Physical Effects on an Assembly of Modeled Teeth and Mandible Using Finite Element Analysis

Sheeraz Athar, Ahmad Raza Usmani, Abid Ali Khan, Prabhat Kumar Chaudhari, and Mohd. Tariq

Abstract The basis of this study is to provide an idea about finite element analysis (FEA) applied to orthodontic treatments. This paper employs FEA to investigate the stresses developed on tooth enamel and adjoining tissues when subjected to a certain load. The analysis is done by computer-aided engineering (CAE)-based software Autodesk Inventor©. For simulation, 3-D models of teeth were created in the shape of truncated pyramids. These modeled teeth were then assembled with the CAD model of human mandible. After the analysis, the software generated a report depicting graphical distribution of stress, strain, and safety factor over the assembly of mandible. In our study, all the physical parameters fall under safe limit, hence causing no damage/injuries if clinically demonstrated. Any other loading condition can easily be analyzed by feeding the required data. Novelty of the proposed approach is in the approximate 3D modeling of teeth based on the knowledge of its dimension, without taking the actual model into consideration. This methodology can be extended to a wider range as this is cost effective and simple.

Keywords Mandible · Orthodontic treatments · Finite element analysis · Simulation

S. Athar (✉) · A. R. Usmani · A. A. Khan
Department of Mechanical Engineering, Aligarh Muslim University, Aligarh 202001, U.P, India
e-mail: shirazathar2@gmail.com

A. R. Usmani
e-mail: ahmadrazausmani@gmail.com

A. A. Khan
e-mail: abidak71@yahoo.com

P. K. Chaudhari
Division of Orthodontics and Dentofacial Orthopedics, Centre for Dental Education and Research, All India Institute of Medical Sciences, New Delhi 110029, India
e-mail: dr.prabhatkc@gmail.com

Mohd. Tariq
Department of Orthodontics and Dentofacial Orthopedics, Aligarh Muslim University, Aligarh 202001, U.P, India
e-mail: jmi786jmi@gmail.com

M. Muzammil et al. (eds.), *Ergonomics for Improved Productivity*,
Design Science and Innovation, https://doi.org/10.1007/978-981-15-9054-2_10

1 Introduction

In the era of modern technology, it is very important to analyze and explain the mechanism of orthodontic appliances through technical methods based on sound scientific principle, before their clinical use.

In the past, stresses in dental structures have been studied by various techniques, e.g., brittle coatings analysis, strain gauges, holography, two- and three-dimensional photoelasticity, and numerical methods [1]. Out of these methods, photoelastic technique is most widely used [1]. However, it requires a birefringent material which is not compatible with complex geometries [1].

The finite element analysis (FEA) is a tool that may lead to better understanding of the complexities and the stresses produced in the human tooth when particular forces are applied to it. This methodology will ultimately be beneficial in performing surgeries and research as it uses mathematical functions that enable orthodontists to understand various convoluted aspects of orthodontics, particularly the effect of orthodontic appliances on teeth and associated structure (e.g., jaw bone) in the form of stress generations.

Orthodontic simulation has helped the dentists to understand the loading conditions in which a tooth can be operated. This paper performs a trial for a new approach in FEA, i.e., using the 3-D model of teeth, and presenting it before the orthodontic research community to investigate the potential of this method in solving realistic problems.

The objective of this study was to create 3D models of teeth based on the knowledge of its dimension for FEM analysis as a tool to understand the effect of orthodontic appliances for experimental study of orthodontic tooth movement.

2 Procedure

The study starts with a 3D CAD model of the mandible. This CAD model also includes teeth in it; hence, these teeth are its integral part, but to work in an FEA environment, both teeth and jaw should be different entities so that the relative motion and forces can be easily analyzed.

Therefore, to tackle this situation, an assembly of teeth and jaw was needed. For this, modeling of the first, second, and third mandibular molars were done in our research using 3D CAD software Autodesk Inventor®. The design of teeth is based on the dimensions given in Table 1. For the simulation, 3D models of teeth were created in the shape of truncated quadrilateral pyramids, taking four diameters as the sides of quadrilateral and overall length as the height of the pyramid.

Tooth enamel has been taken as the material of these modeled pyramids and corresponding material properties were assigned [3–6].

Table 1 Dimensions of teeth [2]

Tooth	Overall length	Length of crown	Length of root	Mesiodistal dia. of crown	Mesiodistal dia. of crown at cervix	Labiolingual dia. of crown	Labiolingual dia. of crown at cervix
First molar	15.8	6.0	9.8	7.7	6.5	7.0	5.3
Second molar	18.8	5.5	11.3	9.9	7.2	8.7	6.4
Third molar	17.5	6.5	11.0	8.5	6.5	10.0	9.5

*all teeth are of the mandible. **dimensions are in millimeters

The teeth were then attached with the mandible, which created a 3D assembly containing jaw and teeth as two different bodies facilitating the FEA analysis of the orthodontic movements.

The material properties assigned to the tooth enamel are as follows (Table 2):

In FEA, first step is to create an assembly of mandible and teeth. For this, cavities were required in the mandible to accommodate teeth. Once the assembly is created, stress analysis can easily be done.

The procedure continued by assigning the material to different components of the assembly. Afterward, the software analyzes the behavior of different contacts in the assembly. Subsequently, constraints and loads were applied, and fixed constraints have been applied to the base of the mandible as shown in Fig. 1.

Loads were applied to the teeth in y-z plane at an inclination of 20 degrees with z-axis, as shown in Fig. 2.

Following this, mesh was generated, which divided the assembly into numerous 'finite elements.'

After creating the mesh and knowing further details of mesh view, basic computer simulation was done.

Table 2 Properties of tooth enamel [3–6]

Material property	Magnitude	Units
Thermal properties [3]		
Thermal conductivity	0.9	W/mK
Specific heat	750	J/Kg/K
Thermal expansion co-efficient	11.4×10^{-6}	/K
Mechanical properties [4–6]		
Young's modulus	1338.2 ± 307.9	MPa
Poisson's ratio	0.28	
Shear modulus	522.656	MPa
Density	2.92	g/ml
Tensile strength	42.2	MPa

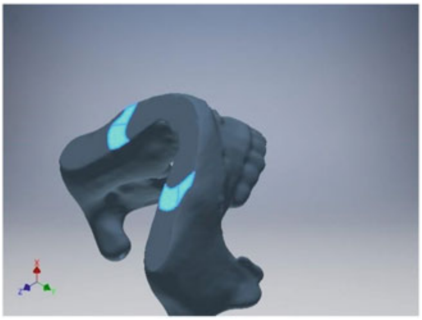

Fig. 1 Fixed constraints on the base of mandible

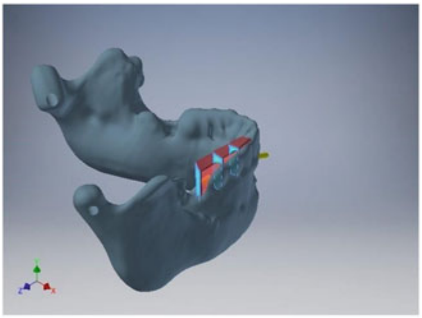

Fig. 2 Loads applied to the teeth

3 Result

After the finite element analysis, following results were concluded:

- Distribution of von Mises stresses over different positions on the assembly, i.e., it shows the minimum and maximum values of stresses that were generated by the applied load as shown in Fig. 3(i).
- Distribution of displacement of various assembly components, i.e., maximum and minimum displacement of different components of the assembly as shown in Fig. 3(ii)
- Variation of principal stresses throughout the assembly as shown in Fig. 3(iii).
- An idea of safety factor which helps in determining whether the applied stresses are within the yield strength of the material, which ultimately provides an idea about the chances of tissue damage as shown in Fig. 3(iv).

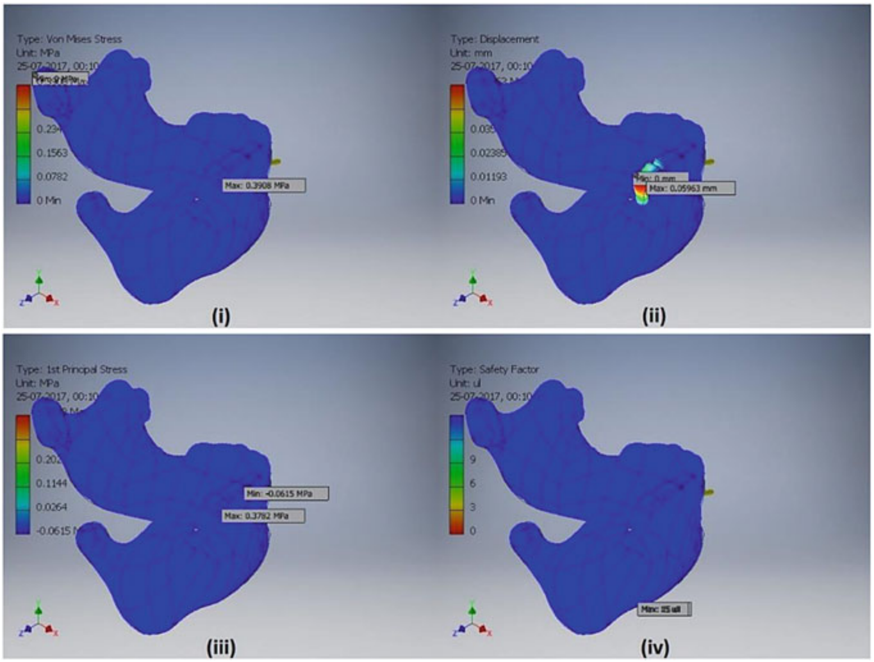

Fig. 3 Variation of (i) von Mises stress; (ii) displacement; (iii) principal stress; (iv) safety factor

4 Future Prospect

By demonstrating a model of an assembly of the human lower jaw and analyzing the stresses caused by different applied loads, a range of the maximum stresses experienced by teeth, without any injury, is achieved. Utilizing this information, various instruments can be made considering the maximum forces bearable by teeth. Several complex surgeries can also be made successful by predicting the complications and providing a better way to solve the case. Moreover, in a country like India, where the number of orthodontic experts is very few, this method can provide a quality treatment even under the guidance of an inexperienced dentist. However, some technical skills will be required to control the software setup.

The future goal of this research is to devise a complete clinical methodology incorporating FEA technology that can be used at a dentist's clinic.

A much detailed and practical model was the aim, but the software needed some clinical data which was not available. So, various assumptions were made which ultimately affected the practicality of the model. Hence to improve this model, details of loading conditions would be required, i.e., the exact magnitude and direction of different loads acting on the teeth under various conditions.

References

1. Vasudeva, G .: Finite Element Analysis: A Boon to Dental Research. Internet J. Dental Sci. 6(2) (2008)
2. Black, G.V.: Descriptive Anatomy of the Human Teeth. 4th edn. S.S White Dental Company, Philadelphia (1897)
3. http://biofiz.semmelweis.hu/run/dl_t.php?id=935&tid=41. Last Accessed 22 Aug 2017
4. Ang, S.F., Scholz, T., Klocke, A., Schneider, G.A.: Determination of the elastic/plastic transition of human enamel by nanoindentation. Dent. Mater. **25**(11), 1403–1410 (2009)
5. Weidmann, S.M., Weatherell, J.A., Hamm, S.M.: Variations of enamel density in sections of human teeth. Archives Oral Biol. **12**(1), 85–97 (January 1967)
6. Giannini, M., Soares, C.J., de Carvalho, R.M.: Ultimate tensile strength of tooth structures. Dent. Mater. **20**(4), 322–329 (2004). May