Dynamic Finite Element Analysis of the Human Maxillary Incisor Under Impact Loading in Various Directions

Haw-Ming Huang, MS, Keng-Liang Ou, PhD,† Wei-Nang Wang, DDS,‡,§ Wen-Ta Chiu, MD, PhD,* Che-Tong Lin, DDS, PhD,† and Sheng-Yang Lee, DDS, PhD**

Abstract

The aim of this study was to investigate fracture patterns occurring when a human upper central incisor is subjected to impact loadings at various angles. A two-dimensional finite element (FE) model of the maxillary incisor and surrounding tissues was established. The structural damping factor for the tooth was then calculated and assigned to the model. Dynamic FE analysis was performed to stimulate the associated impacts. Time-dependent traumatic forces at 0°, 45°, and 90° labially to the long axis of the tooth were applied to the model. Von Mises's equivalent stress contours within the FE models were calculated. Our results indicated that tooth damping lagged behind peak stress by 0.05 ms. In addition, we found that impact direction played an important role in terms of outcome for the fractured incisor. These results can, in part, explain the mechanisms underlying the alternative outcomes when upper incisors are subjected to impact.

Key Words

Finite element, fracture, incisor, dynamic analysis, stress

From the *Graduate Institute of Oral Sciences, †Graduate Institute of Oral Rehabilitation Sciences, ‡Institute of Injury Prevention & Control, Taipei Medical University, Taipei, Taiwan; and §Department of Neurosurgery, Municipal Wan Fang Hospital, Taipei, Taiwan.

Address requests for reprint to Dr. Lee, Sheng-Yang, DDS, Graduate Institute of Oral Rehabilitation Sciences, Taipei Medical University, 250 Wu-Hsing Street, Taipei, Taiwan. E-mail address: seanlee@tmu.edu.tw.

The upper central incisor is the most frequently involved tooth in frontal impact. Clinical findings demonstrate that the outcomes for frontal tooth impacts typically involve crown, oblique root, oblique crown-root, or neck fractures (1). Although theoretical study indicates that force direction plays an important role in the propagation of fracture lines in the impacted tooth (2), the exact relationship between the angle of impact and the resultant fracture lines in the maxillary incisor remain unclear and without experimental evidence.

With the rapid advancement and development of computer technology, finite element analysis (FEA) is now widely used as an effective method for dental trauma analysis. Most FEA studies are simplified and involve a static force applied to the tooth (3–7), however, with the applying force assumed to be unchanged during the impact period and the damping effects of the tissues ignored. In the real world, however, traumatic injuries to the teeth typically result from a dynamic force, the magnitude of which is altered over time. Therefore, for traumatic analysis of the tooth, time-dependent behavior should be considered for different rates of loading (8). To assess the process of stress growth and fracture-line propagation in an impacted tooth, dynamic FEA can provide greater insight into the issue.

To conduct dynamic FEA, the viscoelastic properties of the test subject are needed for computation. The viscoelasticity of a material can be separated into two components: one is a perfect elastic solid, and the other is a viscous liquid. When a viscous material is subjected to an impact, the strain energy can be gradually converted to another energy form. Because of the reduction in the strain energy, the response, such as the deformation of the material, gradually decreases. The mechanism by which the strain energy is gradually converted to another energy form is known as damping. When an impact force is applied, the resultant stress of an elastic material is directly proportional to the strain. However, the resultant stress caused by an external strain is dictated by the rate of deformation (9). Therefore, damping is a nonlinear material property of a viscous material.

Because of the lack of quantitative scientific data, however, few dental studies have assigned nonlinear mechanical properties to their FE model (8, 10–12). In these nonlinear studies, the damping properties of periodontal ligament (PDL) and the intact tooth were evaluated by means of curve-fitting the experimental data to the vibrational behavior of mechanical models. Analyzing the results from these studies reveals that nonlinear FE analysis overcomes the problems inherent in approximation resulting from the adoption of simplified models.

Although the damping property of PDL is the main contributor to tooth viscoelasticity, the cushioning effect of other damping material, such as pulp, can affect internal stress distribution within the impacted structure. To better understand the viscous properties of the human tooth, in our previous study the damping ratio of the human maxillary central incisor was quantified by means of modal testing experiments (13). In the current investigation, we incorporated this experimental damping-ratio data for the incisor into an FE model. Dynamic FEA was used to investigate the stress concentrations and fracture-line propagation in an upper central incisor subjected to dynamic loads in various directions.

Materials and Methods

In this study, the finite element analysis package, ANSYS (Swanson Analysis System Inc., Houston, PA), was used to perform the transient dynamic analysis on a personal

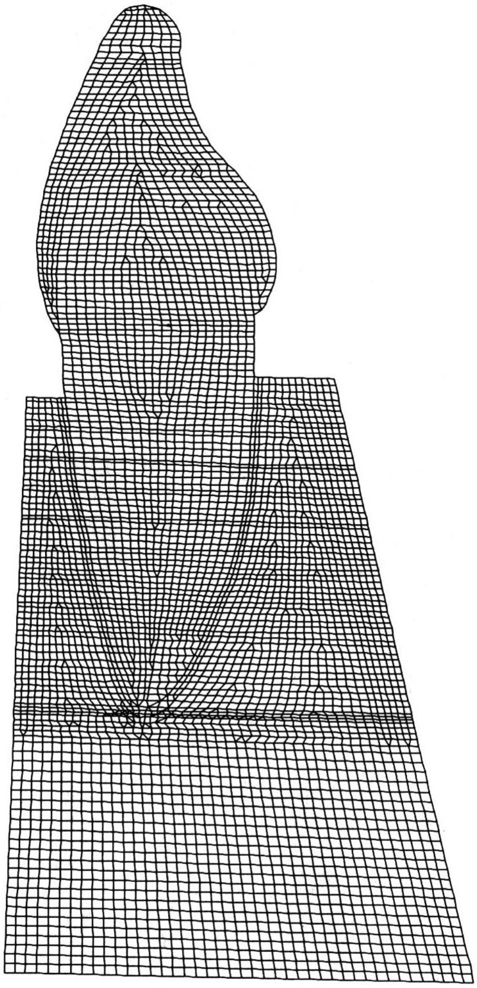

Figure 1. The 2-D strain finite element model used in this study.

computer. Transient dynamic analysis (also called time-history analysis) is a technique used to determine the dynamic response of a structure under the action of any general time-dependent loads. This technique is used for structural analysis of damping effects are considered to be important. To mesh the FE model more finely, a 2-D plane strain FE model of the human maxillary central incisor, containing enamel, dentin, pulp, periodontal membrane, alveolar bone, compact bone, and spongy bone was established (Fig. 1). The geometry and relevant dimensions of the incisor, including the length (23.5 mm) and thickness of the periodontal membrane (0.25 mm) were obtained from an atlas with anthropometry data sourced from a previous study (14). The alveolar process was located 2 mm apically from the cementoenamel junction (CEJ). Our model had a total of 5373 nodes and 5274 2-D quadrilateral elements, with the boundary conditions defined to prevent free body motion. The nodes on the base surface of the alveolar bone were fixed.

As shown in Table 1, the material properties for the FE model were adopted from the literature (14). In addition, the structural damping factor (β) for the model was derived according to the following formula:

$$\beta = \frac{\xi}{\pi f} \quad (1)$$

where ξ is the damping ratio, and f is the first resonance frequency of the upper central incisor (15, 16). In this study, the damping ratio

TABLE 1. Material properties used in the finite element model

	Young's Modulus (GPa)	Density (g/cm^3)	Poisson's Ratio
Enamel	77.90	3.00	0.33
Dentin	16.6	2.20	0.31
Pulp	0.00689	1.00	0.45
PDL	0.05	1.10	0.45
Alveolar Bone	3.50	1.40	0.33
Cortical Bone	10.00	1.40	0.26
Cancellous Bone	0.50	1.40	0.38

(14.6%) (13) and resonance frequency of human maxillary incisor (1388 Hz) (17) were adopted from experimental data. Accordingly, the structural damping factor of the upper central incisor assigned in our model was 0.33×10^{-4}.

A sinusoidal force with a peak of 800 N (18, 19), a rise time of 2 ms, and a total duration of 4 ms (20, 21), was chosen and imposed on a node on the facial crown. To assess the influence of impact direction on stress distribution and fracture-line propagation in the impacted incisor, three impact forces were applied using the model (Fig. 2): F1 simulated the traumatic force and acted horizontally to the labial crown; F2 was a traumatic force at 45° labial to the incisal edge; and F3 was a vertical force acting at the labial middle crown. In the various impact situations, Von Mises' equivalent stress contours within the FE models were displayed for comparison. The stress distribution at the root apex,

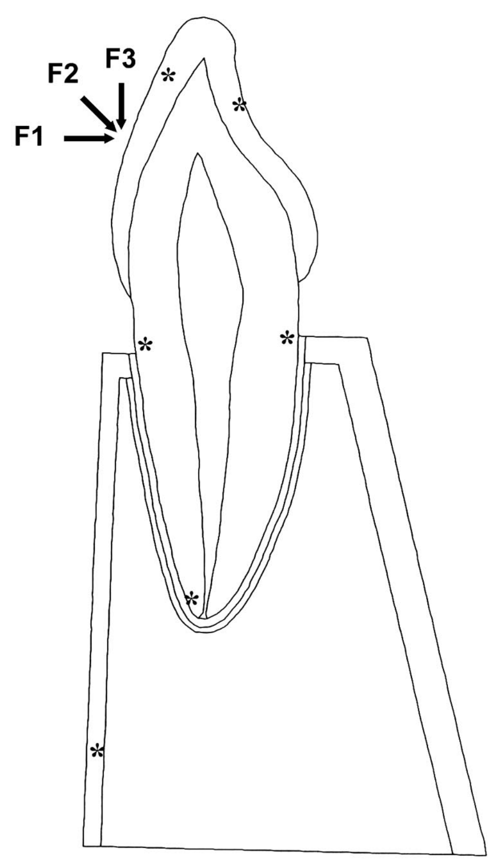

Figure 2. Three loading forces are applied to the labial middle crown of the model. F1, F2, and F3 represent loading at 0°, 45°, and 90°, respectively. Asterisks denote locations where Von Mises' stress was computed for comparison.