Stress Analysis of Different Angulations of Implant Installation: The Finite Element Method

Ting-Hsun Lan,[1] Heng-Li Huang,[2] Ju-Hui Wu,[1] Huey-Er Lee,[1,3] and Chau-Hsiang Wang[1,3]
[1]Department of Prosthodontics, Kaohsiung Medical University Hospital, [3]Faculty of Dentistry, College of Dental Medicine, Kaohsiung Medical University, Kaohsiung, and [2]School of Dentistry, China Medical University, Taichung, Taiwan.

Clinically, many implant cases with different angulation over the lower posterior area have been found. The purpose of this study was to analyze the bony stress with different implant tilting during normal masticatory load using the finite element method (FEM), with the hope of discovering a desirable installation of implant. A three-dimensional finite element method was employed to analyze the bony stress generated by different angulation designs (15°) of implant bodies. Eight solid models of the mandibular first and second molars were built up and then transferred to a mesh model in FEM (ANSYS) to perform a stress analysis. A simulated load (400 N) was applied to the splinted crowns with vertical and horizontal forces. The loading sites were on the central fossa of the splinted crowns. For stress distribution, some designs will be better than a parallel installation. The results suggested that not all implant bodies tilting with the splinted crowns lead to stress concentration.

Key Words: angulated, finite element method, implant body, stress analysis
(*Kaohsiung J Med Sci* 2008;24:138–43)

Dental implantation serves as a foundation for the support of a fixed or removable prosthesis. In recent years, many systems have been introduced to dentists by various manufacturers. With the progress of implant surface management, high survival rates with easy installation have persuaded most dentists to participate in this work. Although surgical stent and radiography assist the implant insertion procedure, angulated implant bodies have still appeared in recent studies [1] and clinical situations.

Finite element analysis has been utilized to evaluate the stress induced around the implant and the surrounding bone tissue, including maxilla posterior edentulism [1], mandibular posterior edentulism [2–5], restoration type [6] and implant body arrangement [7]. This study focused on the compressive stress analysis of different implant angulations when vertical and different directions of horizontal forces were applied.

Methods

According to morphologic data on the Asian mandible [8], including bone size, cortical bone thickness and tooth size, a bone block from the canine to the posterior border of the mandible was built up with a coordinate system. We measured the key point, line, area and volume.

Cortical bone with different thicknesses [8] (Table 1) was defined around the cancellous bone. It was assumed to be isotropic, homogeneous and linearly elastic. The Young's modulus of cortical and cancellous bone was assumed to be 13,700 MPa and

Received: Apr 23, 2007 Accepted: Sep 26, 2007
Address correspondence and reprint requests to: Dr Chau-Hsiang Wang, Department of Prosthodontics, Kaohsiung Medical University Hospital, 100 Tzyou 1st Road, Kaohsiung 807, Taiwan.
E-mail: chhswa@kmu.edu.tw

Table 1. Mandibular buccal and lingual mean cortical thickness over cervical area (mm)

	Canine	1st premolar	2nd premolar	1st molar	2nd molar	3rd molar
Buccal	0.60	0.61	1.05	1.36	2.18	2.78
Lingual	1.54	1.82	2.23	2.39	2.71	2.09

Table 2. Material properties

Material	Young's modulus E (MPa)	Poisson's ratio
Cortical bone	13,700	0.30
Cancellous bone	1,850	0.30
Ti-6Al-4V	117,000	0.35
Ceramic crown	69,000	0.28

1,850 MPa, respectively. Poisson's ratio was 0.3 for both [7].

Implants of 11 mm in length and 5 mm in diameter were installed into the center of the residual ridge according to the coordinate system and an ideal occlusal plane. The distance from the center of each implant to the center of the other was 10 mm.

Titanium-aluminum-vanadium (Ti-6Al-4V) was used as the implant material, and Young's modulus and Poisson ratio were assumed to be 117,000 MPa and 0.35, respectively [4]. Splinted full ceramic crowns were selected for the prosthesis, with Young's modulus and Poisson ratio assumed to be 69,000 MPa and 0.28, respectively [9] (Table 2).

After material properties were applied, a mesh 3-D finite element model was constructed with a tetrahedral 10-node 92 element, resulting in 123,172 elements and 183,948 nodes in the ANSYS system (5.6; ANSYS, Canonsburg, PA, USA).

Models were constrained in all directions at the nodes on the mesial and distal border of the bone surface. Applied loads can be resolved into minimum principle stress to evaluate the bony response. A 200 N load was applied to the central fossa of each prosthetic construction in vertical and horizontal directions. Implants that were perpendicular to the occlusal plane and parallel to each other were called Model 1. We used model 1 as the standard model to analyze the stress-concentrated areas. Minimum principle stress (σ min) was used to express the peak compressive stress values of alveolar bone.

After a preliminary study, the implant was tilted in the same occlusal plane by 0 or 15 degrees mesially or distally (Figure 1). In addition, three forces were applied to these eight solid models, including forces in vertical and horizontal directions from buccal to lingual, and in the horizontal direction from lingual to buccal (Table 3).

Results

Vertical load

The peak compressive stress values were predominantly found in cortical bone around the cervical region of the implants. The value that appeared in the standard model was 17.43 MPa located in the second implant alveolar bone; models 2, 4 and 8 all showed a similar situation. The highest value, which appeared in model 7, was 27.14 MPa, and the lowest value, which appeared in model 6, was 13.66 MPa, located at the first implant alveolar bone. Models 3 and 5 showed a similar situation. We set the eight models in the same bony stress value from 0 to –13 and compared the distributions (Figure 2).

Horizontal load (buccal to lingual)

The peak compressive stress values were also predominantly found in cortical bone around the cervical region of the implants. The value appearing in the standard model was 86.38 MPa at the second implant alveolar bone. The highest value appeared in model 8, and was 93.99 MPa. The lowest value appeared in model 6, and was 77.11 MPa. We set the eight models in the same bony stress value from 15 to –75 and compared the distributions (Figure 3).

Horizontal load (lingual to buccal)

The results were similar to those obtained when a horizontal load was applied from buccal to lingual. The value appearing in the standard model was 73.72 MPa located at the second implant alveolar bone. The highest value appeared in model 8, and was 77.18 MPa. The lowest value appeared in model 5, and was 69.77 MPa. We set the eight models in the same bony stress value from 15 to –60 and compared the distributions (Figure 4).

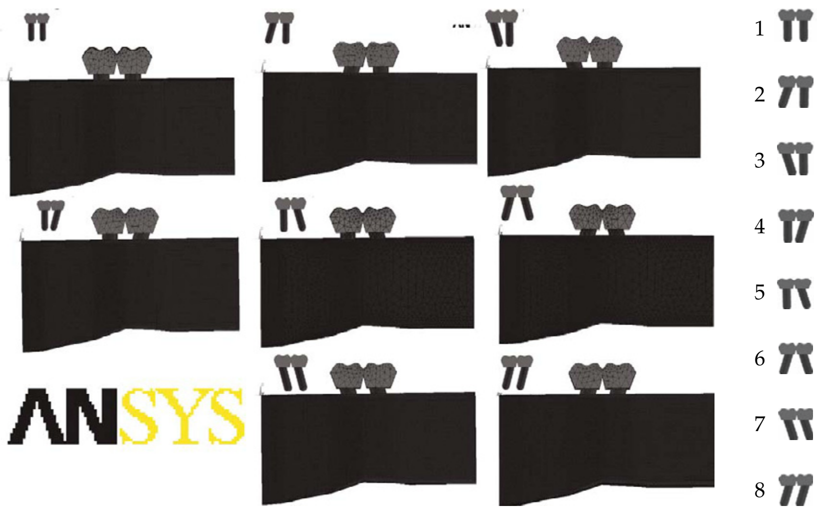

Figure 1. *Eight different mesiodistally-angulated models according to the same prosthetic design. Left implant replaces first molar, and right one replaces second molar. From left to right: top row are models 1, 2, 3; middle row are models 4, 5, 6; and bottom row are models 7, 8.*

Table 3. Peak compressive stresses on cortical bone around implants

Model	Vertical	Horizontal	
		B → L	L → B
1	17.43	86.38	73.72
2	18.01	86.05	73.81
3	26.31	87.06	73.95
4	25.68	93.83	76.92
5	18.23	77.28	69.77
6	13.66	77.11	70.09
7	27.14	77.59	70.08
8	24.98	93.99	77.18

B = buccal; L = lingual.

Figure 5 shows the compressive stresses on cortical bone around the implants, for all three direction loadings.

Discussion

The finite element method [10,11] has advantages for stress analysis, but the thickness of cortical bone is hard to differentiate when using computed tomography [3,12] or surface scanners [4]. Thus, we used a coordinate system to build up a solid model. From the key point, line, and area, volume to model, we obtained different cortical bone dimensions [8] to perform this study.

Identifying the term of overload creates some difficulty, but we could predict the worse tendency for implant failure. For posterior short-span prostheses, by raising the potential negative effects of bending with offset loads as well as lateral force, bending that elevates stresses to the implant and bone should be minimized [13]. In our study, we found the mean peak stress from a horizontal direction was four to six times that in the vertical direction. As theoretical calculation of load distribution on partial prostheses has demonstrated load increases from cantilevers, offset implants and cusp inclination [7], from the angulated installation of implant bodies, we found the lowest peak stress in model 6, but not model 1. In addition, the distal inclination of implant bodies, especially in the second molar area, revealed the worst results.

Aspects to consider for sufficient bone support are primary cortical anchorage and healing time before loading the implants. Reliance on the existing cortical bone seems to be the most predictable condition. This means great care must be taken to preserve any available outer cortex in the posterior regions for anchorage of implant threads. The different thicknesses of cortical bone between the buccal and lingual sides may affect the prognosis of implant. In the same force but in the opposing direction, the thicker the cortical bone, the more the peak value of stress. On the basis of this finding, we suggest that clinicians carefully note the balance side interference from occlusion. Because occlusion may change over time, it is important during follow-up to check these parameters in situations where other load factors are present [7].

Implant treatment is multifaceted: a number of load factors of smaller magnitude may be acceptable. The ability to actively control the occlusal condition

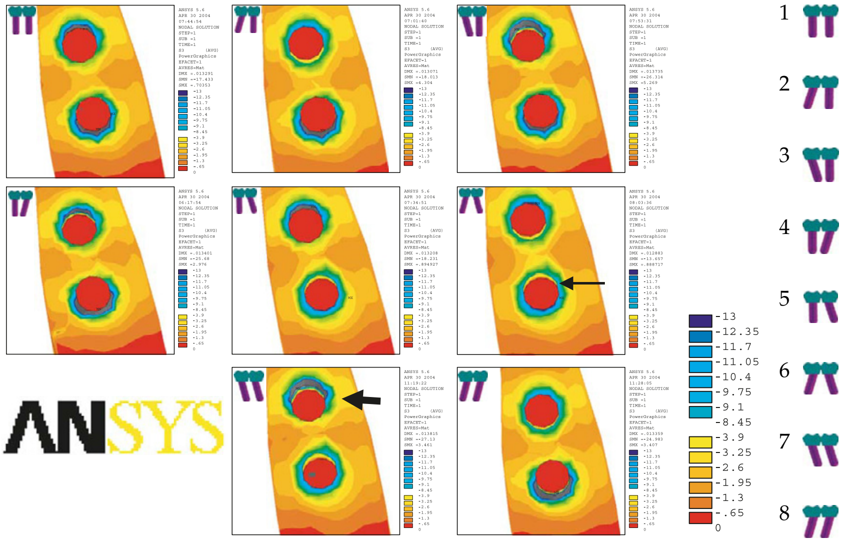

Figure 2. *Peak of maximum compressive stress in eight solid models. The models are arranged as in Figure 1. Model 7 has the highest value (thick arrow) and model 6 has the lowest value (thin arrow).*

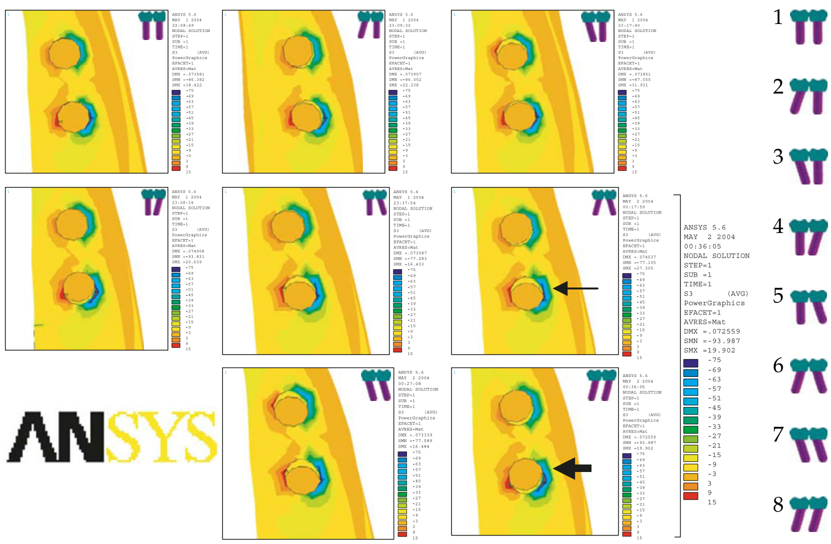

Figure 3. *Horizontal load (buccal to lingual). The models are arranged as in Figure 1. Model 8 has the highest value (thick arrow) and model 6 has the lowest value (thin arrow).*

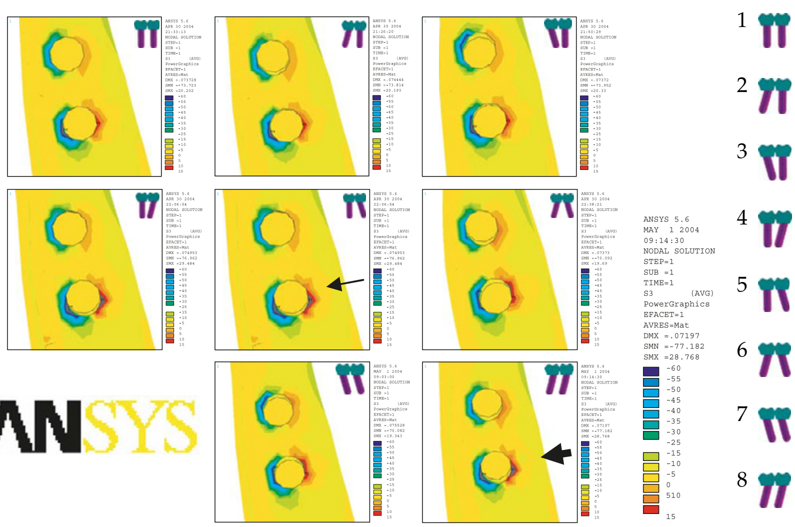

Figure 4. *Horizontal load (lingual to buccal). The models are arranged as in Figure 1. Model 8 has the highest value (thick arrow) and model 5 has the lowest value (thin arrow).*

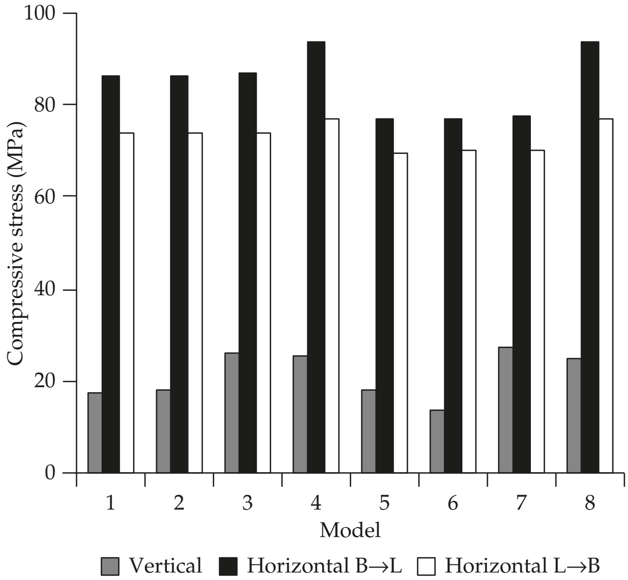

Figure 5. *Compressive stresses on cortical bone around implants from three direction loadings. B = buccal; L = lingual.*

in situations with multiple geometric load factors seems to be especially significant. It is important to react to mechanical problems, as well as excessive bone resorption, with an appropriate response when they occur, for example, by eliminating cantilevers [14], narrowing the buccolingual width or mesiodistal length of the teeth, flattening cuspal inclination, and centering the occlusal contact. In our study, we found better bony responses to angulation of implant bodies (model 6), which might prevent future overload and bending overload.

Implant overload in posterior partial restorations should be prevented by screening patients for load factors. It is possible to identify potential overload situations in advance of treatment.

This result suggests that not all implant bodies tilting with the splinted crowns lead to stress concentration. In changeable clinical situations, preventing worsening angulation (models 4, 8) and achieving better angulation (models 6, 5, 1) is our goal.

Acknowledgments

This work was supported by a grant from the National Science Council of Taiwan (NSC 92-2320-B-037-032).

References

1. Caglar A, Aydin C, Ozen J, et al. Effects of mesiodistal inclination of implants on stress distribution in implant-supported fixed prostheses. *Int J Oral Maxillofac Implants* 2006;21:36–44.
2. Tuncelli B, Poyrazoglu E, Koyluoglu AM, et al. Comparison of load transfer by implant abutments of various diameters. *Eur J Prosthodont Restor Dent* 1997;5:79–83.
3. Akca K, Iplikcioglu H. Evaluation of the effect of the residual bone angulation on implant-supported fixed prostheses in mandibular posterior edentulism. Part I: spiral computed tomography study. *Implant Dent* 2001; 10:216–22.
4. Akca K, Iplikcioglu H. Evaluation of the effect of the residual bone angulation on implant-supported fixed prosthesis in mandibular posterior edentulism. Part II: 3-D finite element stress analysis. *Implant Dent* 2001; 10:238–45.
5. van Zyl PP, Grundling NL, Jooste CH, et al. Three-dimensional finite element model of a human mandible incorporating six osseointegrated implants for stress analysis of mandibular cantilever prostheses. *Int J Oral Maxillofac Implants* 1995;10:51–7.
6. Stegaroiu R, Sato T, Kusakari H, et al. Influence of restoration type on stress distribution in bone around implants: a three-dimensional finite element analysis. *Int J Oral Maxillofac Implants* 1998;13:82–90.
7. Rangert BR, Sullivan RM, Jemt TM. Load factor control for implants in the posterior partially edentulous segment. *Int J Oral Maxillofac Implants* 1997;12:360–70.
8. Yasuhiko K. *Clinical and Applied Craniology*. Tokyo: Anatomy Publishing, 1966.
9. Kamposiora P, Papavasilious G, Bayne SC, et al. Finite element analysis estimates of cement microfracture under complete veneer crowns. *J Prosthet Dent* 1994; 71:435–41.
10. DeTolla DH, Andreana S, Patra A, et al. Role of the finite element model in dental implants. *J Oral Implantol* 2000;26:77–81.
11. Baiamonte T, Abbate MF, Pizzarello F, et al. The experimental verification of the efficacy of finite element modeling to dental implant systems. *J Oral Implantol* 1996;22:104–10.
12. Van Oossterwyck H, Duyck J, Vander Sloten J, et al. Use of microfocus computerized tomography as a new technique for characterizing bone tissue around oral implants. *J Oral Implantol* 2000;26:5–12.
13. Rangert B, Jemt T, Jorneus L. Forces and moments on Branemark implants. *Int J Oral Maxillofac Implants* 1989;4:241–7.
14. Krekmanov L, Kahn M, Rangert B, et al. Tilting of posterior mandibular and maxillary implants for improved prosthesis support. *Int J Oral Maxillofac Implants* 2000; 15:405–14.

不同角度植體植入的應力分析：有限元素分析

藍鼎勛[1] 黃恆立[2] 吳如惠[1] 李惠娥[1,3] 王兆祥[1,3]

[1]高雄醫學大學附設醫院　牙科部補綴科

[2]中國醫藥大學 牙醫學系

[3]高雄醫學大學　口腔醫學院　牙醫學系

臨床上，在下顎後牙區我們常常發現許多不同角度的植體植入。本研究的目的是利用有限元素法針對正常咀嚼施力下不同角度的植體周圍骨應力變化分析，期望找出植體較為理想的植入角度。本研究的植體角度以 15 度為變化，模擬 8 組下顎第一大臼齒、第二大臼齒的三維模型，在模擬的併聯牙冠贋復物上施力 400 牛頓，方向為垂直力與水平力。研究的結果顯示，有些甚至比平行植入的植體在骨應力上更佳。臨床上，有很多理由可解釋為何特別是後牙區在植體植入時會有角度上的差異，由本研究結果可知，不平行植入體不一定會形成有意義的有害骨應力。

關鍵詞：角度，有限元素法，植體，應力分析

(高雄醫誌 **2008;24:138－43**)

收文日期：**96** 年 **4** 月 **23** 日

接受刊載：**96** 年 **9** 月 **26** 日

通訊作者：王兆祥醫師

高雄醫學大學附設醫院牙科部補綴科

高雄市**807**三民區自由一路**100**號