3D Modeling and Stress Analysis of Premolar Tooth Using FEA

Dattatraya Parle  
Lead Consultant  
Engineering Services  
Infosys Limited  
Pune, India

Anirudha Ambulgekar  
Senior Engineer  
Engineering Services  
Infosys Limited  
Pune, India

Dr. Ketan Gaikwad  
Research Fellow  
Tata Memorial Hospital,  
mumbai, India

Abstract

Finite element method (FEM) is in use for analysis of dental structures. Many FE studies carried out using approximate 2D models, assumptions in material properties, and with approximate Loads and Boundary Conditions (LBCs). However, these assumptions are valid only if the tooth structures are planar/symmetrical and regular in shape which does not represent real life problem. Recently, with advances in modeling and meshing techniques, researchers started using 3D modeling approach. There are continuous efforts to reproduce actual tooth model for various purposes such as visualization, stress analysis, manufacturing, etc. Earlier efforts of 3D modeling for FE analysis are again based on many assumptions for simplification of pre-processing activities and reducing computational efforts. Today’s commercial FEA softwares like HyperMesh and Radioss are capable of overcoming these shortcomings. Hence, the objective of this study is to carry out stress analysis using HyperMesh and Radioss on exact 3D model of premolar tooth created using medical imaging tools with the help of Computerized Tomography (CT) scanned images. Such FEA based studies help in improved understanding of biomechanics for dental structures and to produce artificial tooth with improved mechanical strength. Further, these studies help in orthodontic corrections of molar tooth using appliances like pendulum. In this paper premolar tooth is analyzed for few cases using the proposed approach.

Keywords: Premolar Tooth, FEA, Biomechanics, Stress Analysis

Abbreviations:

- CAD – Computer Aided Design
- CT – Computerized Tomography
- FEA – Finite Element Analysis
- ITK – NLM Insight Toolkit
- STL – Stereolitography
- VTK – Visualization Toolkit
- CAE – Computer Aided Engineering
- DICOM – Digital Imaging and Communications in Medicine
- FEM – Finite Element Method
- SAS – Skeletal Anchorage System
- VM Stress – Von Mises Stress
- VTK – Visualization Toolkit

Introduction

In recent years, understanding of stress pattern in dental structures has been of great interest. There have been many objectives to determine stress distribution in dental structures. Stress analysis of dental structures not only help dentist to understand the orthodontic biomechanics involved in various processes such as masticatory and Skeletal Anchorage System (SAS) but also helps to investigate the physical, chemical and biomedical characteristics of different types of materials. Conventionally, experimental stress analysis techniques using strain gauges have been used to understand stress distribution and improvement of the mechanical strength of such dental structures [1, 2]. Many researchers [3-5] have also used photo elastic techniques to understand stress distribution under various loading conditions. While analytical approaches are not available for complex dental structures, experimental approaches are expensive and time consuming. Due to irregular geometry, complex material properties and complicated loading conditions involved in dental structures, Finite Element Method (FEM) based techniques are effective to understand stress distribution in teeth. Since its first use in the orthopedic biomechanics in 1972 [5] various FEA models of teeth have been published. In last two decades there is tremendous increase in use of FEA tools due to advancement in computational techniques using commercially available tools such as ANSYS, Abaqus, LSDYNA, etc. However, most of these studies used 2D analysis approach with or without axi-symmetric approach [7-13] which do not reflect the actual scenario present in dental structures in terms of geometry,
Simulation Driven Innovation

material and loading condition. Detailed three-dimensional models helps to understand the mechanical behavior of teeth structures in better fashion. Hence, there have been efforts to represent different teeth with 3-dimensional approach using FEA tools but in most of the cases geometry of tooth is simplified due to its complex nature [14-16]. Although obtaining FEA solution for complex dental structures is not an extremely complex task given the availability of wide range of commercial FEA solvers but the creation of their exact 3D CAD model compatible to subsequent discretization in suitable pre-processor poses a real challenge. Initially, image analysis in dentistry was mostly confined to 2D BMP images, as scanning technologies such as Magnetic Resonance Imaging (MRI) and Computerized Tomography (CT) have improved, there has been an increasing use of three-dimensional images for clinical medicine and biomedical research in dentistry. Major issue in 3D image analysis is proprietary image formats and inconsistent export formats not only in dentistry but also in bio-medical field in general. Nowadays, there are commercially used image processing software’s which are free from such constraints but they are expensive and require profound training [17]. However, there are many open source tools are available to perform 3D image processing tasks leading to 3D visualization with compatible CAD export to FEA tools. Most famous are visualization toolkit (VTK) [18] and NLM insight toolkit (ITK) [18]which deliver modules for data processing and visualization that are publicly available and can be customized freely.

The objective of this paper is to reproduce actual 3D tooth model using VTK and FE analysis using HyperMesh and Radioss. HyperMesh and Radioss are capable of overcoming shortcomings of discretization and obtaining exact solution for exact tooth model under various loading conditions. This paper presents 3D model of two-root molar tooth created using VTK from CT scanned images. Such 3D models are exported in STL format to HyperMesh for further processing and stress analysis using Radioss for two loading conditions i.e. distalizing force and mascating force. Stress patterns are studied to understand mechanics of premolar tooth. Fig. 1 shows overall procedure used in this work to determine stress pattern in premolar tooth.

![Figure 1: Overall approach used in the present study](image-url)
Process Methodology
Typical FE-analysis for pre-molar tooth involves following steps:

- Geometric modeling: This step involves creation of geometric representation of pre-molar tooth. In present study VTK is used to obtain 3D model. 3D model produced using DICOM images is validated with actual tooth dimensions. This tooth model is exported using STL format.

- Meshing: This step involves sub-division of geometric model in discrete elements. The resulting set is called mesh. HyperMesh provides various options to produce 3D mesh on STL format of tooth model.

- Materials and LBC’s (Loads and Boundary Conditions): In this step, material, element properties along with loads and boundary conditions defined on discretized model of premolar tooth in HyperMesh.

- Analysis: In this step FEA solver Radioss is used to calculate displacement and stress due to two loading conditions i.e. mastication and distalization.

- Visualization of the results: This step involves visualization of various results such as VM stress and comparison of same with the design admissible limits of materials.

In this study premolar tooth is considered for further CAD modeling and FE analysis. The premolar teeth or bicuspids are transitional teeth located between the canine and molar teeth. In humans, there are two premolars per quadrant, making eight premolars total in the mouth[20], see Fig. 2. Premolars can be considered as a ‘transitional tooth’ during chewing, or mastication. A tooth consists of two main parts - the crown and the root. Enamel is white in colour. It is the hardest substance in the human body and covers the outer portion of the crown. It is made up of mineral salts (of calcium and magnesium) and keratin. It can withstand high pressure. Cementum is the layer present covering the root portion of the tooth. It is made up of mineral salts and water and is almost as hard as bone. Dentine is yellow and is a bone-like material which is present along the full height of the tooth. It is enclosed by the enamel in the crown portion and cementum in the root portion. Pulp is the innermost region of the teeth and shows the presence of blood vessels and nerve endings. This portion is very small as compared to the overall-size of the complete tooth and can be neglected in actual analysis.

![Figure 2](image.png)

*Figure 2: (a) Location of premolars in mouth (b) Structure of typical tooth [20]*
Fig. 3 shows series of DICOM images from CT scan for lower teeth of human mouth. Thirty five DICOM images were taken from CT scan device. From given DICOM images, premolar tooth have been identified and 3D model is produced using VTK and ITK open sources. Following procedure is used to transform DICOM (raster format) to STL (mesh). Iso-contouring is the most difficult task and simplified by 'segmentation' process using ITK.

- Read, calibrate and pre-process DICOM images
- Extract iso-contour with the help of segmentation to produce STL model
- Write STL file format for tooth model

Fig. 4a shows the final 3D model for premolar tooth produced from DICOM images of CT scan. Geometrical dimensions are validated against actual tooth by measuring key dimension. Table 1 shows comparison of dimensions. Fig. 4b shows imported STL file in HyperMesh which is further used for producing 3D mesh.

![Figure 3: Series of DICOM images from CT scan for lower teeth of human mouth](image)

<table>
<thead>
<tr>
<th>Major tooth dimensions</th>
<th>Actual</th>
<th>Generated of CT images</th>
</tr>
</thead>
<tbody>
<tr>
<td>Height (mm)</td>
<td>18</td>
<td>18</td>
</tr>
<tr>
<td>Perimeter of crown at middle (mm)</td>
<td>25</td>
<td>25.63</td>
</tr>
<tr>
<td>Perimeter of root (left) at middle (mm)</td>
<td>12</td>
<td>12.28</td>
</tr>
<tr>
<td>Perimeter of root (right) at middle (mm)</td>
<td>11</td>
<td>11.30</td>
</tr>
</tbody>
</table>

*Simulation Driven Innovation*
Next step after 3D model generation is to generate suitable mesh using STL model data. STL (STereoLithography) is a file format native to the stereolithography CAD software created by 3D Systems. This file format is supported by HyperMesh and is also widely used for rapid prototyping in dentistry. STL files describe only the surface geometry of a three dimensional object without any other common CAD model attributes, see Fig. 5a. The STL format specifies both ASCII and binary representations. As STL model is not suitable to carryout 3D FE analysis on tooth, hence following steps are performed in HyperMesh to generated 3D tetrahedral mesh for premolar tooth.

- Import STL file
- Create surface from available triangulated mesh data
- Create solid (volume) from surfaces
- Generate 3D tetra mesh on volume as shown in Fig. 5b

There are 18158 tetra elements and 4684 number of nodes. It is highly impossible to achieve perfect quality parameters for mesh generated for such complex geometry. However, reasonably good quality parameters are maintained in the model in order to get quality results. The mechanical properties of tooth materials are used from past literature [16, 21-24] and given in Table 2. Two loading conditions have been considered in this paper. Typical load values have been taken from past literature to study stress pattern in premolar tooth. Table 3 shows LBC’s for mastification and distalization cases.

**Table 2: Mechanical properties of premolar tooth [16]**

<table>
<thead>
<tr>
<th>Material Property</th>
<th>Young’s Modulus (GPa)</th>
<th>Poisson’s Ratio</th>
<th>No. of Elements in FE Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Enamel</td>
<td>84.1</td>
<td>0.3</td>
<td>7978</td>
</tr>
<tr>
<td>Dentine</td>
<td>13.7</td>
<td>0.3</td>
<td>7648</td>
</tr>
<tr>
<td>Cementum</td>
<td>18.6</td>
<td>0.31</td>
<td>2532</td>
</tr>
<tr>
<td>Pulp [24]</td>
<td>0.002</td>
<td>0.45</td>
<td>This region is neglected</td>
</tr>
</tbody>
</table>
**Figure 5:** Meshed model (a) STL model (b) 3D tetra mesh in HyperMesh

**Table 3:** Different loading conditions for premolar tooth analysis

<table>
<thead>
<tr>
<th>Load Case</th>
<th>LBC</th>
<th>Model</th>
</tr>
</thead>
</table>
| Mastification [16] | P = 100 N vertically downwards (Blue)  
BC = Root fixed in all directions (Red) | ![Mastification Model](image1) |
| Distalization [25]  | P = 2 N Horizontal (Blue)  
BC = Root fixed in all directions (Red) | ![Distalization Model](image2) |
Results & Discussions

Fig. 6a shows the VM stress distribution in premolar tooth during mastication process and load due to mastication is axial in nature. Stress increases from crown to root. Maximum VM stress is near root and maximum value is 40.10 MPa which is less than permissible limit of 48 MPa [26]. From stress distribution it appears that more stress is induced in cementum region. Fig. 6b shows the VM stress distribution in premolar tooth due to distalization load which is bending in nature. The stresses due to distalization force is very less as magnitude of force is also very low. In this case also maximum stress is observed in near root and maximum value is 1.61 MPa. From stress distribution it appears that stress equally distributed in dentine and cementum.

Conclusions

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 premolar tooth is successfully carried out on premolar tooth model using FEA tool – HyperMesh and Radioss under two different loading conditions i.e. mastication and distalization. The FEA model neither simplifies geometry nor makes any assumptions in material. Hence, the developed FEA model is most accurate.
From stress distribution it appears that stress during mastification is higher as compared to loading due to distalization. This is oblivious because load experienced by teeth during mastication is much more than distalization. However, during both the loading situations stress concentration is observed in near root of the tooth and stress is typically observed in cementum and dentine.

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

Similar FEA based approach can be used in variety of applications in dentistry:
- To investigate alternative materials for artificial teeth
- To study effect of various loadings on teeth due to Skeletal Anchorage System
- To understand biomechanics of dental structures for variety of clinical and biomedical research
- To investigate optimal designs for dental implant

**ACKNOWLEDGEMENTS**

Authors acknowledge contribution of Dr. Ravi Kumar GVV for his valuable guidance. Authors also acknowledge support of Mr. Devaraja Vaderaaholi Holla, Mr. Avinash Patil and Mr. Ramanath Suryaprakash during this work.

**REFERENCES**

[18] www.vtk.org
[19] www.itk.org