Development of a Finite Element Analysis Model With Curved Canal and Stress Analysis

Ran Cheng, MDS, Xue-Dong Zhou, PhD,*‡ Zhan Liu, PhD,† and Tao Hu, PhD*‡*

Abstract

This study aimed to establish a model of a curved canal by finite element analysis (FEA). To develop a repeatable and comparable model, simulated curved canals with uniform shape were selected as prototype canals, and a suitable extracted single-root tooth was chosen as the outline. Subsequent combinations and modifications were performed by using the analysis program ABAQUS. By using a series of imitational occlusive forces loaded onto the incisive edge of the model, color plots and the maximal stresses at the apical region were analyzed. Stresses increased when the angle of the loads rose, and loads in the distal and mesial directions induced more stresses in the target region than did loads in the buccal and lingual directions. This study has established a standardized FEA model with a curved canal. Consequently, this model may be applied for future estimations in endodontic procedures for a curved canal. *(J Endod 2007;33:727–731)*

Key Words

Finite element analysis, microfocus computed tomography, simulated canals

From the *Department of Operative Dentistry and Endodontics, West China College of Stomatology, Sichuan University, Chengdu, China; †Laboratory of Biomechanical Engineering, Sichuan University, Chengdu, China; and ‡Key Laboratory of Oral Biomedical Engineering Ministry of Education, Sichuan University, Chengdu, China.

Supported in part by National Key Technologies R&D Programme of the Tenth Five-Year Plan of the Ministry of Science and Technology, China (2004BA720A23) and Innovation Fund of Sichuan University.

Address requests for reprint to Dr. Tao Hu, Key Lab for West China College of Stomatology, Sichuan University, Chengdu, China, 14# 3rd section, Renmin South Road, Chengdu 610041, P.R. China. E-mail address: acomnet@263.net.

0099-2399/$0 - see front matter

doi:10.1016/j.joen.2007.02.015

Vertical root fracture (VRF) is a major clinical problem. Several studies have proved that, regardless of whether VRF occurs during or after an endodontic procedure, a thin dentin resulting from excessive preparation is an important factor (1, 2). Furthermore, other factors, such as internal and external root morphology have potential to influence fracture susceptibility. However, among these reasons, dentin thickness is regarded as the key factor (3, 4).

In the clinic, root canal therapy for a curved canal is an important problem, especially during the preparation stages, because there is a tendency to produce unwanted alterations in the canal shape. If this results in thin dentin in some parts, VRF is likely to happen. Stainless steel files and traditional preparation techniques often lead to transportation, ledge formation, and perforation (5); however, improved instruments and techniques have been applied that can effectively decrease aberrations in canals and keep the original shape (6).

Over decades, alterations in canal shape and obturation at the root apex have been major evaluation points in comparisons between various techniques. It has been proven that excessive preparation at the curvature and at the apex causes transportation and enlargement of the canal orifice (7–11). Whether these changes have influences on stress distribution in pulpless teeth or whether the thin dentine at the curve site or root apex are likely to be the initial part of the VRF is unknown. The objectives of this study were (1) to establish a standardized finite element analysis (FEA) model with a curved canal that can be used in comparisons of different preparations and (2) to investigate the stress distribution around the curved part under several loadings.

Materials and Methods

Development of the FEA Model

The contour of a tooth can be divided into two parts: the outline form and the inner canal. Accordingly, the FEA model was obtained from prototypes of a canal and a tooth. A simulated canal (Dentsply Maillefer, Ballaigues, Switzerland) was selected as the canal, and a mandibular incisor with a curved single root, extracted for routine clinical reasons, was chosen to be representative of the typical outer morphology.

The simulated canal had a working length of about 17 mm. The beginning of the curve was at 7 mm from the canal orifice and the degree of curvature was about 38°. The radius of the apical orifice was 0.15 mm, with a taper of 0.02. As judged by eye and X-ray, a mandibular incisor, the root of which had a similar form to the simulated root canal, was selected from collected teeth. It was fixed with wax to maintain position so that the curvature of the canal was parallel to the simulated canal.

All scans were undertaken by using a microfocus computed tomography scanner (Institute of Applied Electronics, China Academy of Engineering Physics, Mianyang, China). The first scanning procedure was completed for the tooth using settings of 220 kV, 100 μA, a focus of 5 μm, and a slice sickness of 100 μm. The second procedure was for the simulated root canal, using settings of 220 kV, 100 μA, focus of 5 μm, $3 \times 3 \times 3$ magnification, and a slice sickness of 42.3 μm. In all, 220 images of the tooth and 360 images of the canal were collected. These two-dimensional data were saved in a computer in BMP form and were reconstructed to give the 3-dimensional model (Fig. 1A and B). According to the anatomical guidance, two 3-dimensional models were put together to give a rough model. After proper modification, a final model was completed with a smooth surface and a mostly centered canal (Fig. 1C).

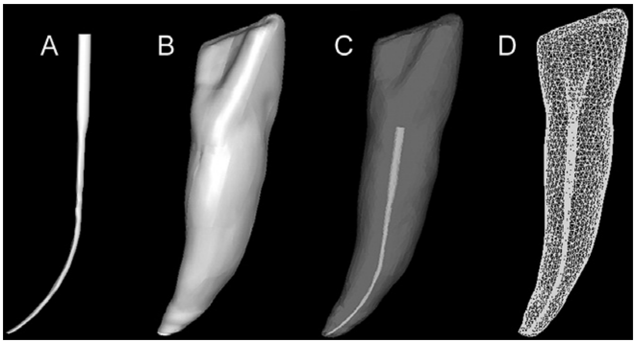

Figure 1. The process of development of a FEA model: (A) canal's model, (B) tooth's model, (C) combined model, and (D) meshed FEA model

Using clinical guidelines, the model was altered to accommodate an endodontic treatment. A reverse cone-shaped opening was set in the middle of the lingual fossa, with its end connected with the top of the canal (Fig. 1D). To give a blank sample for future use, the canal was filled with gutta percha instead of air from the apex to root canal orifice, with 1 mm of zinc phosphate cement on it and amalgam on the surface. The thin cementum was neglected, and the wall of the root was assumed dentin only. A periodontal ligament was modeled as a 250-μm thick shell surrounding the root and finishing 1.5 mm apical to the cementoenamel junction. A cube of alveolar bone was modeled around the periodontal ligament. Typical properties of dentin and other tissues were taken from the literature (Table 1) (12).

Boundary Conditions and Applied Load

The tooth structure was loaded within its elastic range, and static linear analysis was performed. All loads were applied slowly and gradually until they came to the maximal values. When the loads remained constant and the calculations of the finite element model showed no variation, the FEA was performed with FEA software (ABAQUS6.5; ABAQUS Inc, Providence, RI).

The loading patterns were selected to simulate occlusive forces. The model was loaded in four directions (buccal, lingual, mesial, and distal) at 0° (vertical), 30°, 45°, and 60° to the longitudinal axis of the tooth. All of the loads were transmitted onto the middle 2×2 mm^2 area of the incisal edge.

Results

Figure 2 show that stresses decrease from the crown to the apex. At a horizontal level, the maximal stress value appeared around the internal wall of dentine. However, none of the occlusive loads led to concentrated stress distribution in the lower part of root.

A comparison of the different loads indicates that the FEA model shows reduced stress when vertically loaded (Figure 3). Increased angles resulted in higher stress values, especially in the regions above 4 to 6 mm, but the differences were small around the apical 4-mm region. When the same force was loaded in four different directions (buccal, lingual, mesial, and distal), higher stresses tended to appear with distal loading, then mesial loading, and finally lingual and buccal loading (Figure 4).

Discussion

In previous studies, original models had been modified to simulate endodontic procedures. For example, the thickness of dentin changed as an effect of preparation (1–4). It was easier to have these changes in cross-section. Moreover, a model with a straight canal was suitable for modifications because major changes of canal diameter were foreseeable (13–15). However, it is not fit for curved canals. First, curved canals have complicated alterations immeasurable by existing instruments, which denies the possibility of accurate modification of models. Second, the variations of curved canals, including the degree of curvature and the curvature direction and position, make canal systems complicated. To decrease the possible error, more samples are required, which results in a huge workload in FEA.

Therefore, using similar teeth as a prototype is necessary for FEA models. For the reasons mentioned earlier, it is not possible to use extracted teeth. Instead, standard simulated canals of uniform shape are acceptable. In this study, a new method of establishing FEA models was used. First, a series of simulated canals, which represented a type of simple canal (16), was selected as the canal shape. Then, a matching extracted tooth provided the outline of the model. There were some computer-generated modifications to the outline; the two parts were combined to give a 3-dimensional model. This new FEA model also has the advantage that the original canal can be replaced. The same outline form and the standard shape of canals largely decrease the possible error in experiments. In this way, it is possible to have a number of comparable and uniform FEA models.

The FEA model in this study is based on a mandibular incisor with a distally curved root and an elliptical cross-section, buccolingually longer in diameter. Four typical directions were selected as loading directions to simulate occlusive courses and the load used (500 N) (17) was close to the average value at the molar. The analysis provides an insight into the stress distribution of a curved canal, but it only represents a medium-curved canal, and the results should be seen as qualitative not quantitative because there will be differences between canals.

Compared with the studies of straight canals, the results have something in common. For example, the research about an endodontically prepared maxillary central incisor showed the highest stress magnitudes located between the middle and coronal thirds of the root (18). Hong et al. (13) found the stresses decrease from the loading site to the apex. Besides, there are two more findings. First, the maximal stresses in the apical 4-mm region show few changes. Second, an increase is seen coronal to this region. The possible reasons lie in the increasing angle of loading, which transmits higher lateral forces. From the loading in the four directions, it is evident that the maximal stresses appear with distal loading. This is probably because of the curvature coinciding with the direction of loading and the elliptical shape of the cross-section. In this study, the stresses are less than the reported tensile strength for dentine (50-100 N/mm^2) (19, 20). However, they suggest when and where stresses increase.

In a previous study (21), microstructural defects were observed on teeth undergoing VRF (eg, dilations, ruptures, and a decreased number of dentinal tubules, which influence the resistance of dentine under loads). Ruptures probably start from a crack under a traumatic occlu-

TABLE 1. Material Properties

Material	Young's Modulus (N/mm^2)	Poisson's Ratio
Enamel	8.41×10^4	0.300
Dentine	2.00×10^4	0.310
Periodontal ligament	5.00×10^1	0.490
Alveolar bone	1.40×10^4	0.150
Cold gutta percha	3.00×10^2	0.485
Warm gutta percha	3.00×10^0	0.485