

## Finite Element Modelling in Premolar Teeth

C. Rivadeneira<sup>1</sup> and E. A. Bonifaz<sup>1\*</sup>

<sup>1</sup>Universidad San Francisco de Quito, Colegio de Ciencias e Ingeniería - El Politécnico.  
Calle Diego de Robles y Vía Interoceánica  
17-1200-841, Quito, Ecuador.

\*Autor principal/Corresponding author; e-mail: ebonifaz@usfq.edu.ec

Editado por/Edited by: F. J. Torres, Ph.D.

Recibido/Received: 13/09/2013. Aceptado/Accepted: 02/11/2013.

Publicado en línea/Published on Web: 09/12/2013. Impreso/Printed: 09/12/2013.

### Abstract

A three-dimensional finite element model was developed using the ABAQUS code to characterize dental systems in their different functional conditions, geometry and loading modes. The outputs of the model were used to evaluate quantitatively the magnitude and distribution of the stress-strain-displacement relations in sound (drilled and enamel crown) and restored with prefabricated intraradicular posts (conical and cylindrical) premolar teeth specimens. Under the same loading and boundary conditions, differences in stresses, displacements and elastic strains were observed for drilled, enamel crown, conical and cylindrical post teeth specimens. It was found that the cylindrical glass fiber post geometry minimizes much better the stress, displacement and elastic strain magnitude in premolar teeth. The results indicate that the finite element methodology is adequate and convenient for stress, displacement and strain assessment in orthodontics.

**Keywords.** Finite element, premolar teeth, stress-strain-displacement, prefabricated intraradicular post, periodontal ligament

### Resumen

Un modelo tridimensional para caracterizar sistemas dentales en sus diferentes condiciones funcionales, geometría y modos de carga fue desarrollado mediante el código de elementos finitos ABAQUS. Los resultados del modelo fueron utilizados para evaluar cuantitativamente la magnitud y distribución de las relaciones esfuerzo-deformación-desplazamiento en especímenes sanos y restaurados con postes intrarradiculares prefabricados (cónicos y cilíndricos). Bajo las mismas condiciones de carga y contorno, se observaron diferencias en esfuerzos, desplazamientos y deformaciones elásticas en especímenes agujereados, de corona esmaltada, y reforzados con postes cónicos y cilíndricos. Se encontró que el poste de fibra de vidrio con geometría cilíndrica minimiza de mejor manera la magnitud de los esfuerzos, desplazamientos y deformación elástica en dientes premolares. Los resultados indican que la metodología de los elementos finitos es adecuada y conveniente para evaluar esfuerzos, desplazamientos y deformaciones elásticas en ortodoncia.

**Palabras Clave.** Elemento finito, dientes premolares, esfuerzo-deformación-desplazamiento, poste intrarradicular prefabricado, ligamento periodontal

### Introduction

Finite Element Analysis (FEA) to determine stresses and displacements in teeth were first used in Dentistry in the 70's to replace photo elasticity tests [1]. In those days, coarse two dimensional (2D) finite element models had been developed by association of dentists and engineers. At the present, due to the development of new image acquisition techniques and the development of advanced technical design programs, the use of complex 3D models has significantly increased [1]. Nevertheless, experimental results (necessary to compare

the numerical counterparts) are very scarce due to the laboratory techniques used to analyze the mechanical behavior of sound or restored teeth are not very precise. The difficulties of using experimental or "in vivo" methods are the complexity of teeth geometry, the laboratory equipment exactness, the different involved materials, the different load modes, and the little availability of teeth extraction and patient participation [2–10]. On the contrary, FEA allows the creation of complex 3D models under complex load and boundary conditions schemes including direct restoration (amalgam and composite resin) and indirect restoration (dental crown

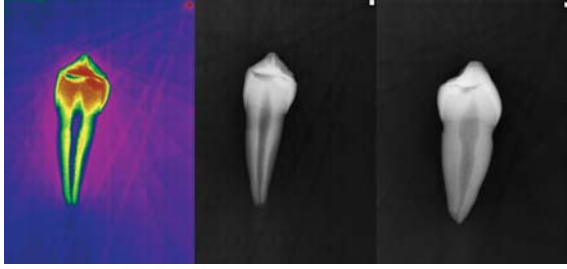


Figure 1: Premolar teeth radiographs obtained in the USFQ dental clinic [13].

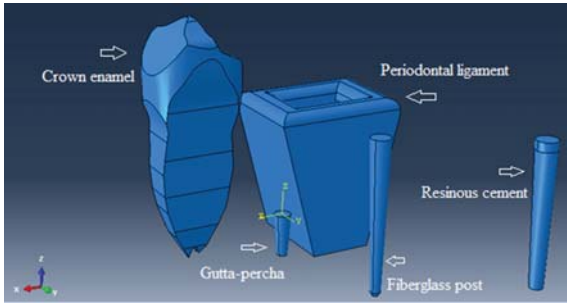


Figure 2: Internal and external components generated with CATIA to be used in finite element analysis assembly.

or fixed partial prosthesis). The developing of 3D models is also useful to verify the volumetric/geometric implications in results when compared with 2D models. In this work, the analysis of stress-strain distribution and displacements were performed with ABAQUS code [11]. An estimated load of 20 N per tooth (170 N in the total denture) was chosen to perform the simulations. The distributed load was applied over the palatal surface of the crown over a surface area of  $6.65E-5m^2$ .

### Models and methods

The geometry of the teeth was generated using CATIA code [12] according to the radiographs obtained from surgery in the USFQ dental clinic (Fig. 1).

The entire analysis was accomplished using a combination of internal and external components (Fig. 2). The internal components composed by the post endodontic gutta-percha, the fiberglass posts (cylindrical and conical), the crown enamel and the resinous cement which covers the top of the post up to the crown. The external component related with the periodontal ligament.

The components geometries (internal and external) generated with CATIA were imported into ABAQUSCAE

Material	Mechanical Properties	
	E (MPa)	$\nu$
Gutta-Percha	690	0.45
Resinocement	8000	0.25
Enamel	84100	0.33
Fiberglass post	11000	0.07
Periodontal ligament	68900	0.45

Table 1: Mechanical properties of the materials involved in the numerical analysis.

6.9 to create the 3D finite element meshes (Fig. 3). Constant material properties, distributed loads and appropriate boundary conditions were included in the FE pre-processing stage.

The FEM code ABAQUS used in all simulations computes dimensional variations and distortions of parts, strength and deformability of the material in use and residual stresses, during and the end of the loading process using the stress equilibrium equations. Following Dieter [14], and Dunne and Petrinic [15], a general frame for the model is given as follows:

The components of the tensor of total strain rate  $\dot{\epsilon}_{ij}$  are given by the sum of the elastic and plastic components, and  $\dot{\epsilon}_{ij}^e$ ,  $\dot{\epsilon}_{ij}^p$ , respectively:

$$\dot{\epsilon}_{ij} = \dot{\epsilon}_{ij}^e + \dot{\epsilon}_{ij}^p \quad (1)$$

The elastic part obeys Hooke's law and reads:

$$\dot{\epsilon}_{ij}^e = C_{ijkl} \dot{\sigma}_{kl} \quad (2)$$

$$C_{ijkl} = \frac{1 + \nu}{E} \left( \delta_{ik} \delta_{jl} - \frac{\nu}{1 + \nu} \delta_{ij} \delta_{kl} \right) \quad (3)$$

where  $C_{ijkl}$  is the elastic stiffness matrix,  $\dot{\sigma}_{ij}$  is the stress tensor and a dot denotes differentiation with respect to time  $t$ .  $E$ ,  $\nu$ , and  $\delta_{ij}$  are Young's modulus, Poisson's ratio, and Kronecker's delta, respectively.

The plastic part of the strain rate tensor is expressed in the form of the Levy-von Mises equation:

$$\dot{\epsilon}_{ij}^p = \frac{3}{2} \frac{\dot{p}}{q} \sigma'_{ij} \quad (4)$$

Here  $\sigma'_{ij}$  is the deviatoric stress tensor; the quantities:

$$\dot{p} = \left( \frac{2}{3} \dot{\epsilon}_{ij}^p \dot{\epsilon}_{ij}^p \right)^{\frac{1}{2}} \quad (5)$$

and

$$q = \left( \frac{2}{3} \sigma'_{ij} \sigma'_{ij} \right)^{\frac{1}{2}} \quad (6)$$

denote the equivalent von Mises plastic strain rate and stress, respectively. Substitution of the above equations into the equilibrium equation (6)

$$\frac{\partial \sigma_{ji}}{\partial x_j} = 0 \quad (7)$$

leads to the governing equation that represents the physical problem to be solved by means of the finite element method. The elastic-plastic problem is a boundary-value

Teeth specimens (studied cases)	Mises Stress (MPa)	Maximum Displacement (m)	Maximum Principal Strain
Drilled tooth	2.27	5.08e-4	2.79e-4
Enamel Crown	2.29	4.88e-4	2.65e-4
Cylindrical post	2.06	4.25e-4	1.62e-4
Conical Post	2.09	4.27e-4	1.93e-4

Table 2: Comparison of the magnitude of Mises stress, displacement and elastic strain found in different teeth specimens studied cases.

problem, requiring appropriate boundary conditions at each boundary point. Burnet [16], has shown for instance, that for a well-posed problem we must specify enough displacements (essential BCs) to prevent rigid-body motion of the entire structure. In the present study, appropriate mechanical boundary conditions were applied to represent the physics of the problem. The boundary conditions are located on the outer surface of the periodontal ligament (i.e., in the alveolar bone border). Displacements in X, Y and Z directions and rotations around the three axes were set to zero (see Fig. 4).

The von Mises stress  $q$  is, of course, a scalar quantity and its origin lies in the postulate that yielding occurs when a critical elastic shear energy is achieved, Dunne and Petrinic, [15]. The von Mises yield function is defined by

$$f = q - \sigma_y = \left( \frac{3}{2} \sigma'_{ij} \sigma'_{ij} \right)^{\frac{1}{2}} - \sigma_y \quad (8)$$

Here,  $\sigma_y$  is the yield strength at temperature  $T$ . The yield criterion is given by

- $f < 0$ : Elastic deformation
- $f = 0$ : Plastic deformation

This is a fairly general formulation accounted for in most of the commercial finite element codes. The use of von Mises equivalent quantities implies plastic isotropy of the material. In this investigation, an ABAQUS elastic deformation analysis was performed. The isotropic temperature non-dependent mechanical properties (modulus of elasticity  $E$  and Poisson's ratio  $\nu$ ) of the materials involved in the calculations are presented in Table 1.

## Results

The application of distributed forces causes stresses, strains and displacements in the whole teeth structure. The magnitude of these stresses, strains and displacements depends on the geometry, studied situation and boundary conditions. For instance, the inclusion of the periodontal ligament into the analysis produces different stress, strain and displacement distribution results. The reason is because the periodontal ligament allows smooth displacements of the internal components and consequently stresses relief. Figures 5, 6, 7 and 8 show Mises stress, displacement and elastic strain contours in four different studied situations. Results reveal that under the same loading and boundary conditions, differences in stresses, displacements and elastic strains were observed for drilled, enamel crown, conical and cylindrical post teeth specimens.

The four different situations (studied cases) were analyzed using a finite element mesh of 212015 ten node tetragonal C3D10 elements. The Mises stress, displacement and elastic strain maximum values generated under the action of the applied static distributed load (20N) are summarized in Table 2. These maximum values are obtained from Figures 5, 6 7 and 8. The small observed values correspond to the cylindrical post studied case (cylindrical geometry). Results of the present study obtained from 3D FEM analyses of sound and restored teeth, indicate that this methodology is adequate

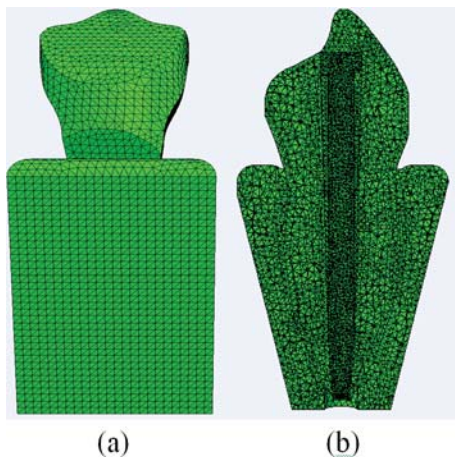


Figure 3: The finite element grids (a) lateral view (b) frontal view. Average element size 0.3 mm.

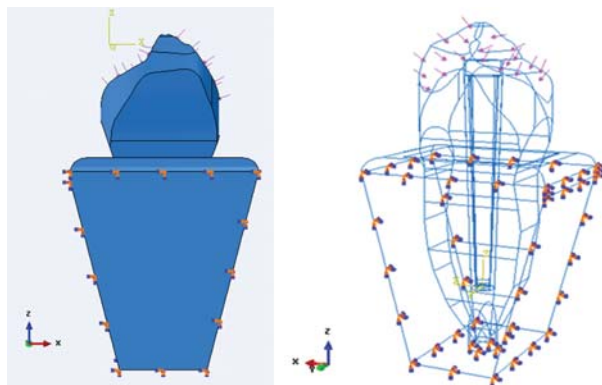


Figure 4: The finite element domain showing the distributed load applied over the palatal surface of the crown (surface area of  $6.65E-5m^2$ ), and boundary conditions located on the outer surface of the periodontal ligament.

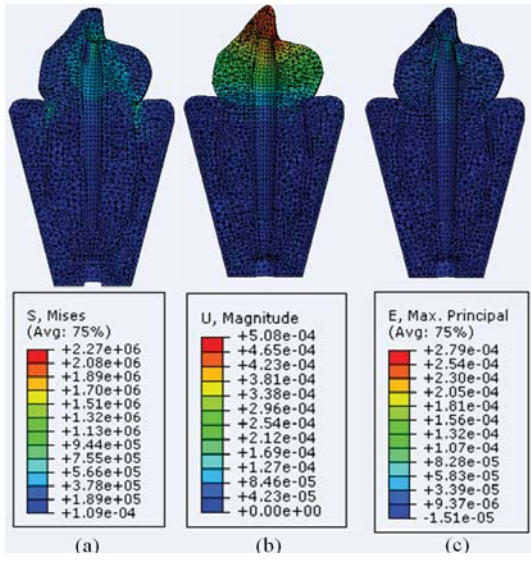


Figure 5: Stress, displacement and strain distribution in a drilled tooth specimen (a) Mises stress distribution (b) displacement distribution (c) elastic strain distribution.

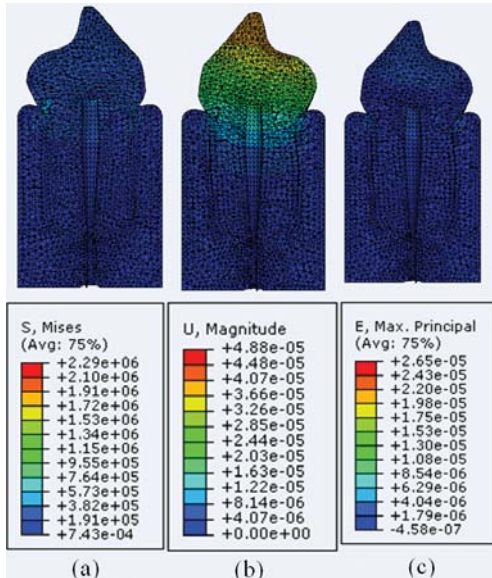


Figure 6: Stress, displacement and strain distribution in a drilled tooth specimen with enamel crown (a) Mises stress distribution (b) displacement distribution (c) elastic strain distribution.

and convenient for stress, displacement and strain assessment. The methodology enables an efficient approach to mechanical and fracture behavior of systems with complex geometry and with materials with different mechanical properties submitted to different loading modes. Therefore, FEM analysis is a powerful tool to characterize dental systems in their different functional conditions, being of great importance to both dentists and engineers, who are involved with the development of new materials and technical dental practices. In vitro tests of sound and restored teeth (static and cyclic loading under functional conditions) are being developed and the experimental results will be presented in a new publication. According to the numerical results, the best endodontic glass fiber post geometry that mini-

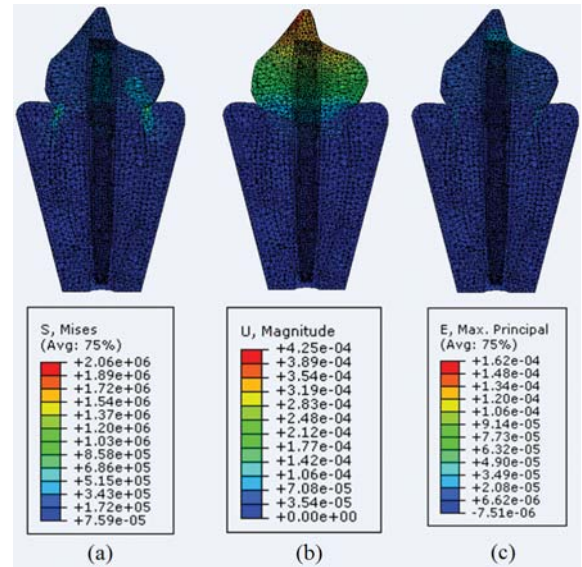


Figure 7: Stress, displacement and strain distribution on tooth with cylindrical glass fiber post (a) Mises stress distribution (b) displacement distribution (c) elastic strain distribution.

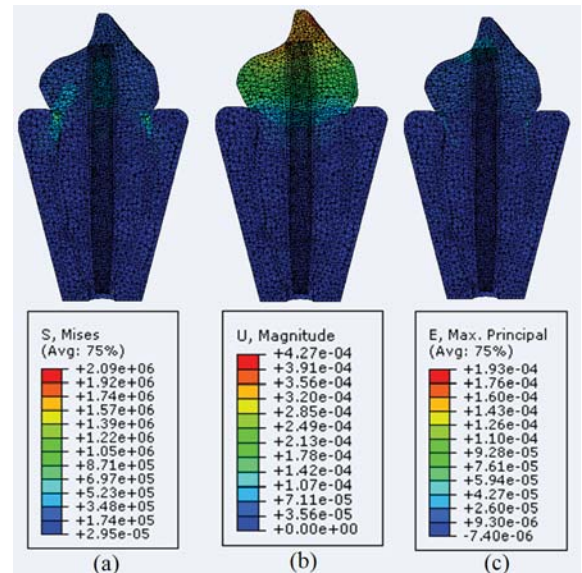


Figure 8: Stress, displacement and strain distribution on tooth with conical glass fiber post (a) Mises stress distribution (b) displacement distribution (c) elastic strain distribution.

mize stresses, displacement and strain distribution in premolar teeth, is the cylindrical one.

### Conclusions

A new finite element mechanical modeling procedure applied to 3-D grids was developed using the ABAQUS-FE code to characterize dental systems in their different functional conditions, geometry and loading modes. The outputs of the model were used to evaluate quantitatively the magnitude and distribution of the stress-strain-displacement relations in premolar teeth specimens. Under the same loading and boundary conditions, differences in stresses, displacements and elastic strains

were observed for drilled, enamel crown, conical and cylindrical post teeth specimens. The cylindrical glass fiber post geometry absorbs most of the external energy and consequently minimizes stresses, displacement and strain distribution in premolar teeth.

### Acknowledgments

The authors would like to thank to Ana Belen Parra from the USFQ dental clinic, Ecuador, who provided dental technical help and the premolar teeth radiographs presented in Fig. 1.

### References

- [1] Motta, A. B.; Pereira, L.; Da Cunha, A. R. C. C. “*Finite Element Analysis in 2D and 3D Models for Sound and Restored Teeth*”, *ABAQUS Users’ Conference*. 2006.
- [2] Barguil, J.; Chica, E.; Latorre, F. 2008. “Distribución de los esfuerzos en tres tipos de elementos intrarradiculares con diferentes longitudes”. *Rev. Fac. Odontol. Univ. Antioq.*, 19:24–37.
- [3] Bessone, I.; Fernandez, B. 2010. “Evaluation of different post systems: finite element method”. *Int. J. Odontostomat.*, 4:229–236.
- [4] Canalda, C. “*Endodoncia: Técnicas clínicas y bases científicas*”, volume Barcelona, España: Editorial Elsevier 2da edición. 2006.
- [5] Celigüeta, J. T. “*Método de los Elementos Finitos para Análisis Estructural*”, volume Tesis de maestría no publicada, Campus Tecnológico Universidad de Navarra, España. 2008.
- [6] Cohen, S. “*Vías de la pulpa*”, volume Barcelona, España: Editorial Elsevier 10ma edición. 2008.
- [7] Lopera, P.; Latorre, F.; Villaraga, J. A. 2012. “Evaluación no lineal de dos postes diferentes”. *Rev. Fac. Odontol. Univ. Antioq.*, 23:240–255.
- [8] Lopera, P.; Latorre, F.; Villaraga, J. A. 2010. “Estudio in vitro de la resistencia a la tracción de postes de fibra de vidrio cementados con cuatro agentes cementantes”. *Avances en odontoestomatología*, 5:255–262.
- [9] Saldarriaga, E. A.; Uribe, C.; Chica, E.; Latorre, F. 2009. “Distribución de los esfuerzos en tramos protésicos fijos de tres unidades con elementos intrarradiculares colados y prefabricados: análisis biomecánico utilizando un modelo de elementos finitos”. *Rev. Fac. Odontol. Univ. Antioq.*, 21:33–41.
- [10] Gómez, A. F.; Chica, E.; Latorre, F. 2008. “Análisis de la distribución de esfuerzos en diferentes elementos de retención intrarradicular prefabricados”. *Rev. Fac. Odontol. Univ. Antioq.*, 20:31–42.
- [11] Hibbit; Karlsson; Sorensen. “*ABAQUS v.6.9 Finite Element Software*”, volume ABAQUS, Inc. 2009.
- [12] Dassaults Systemes. “*CATIA code*”, 02/21/13.
- [13] Clínica Odontológica de la Universidad San Francisco de Quito. Pampite S/N, C. E.
- [14] Dieter, G. E. “*Mechanical Metallurgy*”, volume McGraw-Hill International Book Company. 1986.
- [15] Dunne, F.; Petrinic, N. “*Introduction to Computational Plasticity*”, volume Oxford University Press. 1988.
- [16] Burnett, D. S. “*Finite Element Analysis: From Concepts to Applications*”, volume Addison-Wesley Publishing Company. 1988.