ENGINEERING JOURNAL

Article

Effect of Periodontal Ligament on Stress Distribution and Displacement of Tooth and Bone Structure Using Finite Element Simulation

Thongchai Fongsamootr[a,*] and Pana Suttakul[b]

Department of Mechanical Engineering, Faculty of Engineering, Chiang Mai University, 239 Huay Kaew Rd., Muang District, Chiang Mai 50200, Thailand
E-mail: [a]tfongsamootr@yahoo.com (Corresponding author), [b]panacae@gmail.com

Abstract. Periodontal ligament (PDL) is a thin layer of collagen fiber that can absorb or reduce the transfer of stress from a tooth to the alveolar bone. To understand the function of PDL, stress distributions and displacement over the tooth and bone structure was simulated using Finite Element Analysis (FEA). To evaluate the credibility of FEA simulation, a simple model was created: a simply rod connection consists of tooth bar, PDL layer, and bone bar. It was performed with theoretical analysis and FEA using simulation to compare and validate the results. It was found that the theoretical analysis and FEA produced similar results. Therefrom, FEA was used to predict in the Orthodontics problem. Due to the complexity of dental geometry, the Computed Tomography was used to create the real tooth into the 3-dimensional model. FEA was then applied to study the role of the PDL layer with tooth and bone when realistic dental forces were applied using simulation models with and without PDL. The results showed that the maximum stress was higher and very small displacement in the model without the PDL layer. Thus, the PDL acts as a sustaining pad that decreases and intersperses the stress in the alveolar bone. Furthermore, the soft pad of the PDL layer allows the tooth to move more easily.

Keywords: Finite element analysis, periodontal ligament, stress distribution, dental simulation, tooth movement.

ENGINEERING JOURNAL Volume 19 Issue 2
Received 23 June 2014
Accepted 22 September 2014
Published 30 April 2015
Online at http://www.engj.org/
DOI:10.4186/ej.2015.19.2.99

DOI:10.4186/ej.2015.19.2.99

1. Introduction

Numerical simulation (using FEA) is widely used in several engineering fields to predict problems. Finite element analysis is a well-known method that can solve problems of complex geometry and loading conditions that are not easily solved analytically. FEA is an approximation method, with dividing the structure into various small elements that are sufficient to describe the geometry of the subjects, this is called Meshing that are connected by mesh intersections or nodes. The forces and boundary conditions are defined to simulate applied loads and constraint of the structure [1, 2, 3]. For example, with the freeform shape of a human tooth, finite element analysis can be used to compute the deformed shape of the tooth and supporting structures – cancellous bone, cortical bone, and periodontal ligament (PDL) – when subjected to dental forces [4].

PDL is a thin layer of collagen fiber between a tooth and the alveolar bone. It was conducted a sensitivity study found that PDL as linear elastic, isotropic and homogeneous material affects strains only in the alveolar bone adjacent to the loaded tooth [5]. In numerical analysis of a tooth and its alveolar bone structure (cancellous bone and cortical bone), the periodontal membrane is often ignored due to its complicated shape to reduces the difficulties of analysis and modeling [5]. This research has shown that eliminating PDL may significantly affect simulation results. With the concern of dentists involving the distalize premolar. In this study, the first premolar was considered to modeling to predict results. To determine whether PDL should be incorporated in finite element models, it is essential to know how the effects of the structure supported with and without inclusion of the PDL.

This study aimed to determine the effect of PDL on stress distribution and displacement of a tooth and the underlying alveolar bone structure by investigating the effects of PDL as solid material layer. The dental structure included tooth and alveolar bones was considered in two cases studies, with and without PDL. Due to the simulation are now more powerful and efficiently to manage more complicated problems with calculation in a shorter time with a high accuracy [6, 7]. Therefore, the real tooth are modeled, the numerical simulation has developed from dental geometry using Computed Tomography (CT) scans, recorded as Stereo Lithography (STL) files, and exported as a 3-dimensional (3D) model that can be evaluated by using finite element analysis [8]. The real force of orthodontic system was considered to use in the simulation. Stress distribution over the structure and corresponding deformation were analyzed and discussed.

2. Material and Methods

2.1. Analytical Model of Tooth, PDL and Alveolar Bone (Simple Model)

Before determining the effect of the PDL layer on the structure of the tooth and alveolar bone, a simple model was created. Simulation results of the model were compared and validated with theoretical analysis results. The analytical results are based on a one-dimensional rod system. As in Fig. 1, a rod is subjected to either tensile or compressive loadings. When the rod is under tension, it will extend with a specific length, as follows:

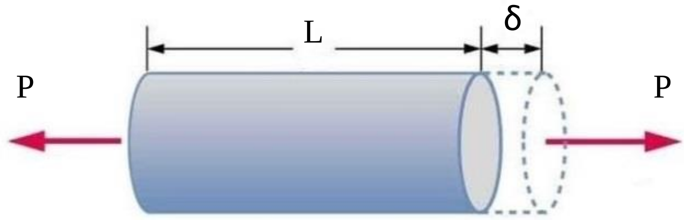
$$\delta = \frac{PL}{AE} \tag{1}$$

where P is the applied load, L is the length of rod, A is cross-sectional area, and E is Young's modulus of material. For compressive loading, the rod will shorten with the same specific length, δ.

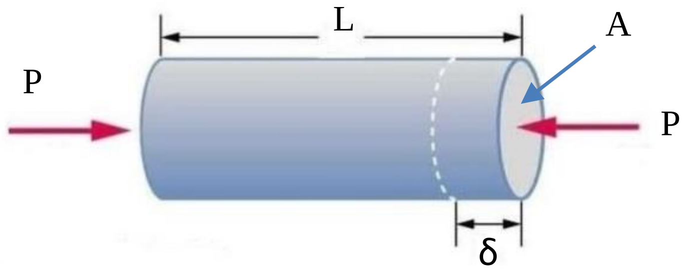
For a rod that consists of several portions of cross sections and materials, the net deformation (δ_{net}) can be calculated as:

$$\delta_{net} = \sum_i [\frac{P_i L_i}{A_i E_i}] \tag{2}$$

where i is the number of portions. The load must be axial, the bars must have a uniform cross-sectional area, and the stress must not exceed the proportional limit [9].



(a)



(b)

Fig. 1. The behaviour of the rod under tensile (a) and compressive (b) forces.

The simple model used to validate the finite element analysis results was a composite rod in axial direction, as shown in Fig. 2. Three different materials of rod diameter 10 mm, with identical cross-sectional areas of $\pi r^2 = 78.54$ mm^2, were used to represent the tooth, PDL, and bone. The tooth and bone portions were each 10 mm thick, with a 0.25 mm PDL sheet layered in between. The interfaces between the bone, PDL, and tooth layers were defined as fully bonded surfaces. The end side of the tooth was loaded of 1 N, and the other side of the bone was fixed. Points 1, 2, and 3 are the selected points of each portion: bone, PDL, and tooth portion, respectively. The simulation displacements of point 1, 2 and 3 were compared with results from the analytical calculation.

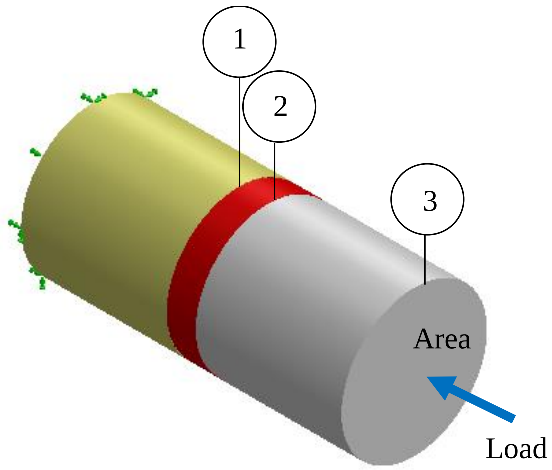


Fig. 2. The bone bar, PDL layer, and tooth bar of the simple model.

2.2. Finite Element Model of the Practical Dental Structure, Tooth, and Alveolar Bone

The actual dental structure model consists of three components: tooth (First-Premolar), PDL, and alveolar bone (Fig. 3). The alveolar bone is composed of cortical and cancellous bones. As the tooth geometry is complicated, it was modelled using micro-computed tomography (or CT scan). The method has generated point cloud geometry into STL format. Then it was revamped and transformed to IGES file. To create the surface shape and solid model of the tooth, the IGES file was imported into SolidWorks software. The finite element model of the PDL and alveolar bone was created directly using the same software [10, 11, 12].

DOI:10.4186/ej.2015.19.2.99

The PDL membrane was modelled as a 0.25 mm thin layer around the root [10, 13, 14]. The alveolar bone was composed of cancellous block covered on the top by a 2 mm thick cortical layer [13, 15], as shown in Fig. 3. The interfaces between all components were treated as perfectly bonded interfaces [4, 8].

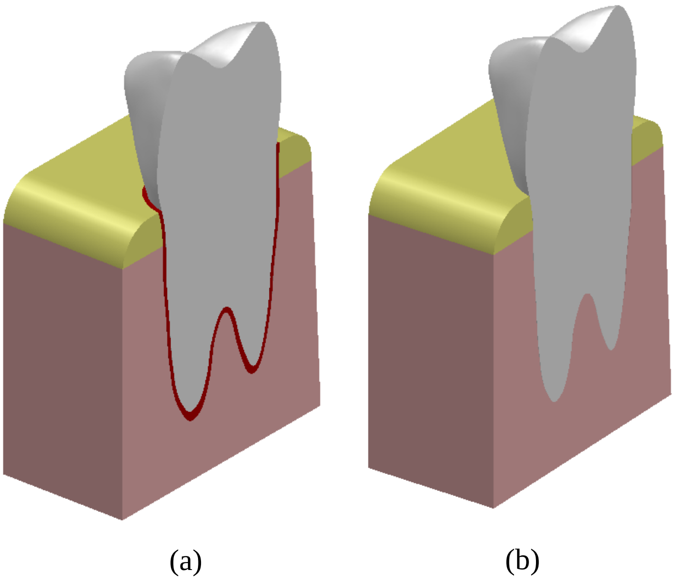


Fig. 3. The cross-section model of dental structure with PDL (a) and without PDL (b).

The finite element model simulates an orthodontic system by using simple loading derived from an elastic chain. The orthodontic force applied to the elastic chain was distributed by a bracket, which was bonded onto the crown of the first-premolar tooth with bonding cement (as exposed surfaces) [13, 16]. A force of 2 N was applied to the elastic chain through the bracket in distal direction (in x direction) [17]. Fig. 4(b) illustrates the simulated load in this study.

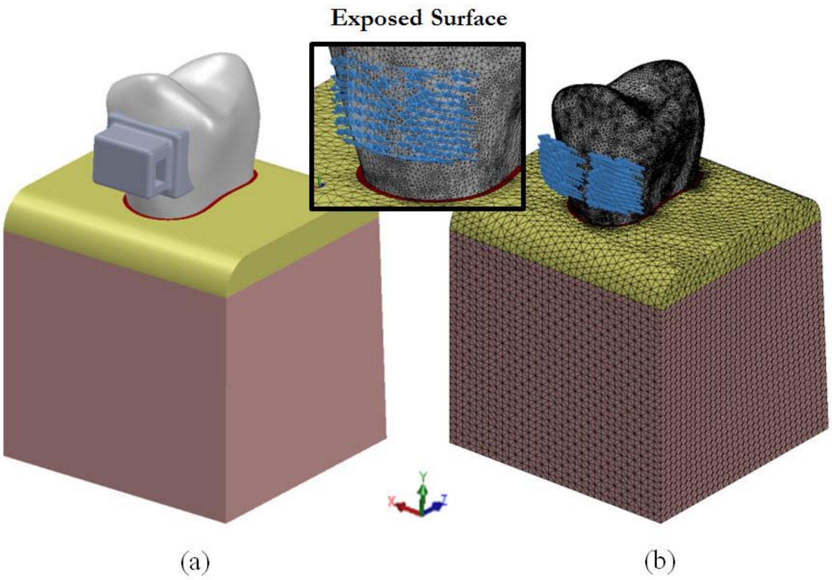


Fig. 4. The structural model (a) and the finite element model, subjected to orthodontic forces (b).

FEA is an approximation of numerical method. The solution can meliorate with using more elements. The result will converge to more exact solution. However, the using numerous elements is not always the corrected solution. Because of a large number of elements affects the calculation of the computer, increasing time in process and need to high effective computer.

The validation of suitable number of element was performed by using dental structure model without PDL (Fig. 3b), to find the range of the number of element that the simulated result approaches one constant value. The relation of the number of element and the maximum displacement of dental model simulation as shown in Fig. 5, it was found that between 800,000-2,500,000 element still provide similar results. With the reason aspect time and computer performance, the suitable range of number of element can be operated since 800,000 elements.

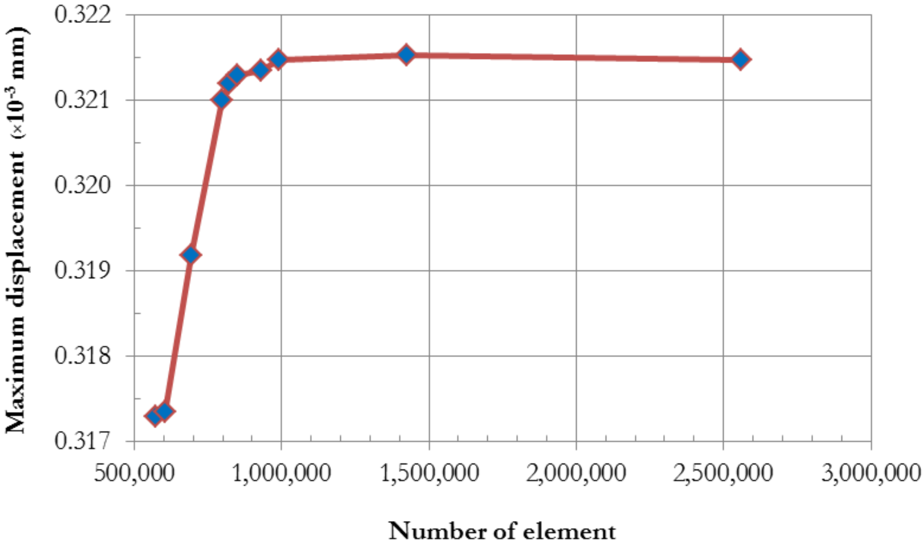


Fig. 5. The relation of number of element and the simulation of maximum displacement using FEA.

Tetrahedral element with four nodes was used as finite element mesh of all parts. The structure with PDL consisted of 2,059,684 nodes and 1,408,862 elements. The structure without PDL consisted of 1,145,558 nodes and 836,197 elements. The exterior nodes of cortical (except the top surface) and cancellous bones were fixed in all directions, since they are part of the jawbone [18, 19].

Table 1 Lists the mechanical properties of cancellous bone, cortical bone, PDL, and tooth. In this preliminary study, all materials are assumed homogeneous, isotropic, and linear elastic [2, 10]. Although PDL is a nonlinear viscoelastic material, the mentioned properties were assigned as the load response lies within the linear elastic range [3, 4, 5, 8, 16].

Table 1. Mechanical properties of materials used in the models [15].

Material	Young's modulus (MPa)	Poisson's ratio
Cancellous	1,370	0.30
Cortical Bone	13,700	0.26
PDL	0.6668	0.49
Tooth	19,613.3	0.15

3. Results and Discussion

3.1. Simple Model

The simple model was used to investigate whether finite element simulation can explain the reliability of the biomechanical results. Table 2 compares the analytical and simulation displacements of the simple model, as described above (see Fig. 2). Points 1, 2, and 3 represent the end of bone, PDL, and tooth portion, respectively. The compared results show that finite element analysis can predict the material responses of tooth and bone structure, with the error less than 0.2%.

Table 2. The displacement comparison of analytical calculation and finite element simulation (Simple model).

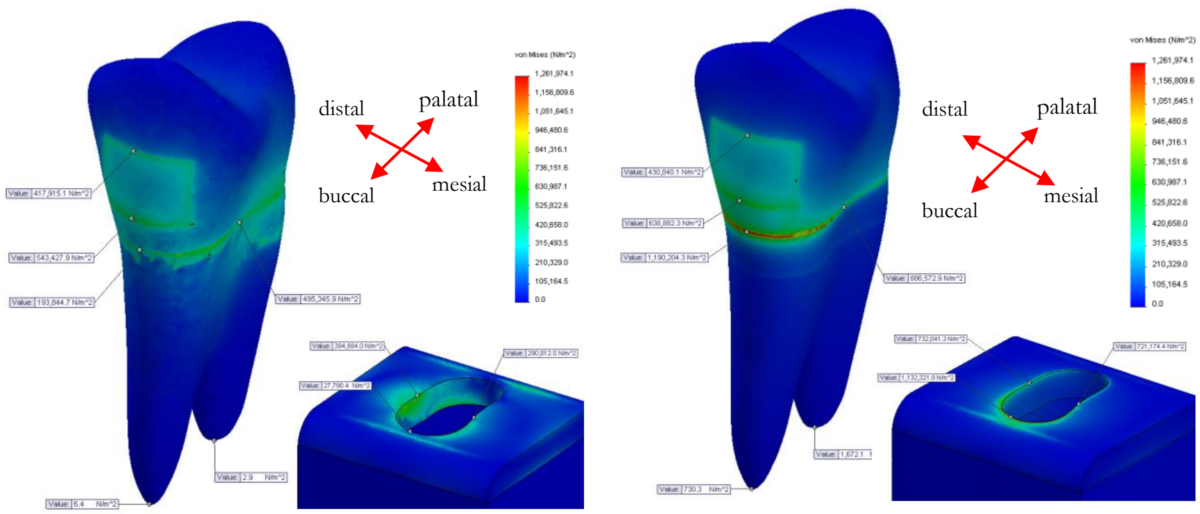
Point	Displacement		
	Analytical	Numerical, FEA	%Error
1	9.29843×10^{-6}	9.28487×10^{-6}	0.146%
2	4.78541×10^{-3}	4.77724×10^{-3}	0.170%
3	4.79191×10^{-3}	4.78371×10^{-3}	0.171%

3.2. Finite Element Model of Practical Dental Structure in Orthodontics

Stress distributions over the first premolar tooth and alveolar bone, with and without PDL, are shown in Fig. 6a and Fig. 6b, respectively.

As shown in the figure, the stress distribution on the crown (surface subjected by force) of the first premolar was the same for both cases. The von Mises stresses of with and without PDL were 543 kPa and 639 kPa, respectively. Without PDL, the highest stress concentration of 1,190 kPa was observed around the cervical portion of the tooth. While with PDL has stress concentration of 194 kPa in the same portion, measured from the buccal side. On the other hand, the case with PDL displayed a more widely distributed area of stress on the cervical region (mesial and distal side) than the case without PDL.

For the alveolar bone – a combination of the cortical and cancellous bone – the stress distributions were shown separately for better visualization. For the cortical bone, the highest stresses were concentrated around the top edge of cervical area. With PDL, the stresses measured were 291 kPa, 28 kPa and 395 kPa. Without PDL, the stresses measured were 721 kPa, 1,132 kPa and 732 kPa, measured from the mesial buccal and distal side, respectively. Clearly, the case with PDL has decreased stress on dental structure with supporting by the soft pad, PDL.



(a) (b)

Fig. 6. von Mises stress contours on the tooth and alveolar bone with PDL (a) and without PDL (b) [7].

To view the inside stress behavior, Fig. 7 illustrates the cross section of the dental structure with PDL. The PDL absorbed most of the stress, with the little that remained passed onto the alveolar bone.

These results indicate that the structural responses differ because the absorption of PDL significantly altered the stress distributions. The PDL was as a cushion pad decreasing the stress concentration occurred to the alveolar bone.

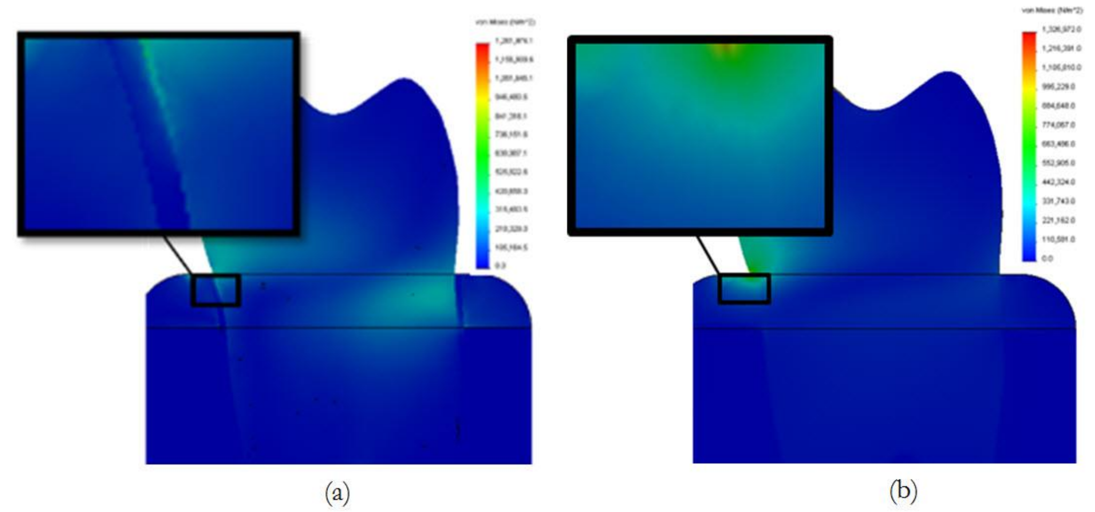


Fig. 7. The von Mises stress contours in the cross section of dental structure with PDL (a), without PDL (b).

Table 3. The significant stresses comparison of model with PDL and without PDL.

The von Mises stress value of first premolar and alveolar bone, case with PDL and without PDL		
Portion	**PDL**	**without PDL**
	First premolar	
Crown	543 kPa	639 kPa
Cervical portion	194 kPa	1,190 kPa
	Alveolar bone (Cortical bone)	
Measured point 1	291 kPa	721 kPa
Measured point 2	28 kPa	1,132 kPa
Measured point 3	395 kPa	732 kPa
* measured by clockwise counting started from right side (as Fig. 6)		

The displacement contour of first premolar was illustrated in Fig. 8, a red tone indicates the maximum displacement and a blue tone indicates the minimum displacement that approaches zero. With PDL, the maximum displacement was located in the root apex of the first premolar and the minimum displacement was at Center of Rotation, CR point (Fig. 8a). This movement behavior as Tipping that is defined as the rotation about the buccal-palatal axis [1]. Without PDL, the maximum displacement was located in the crown component of the first premolar and the minimum displacement was the area along root region (Fig. 8b). It just has movement at the crown only.

The comparative views of the displacement in the same direction of orthodontic force (X - axis) in both cases are shown in Fig. 9. A blue tone indicates displacement in the same direction as the force (positive displacement) and a red tone indicates displacement in the converse direction of the force (negative displacement). The maximum positive and negative displacement of the first premolar with PDL was 0.2860 mm and 0.3182 mm, respectively (Fig. 9a). The maximum positive and negative displacement of the first premolar without PDL was 3.210×10^{-4} mm and 1.230×10^{-5} mm, respectively (Fig. 9b).

Explicitly, the displacements in the first premolar with PDL exceeded those of the first premolar without PDL, reflecting the effect of PDL for tooth movement in the structure. The mechanical properties of PDL that were well flexibility caused the tooth can move independently within its soft volume supported.

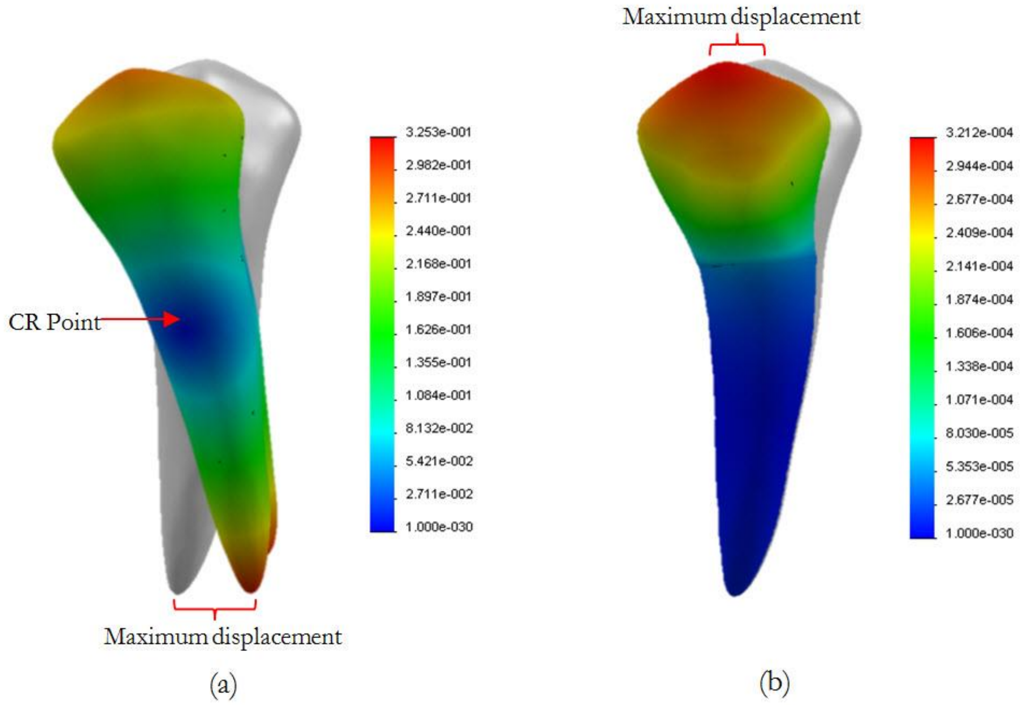


Fig. 8. The maximum and minimum displacement contour of first premolar, with PDL (a) and without PDL (b). The displacement shows the magnified tooth displacement over a scale of 10 times and 5,000 times, respectively [20].

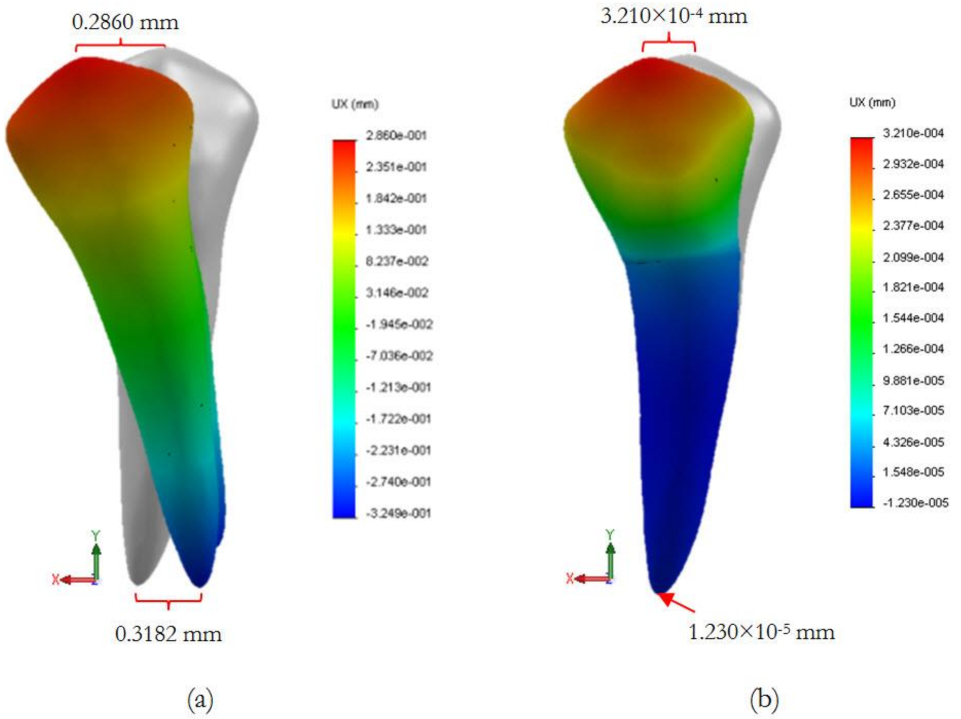


Fig. 9. The first premolar contour showing the displacement in the similar direction of orthodontics force (X - axis), with PDL (a) and without PDL (b). The X – axis displacement shows the magnified tooth displacement over a scale of 10 times and 5,000 times, respectively [20].

Table 4. The maximum displacement comparison of model with PDL and without PDL.

The maximum displacement comparison of first premolar, case with PDL and without PDL		
Direction	**PDL**	**without PDL**
Positive displacement	0.2860 mm	3.210×10^{-4} mm
Negative displacement	0.3182 mm	1.230×10^{-5} mm
* positive displacement indicates in the same direction of orthodontic force		

4. Conclusion

Finite element analysis was used to simulate the structural model with PDL and without PDL, it was assumed that the tooth was subjected force by orthodontic hardware, in order to investigate the effect of PDL on the tooth and bone structure. The conclusions are as follows:

- PDL significantly influences the stress distribution within a tooth's supporting structure. It absorbed much of the stress in the transition of forces from tooth to alveolar bone.
- PDL is important for tooth displacement. The tooth can more independent for the movement within the soft volume surrounded.

To analyze some orthodontic like this, PDL should not be ignored in modeling. It is more important to stress distribution and displacement of any tooth that it covered.

Finite element analysis shows promise as a convenient and efficient tool for addressing the aforementioned biomechanical problems. However, these results should be verified experimentally to make the method more reliability.

Acknowledgement

This research was supported by graduate research grants funded by The Graduate School Chiang Mai University.

References

[1] P. D. Jeon, P. K. Turley, H. B. Moon, and K. Ting, "Analysis of stress in the periodontium of the maxillary first molar with a three-dimensional finite element model," *American Journal of Orthodontics and Dentofacial Orthopedics*, vol. 115, pp. 267–274, 1999.

[2] M. Motoyoshi, S. Ueno, K. Okazaki, and N. Shimizu, "Bone stress for a mini-implant close to the roots of adjacent teeth—3D finite element analysis," *International Journal of Oral and Maxillofacial Surgery*, vol. 38, no. 4, pp. 363–368, 2009.

[3] J. H. Jang, S. J. Park, K. S. Min, B. N. Lee, H. S. Chang, W. M. Oh, H. Lim, Y. T. Cho, J. T. Koh, H. H. Son, Y. C. Hwang, and I. N. Hwang, "Stress behavior of cemented fiber-reinforced composite and titanium posts in the upper central incisor according to the post length: Two-dimensional finite element analysis," *Journal of Dental Sciences*, vol. 7, no. 4, pp. 384–389, 2012.

[4] A. Merdji, R. Mootanah, B. B. Bouiadjra, A. Benaissa, L. Aminallah, E. O. Chikh, and S. Mukdadi, "Stress analysis in single molar tooth," *Materials Science and Engineering: C*, vol. 33, no. 2, pp. 691–698, 2013.

[5] F. Groning, M. J. Fagan, and P. O'Higgins, "The effects of the periodontal ligament on mandibular stiffness: a study combining finite element analysis and geometric morphometrics," *Journal of Biomechanics*, vol. 44, no. 7, pp. 1304–1312, 2011.

[6] A. Hohmann, U. Wolfram, M. Geiger, A. Boryor, C.Kober, C. Sander, and F. G. Sander, "Correspondences of hydrostatic pressure in periodontal ligament with regions of root resorption: A clinical and a finite element study of the same human teeth," *Computer Methods and Programs in Biomedicine*, vol. 93, pp. 155–161, 2009.

[7] T. P. Bezerra, F. I. Silva Junior, H. C. Scarparo, F. W. G. Costa, and E. C. Studart-Soares, "Do erupted third molars weaken the mandibular angle after trauma to the chin region? A 3D finite element study," *International Journal of Oral and Maxillofacial Surgery*, vol. 42, no. 4, pp. 474–480, 2012.

[8] E. R. Miyashita, B. C. Mattos, P. N. Noritomi, and H. Navarro, "Finite element analysis of maxillary bone stress caused by Aramany Class IV obturator prostheses," *The Journal of Prosthetic Dentistry*, vol. 107, no. 5, pp. 336–342, 2012.
[9] A. Pytel and F. L. Singer, "Simple stress," in *Strength of materials*, 4th ed. Harpercollins College Div, 1987.
[10] A. Geramy, M. Adibrad, and M. Sahabi, "The effects of splinting periodontally compromised removable partial denture abutments on bone stresses: a three-dimensional finite element study," *Journal of Dental Sciences*, vol. 5, no. 1, pp. 1–7, 2010.
[11] T. Kondo and N. Wakabayashi, "Influence of molar support loss on stress and strain in premolar periodontium: A patient-specific FEM study," *Journal of Dentistry*, vol. 37, pp. 541–548, 2009.
[12] Q. Li, "An expert system for stress analysis of human teeth dissertation," Ph.D. thesis, Mechanical Engineering, The Graduate School of Vanderbilt University, Nashville, Tennessee, 2009.
[13] M. I. F. Jasmine, A. A. Yezdani, F. Tajir, and R.M. Venud, "Analysis of stress in bone and micro implants during en-masse retraction of maxillary and mandibular anterior teeth with different insertion angulations: A 3-dimensional finite element analysis study," *American Journal of Orthodontics and Dentofacial Orthopedics*, vol. 141, no. 1, pp. 71–80, 2012.
[14] J. Lindhe and T. Karring, "The anatomy of the periodontium," in *Textbook of Clinical Perodontology*, 2nd ed. Copenhagen: Munksgaard, 1989, pp. 19–69.
[15] M. Cifter and M. Sarac, "Maxillary posterior intrusion mechanics with mini-implant anchorage evaluated with the finite element method," *American Journal of Orthodontics and Dentofacial Orthopedics*, vol. 140, no. 5, pp. e233–e241, 2011.
[16] C. Field, I. Ichim, M. V. Swain, E. Chan, M. A. Darendeliler, W. Li, and Q. Lig, "Mechanical responses to orthodontic loading: A 3-dimensional finite element multi-tooth model," *American Journal of Orthodontics and Dentofacial Orthopedics*, vol. 135, no. 2, pp. 174–181, 2009.
[17] N. Yoshida, Y. Koga, C. Peng, E. Tanaka, and K. Kobayashi, "In vivo measurement of the elastic modulus of the human periodontal ligament," *Medical Engineering & Physics*, vol. 23, no. 8, pp. 567–572, 2001.
[18] L. M. Ren, W. X. Wang, Y. Takao, and Z. X. Chen, "Effects of cementum–dentine junction and cementum on the mechanical response of tooth supporting structure," *Journal of Dentistry*, vol. 38, no. 11, pp. 882–891, 2010.
[19] H. H. Ammar, P. Ngan, R. J. Crout, V. H. Mucino, and O. M. Mukdadid, "Three-dimensional modeling and finite element analysis in treatment planning for orthodontic tooth movement," *American Journal of Orthodontics and Dentofacial Orthopedics*, vol. 139, no. 1, pp. e59–e71, 2011.
[20] I. J. Yu, Y. A. Kook, S. J. Sung, K. J. Lee, Y. S. Chun, and S. S. Mo, "Comparison of tooth displacement between buccal mini-implants and palatal plate anchorage for molar distalization: a finite element study," *The European Journal of Orthodontics*, vol. 36, no. 4, pp. 394–402, 2011.