design package, 3rd party universal database, scan data or point data. Present work uses STL geometry as discussed in 2.2.1. The resulting structured or unstructured meshes, topology, inter-domain connectivity and boundary conditions are then stored in a database where they can easily be translated to input files formatted for a particular solver.

Present work uses Tetra mesher of ICEM for meshing of molar tooth. Tetra mesher is automated to the point that user have only to select the geometry to be meshed, the Tetra mesher generates tetrahedral meshes directly from the CAD geometry or STL data, without requiring an initial triangular surface mesh. Figure 2.4 shows meshed model ANSYS-ICEM. Meshing is nothing but converting a whole geometry into number of elements and these elements are connected by nodes.

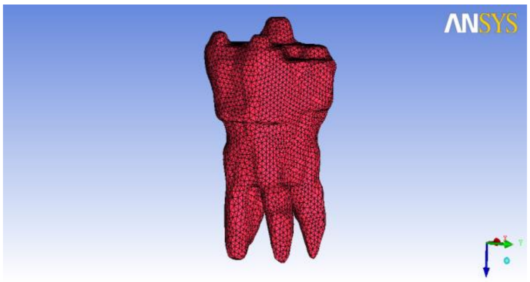

Fig. 2.4. Meshed model of molar tooth in ANSYS-ICEM

Molar tooth is meshed with tetrahedral solid elements of ANSYS. Figure 2.4, 2.5, 2.6 and 2.7 shows meshed models of molar tooth in ANSYS MAPDL.

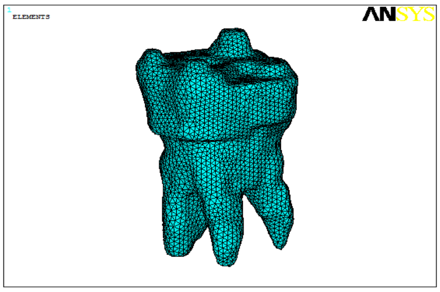

Fig. 2.5. Meshed model of molar tooth in ANSYS MAPDL

Figure 5.6 below shows meshed models bottom view of molar tooth in ANSYS MAPDL.

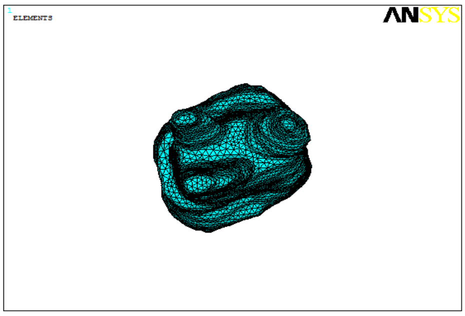

Fig.2.6. Molar tooth - meshed model (bottom view)

Figure 2.7 below shows meshed models front view of molar tooth in ANSYS MAPDL.

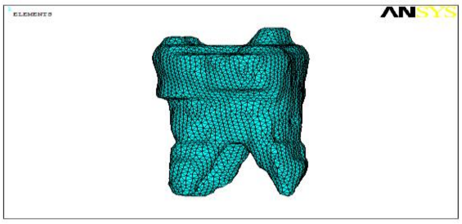

Fig. 2.7. Molar tooth - Meshed model (Front view)

Figure 2.8 shows a SOLID element which is tetrahedral element used in meshing of molar tooth. It is well suited to modeling irregular geometries such as biomedical organs. The element is having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

Figure 2.8 shows a typical element used in meshed models.

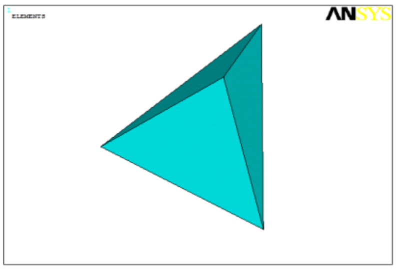

Fig. 2.8. Structure of solid tetrahedron element

Table 2.1 shows mesh statistics for the complete model.

Element Type	Solid Tetrahedron
Number of Elements	91504
Number of Nodes	17286

Table 2.1. Mesh Statistics

2.2.3 Material properties

Molar tooth material is assumed to be homogeneous, isotropic and have linear elastic behavior. Table 2.2 shows the mechanical properties of tooth that are used for FE analysis. The mechanical properties of tooth materials are used from past literature. As discussed in Chapter 1, tooth has different regions and each region is made of different materials as shown in Table 2.2. Major strength bearing region is made of Enamel and hence present work uses properties of enamel during FE analysis.

Material Property	Young's Modulus (GPa)	Poisson's Ratio
Enamel	84.1	0.3
Dentine	13.7	0.3
Cementum	18.6	0.31
Pulp [24]	0.002	0.45

Table 2.2: Mechanical properties of premolar tooth [13]

2.2.4 Loads Boundary Conditions (LBC's) and Other Input Parameters

The values of forces acting on teeth have been extensively addressed in the literature [1-11]. These values are usually obtained when a tooth is under loading due to either mastication or biting. Biting is understood as the act of breaking food in large pieces that will be further chewed. In this process, a tooth is subjected to what is called occlusal loads. In mastication, the forces are briefer and less intense. In this work, mastication load equal to 800 N inclined at 25 degree is applied on tooth as shown in Figure 2.9. The mastication load value is obtained from literature Roots of tooth are fixed in all directions.

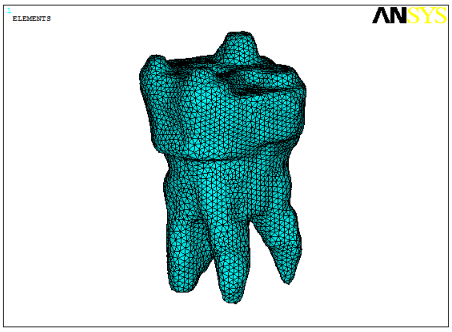

Fig. 2.9. Mastication load on molar tooth

5.2.5 Solution

The given configuration of molar tooth is solved using ANSYS with inputs discussed in above section. Linear static structural analysis is performed.

2.2.6 Post-processing

Post processing involves the review of various results such as stresses, strain and deformations. Due to loading conditions stress will be induced in the tooth. In result and discussion section the stress results on account of Axial load and Bending load are shown and discussed.

From results of post processing one can easily visualize the stress distribution in the object and use the results for further optimizations or improvements for finding solutions for respective design conditions.

III. ANALYTICAL APPROACH

In the present chapter the stress results are calculated by Analytical and FEA approach. The results are compared with each other. Analytical results are calculated based on methodology as explained in chapter 6. FEA results are calculated as per methodology briefed in chapter 2. 3D CAD model of humans real molar tooth is generated as per detailing .These results are presented and compared as below.

3.1 3D CAD model of human's real molar tooth

As explained in, there were 34 slices for maxilla scan of female of age 56 years obtained from dentist. These CT scan images transferred to 3D doctor software. From maxilla scan molar tooth is selected and by use of 3D doctor software approximate 3D CAD model is generated for further use in FE analysis.

This work is different from other works because the analysis is done on real humans tooth without extraction of tooth or without making prototype .So in this way a solution is obtained on live human tooth by sitting chair side and further with the help of FE analysis exact solution is obtained in accurate way in shortest time.

Figure 7.1 shows the 3D CAD model generated for 3D doctor. Fortunately the lab of dentist made available the physical model of this tooth for experiment purpose .Major dimensions of jaw compared from this physical model versus CAD model. As tooth is extracted form Jaw so indirectly dimensions of molar tooth is compared and matches to each other.

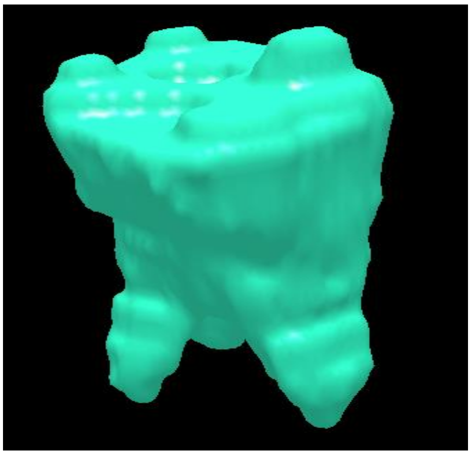

Fig.3.1. 3D CAD model of human's real molar tooth

3.2 Analytical Result for Axial stress

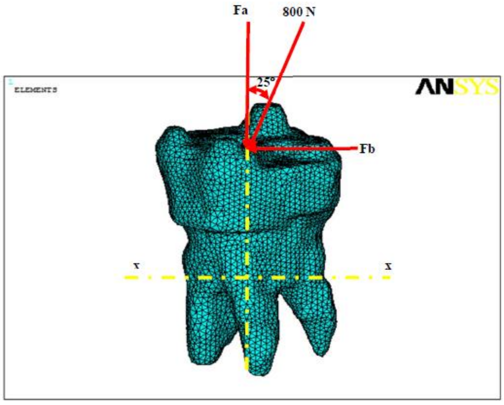

Fig.3.2. Axial stress in molar tooth by analytical approach

Figure 3.2 shows the force acting during mastication and resolution of vertical and horizontal component of forces for calculating Axial stress.

Resolving the mastication load in vertical direction, Force Fa is calculated as below,

F = 800 N

Fa=Axial Force = F x cos (25°) (7.1)

= 800 x cos (25°)

= 725 N

Using the value of the Fa in the basic formula of stress as below the Axial stress is calculated.

$$Stress = \frac{Force}{Area}$$

$$\sigma = \frac{Fa}{A}$$

Axial stress calculation for present case is as given below:

As stated in the chapter 6 and with reference to the figure 3.4 , the area for the plane X-X calculated form 3D doctor software considered as 41.57 mm^2.

Fa = 725 N, A=41.57 mm^2.

$$\sigma_{axial\,x-x} = \frac{725}{41.57} = 17.44MPa$$

3.3 Analytical Result for bending stress

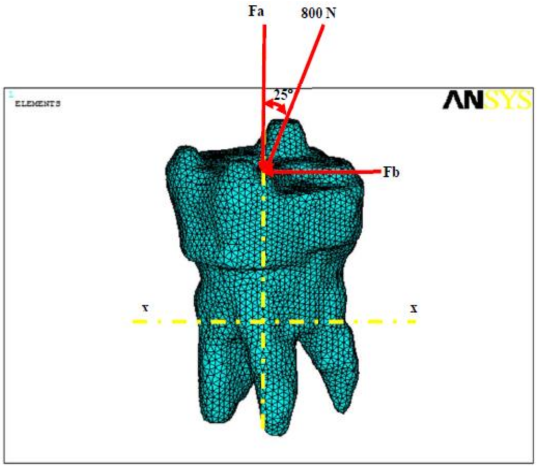

Fig.3.3. Bending stress in molar tooth analytical approach

Figure 3.3 shows the force acting during mastication and resolution of vertical and horizontal component of forces for calculating Bending stress.

Resolving the mastication load in horizontal direction, Force Fb is calculated as below,

F = 800 N

Fb = Bending Force = F x sin (25°)

= 800 x sin (25°)

= 338 N

Area at X-X calculated form 3D doctor software considered as 41.57 mm^2.following calculations are obtained:

Fb = 338 N, A = 41.57 mm^2.

$$R = \sqrt{\frac{A}{\pi}} = \sqrt{\frac{41.57}{3.14}} = 3.63mm$$

$$I = \frac{\pi}{4} R^4 = 137.51mm^4$$ Z=I/y

= 137.51/3.63

= 37.80 mm3

Finally, bending stress calculation for present case is as given below:

$$\sigma_x = \frac{M}{Z_{x-x}}$$

$$\sigma_{bending\,x-x} = \frac{338x13.5}{37.80} = 120.73MPa$$

3.4 FEA Result for Axial stress

FE analysis of molar tooth is performed by applying only axial component separately. Fig. 3.4 shows axial stress obtained using FE simulations.

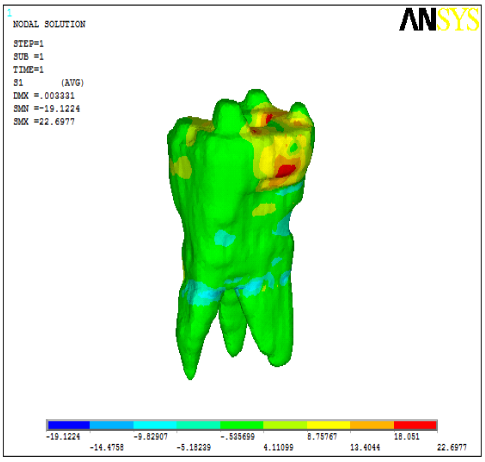

Fig. 3.4. Axial stress in molar tooth by FEA approach

3.5 FEA Result for bending stress

FE analysis of molar tooth is performed by applying only bending component separately. Fig. 3.5 shows bending stress obtained using FE simulations. The basics of FEA followed as explained.

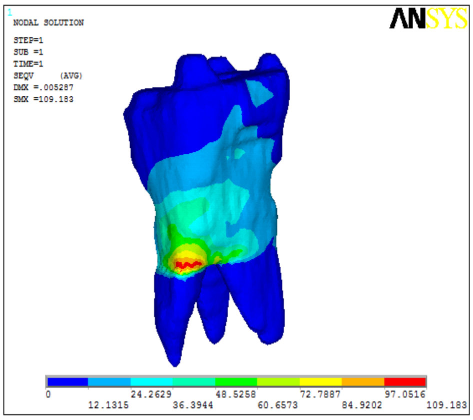

Fig.3.5. Bending stress in molar tooth by FEA approach

Table 3.1 summarizes the comparison of various results. There is a small difference in analytical and FE results. The difference is potentially related to assumptions in analytical formulations, load approximation etc. Since difference is small so results are considered as authentic.

Table 3.1. Comparison of Analytical and ANSYS Results

Type of Result	Analytical	ANSYS	% Error
Bending stress(MPa)	120.73	109.18	9.56
Axial Stress (MPa)	17.44	18.20	8.91

IV.CONCLUSION

The main conclusions of the current study are:

- Accurate three-dimensional CAD model is reproduced from DICOM images and converted to STL format to use it further in FE analysis.
- 3D stress analysis of molar tooth is successfully carried out on molar tooth modeled using FEA tool – ANSYS under mastication loading conditions. The FEA model neither simplifies geometry nor makes any assumptions in material. Hence, the developed FEA model is most accurate.
- Approximate analytical calculations have been performed for axial and bending stress comparison in molar tooth. Difference in analytical and FE results doesn't exceed 10% so results and method considered as authentic and valid.
- From stress distribution it appears that stress due to bending is higher as compared to loading due to axial component although bending component is lower than axial component.

Use of FEA tool for such complex geometry and complex loading conditions overcomes the limitations of experimental and analytical approaches used for stress analysis.

REFERENCES

1. Zienkiewicz OC, Kelly DW. Finite elements-A unified problem-solving and information transfer method. In: Finite elements in biomechanics. Gallagher RH, Simon. 1982.
2. Hughes TJR. The finite element method: linear static and dynamic finite element analysis. Prentice-Hall, Inc. 1987.
3. Segerlind LJ. Applied finite element analysis. 2nd ed. John Wiley & Sons, Inc. 1984.
4. Brekelmans WAM, Poort HW, Slooff TJ,"A New Method to Analyze the Mechanical Behaviour of Skeletal Parts",ActaOrthopScand, 1972, 43: 301-317.
5. Meijer HJ, Starmans FJ, Bosman F, Steen WH,"A Comparison of Three Finite Element Models of an Edentulous Mandible Provided with Implants", J Oral Rehabil, 1993, 20: 147-157.
6. Kamposiora P, Papavasilious G, Bayne SC, Felton DA, "Finite Element Analysis Estimates of Cement Microfracture Under Complete Veneer Crowns", J Prosthet Dent, 1994, 71: 435-441.
7. Rees JS, Jacobsen PH, "Modellingthe Effects of Enamel Anisotropy with the Finite Element Method", J Oral Rehabil, 1995, 22: 451-454.
8. Holmes DC, Diaz-Arrold AM, Leary JM, "Influence of Post Dimension on Stress Distribution in Dentin", J Prosthet Dent, 1996, 75: 140-147.
9. Kheraet al., "Parameters of MOD cavity preparations: A 3-D FEM Study",OperDent, 1991(16): 42-54.
10. Yamna SD, Alacam T, Yaman Y,"Analysis of Stress Distribution in a Maxillary Central Incisor Subjected to Various Post andCore Application", J Endod, 1998, 24: 107-111.