Effects of Mesiodistal Inclination of Implants on Stress Distribution in Implant-Supported Fixed Prostheses

Alper Çağlar, PhD[1]/Cemal Aydin, PhD[2]/Jülide Özen, PhD[3]/Caner Yilmaz, PhD[2]/Turan Korkmaz, PhD[4]

Purpose: *Patterns of von Mises stress values surrounding implants supporting fixed prostheses in the posterior edentulous maxilla were evaluated using 3-dimensional finite element analysis.* ***Materials and Methods:*** *Implants were placed in maxillary bone in 2 different configurations. In the first configuration, implants were placed in the first premolar, second premolar, and second molar regions; in the second configuration, implants were placed in the second premolar and second molar regions, and a mesial cantilever was extended to the space of the first premolar tooth on the superstructure. On the implant placed in the socket of the second molar, 3 different inclinations were used (0, 15, and 30 degrees). Loading was applied in the vertical, oblique, and horizontal axes.* ***Results:*** *Inclination of the implant in the molar region was found to result in increased stress. Significant increase in stress on the implant embedded in the premolar region was also seen in the design with the cantilever as compared to the conventional prosthesis design.* ***Discussion:*** *The stress concentrations observed at the neck of the implant were similar to results reported in the literature.* ***Conclusion:*** *The highest stress value obtained in the study was 194.2 MPa with oblique loading. This value did not exceed the endurance limit of pure titanium, which is 259.9 MPa.* INT J ORAL MAXILLOFAC IMPLANTS 2006;21:36–44

Key words: *finite element analysis, fixed partial dentures, implant inclination, implant-supported prostheses, stress distribution*

In recent years, dental implants have been used in a variety of situations in dental treatment, from single tooth replacement to restoration of complete arches. Biomechanical principles are one of the important factors in implant-supported prosthesis success. Implant treatment involves both biologic and mechanical components. In this biomechanical system, overloading can involve both marginal bone loss and component failure caused by biting forces.[1–4]

Finite element analysis (FEA) has been utilized to evaluate the stresses induced around implant components and surrounding bone tissue. To study a complex mechanical problem, FEA can be used to divide the problem into a collection of much smaller and simpler elements. Complicated geometric structure is converted into a mesh in a computer set. This structure consists of elements, their respective nodes, and predefined boundary conditions. Nodes are divided into same-sized finite number elements interconnecting each other at angles. Displacement and stress caused by loading on each node can then be calculated by a computer program.[5] Image data obtained with the aid of computed tomography or magnetic resonance imaging is used to get 2- or 3-dimensional information and the mesh necessary for finite element analysis.[5,6]

[1]Research Fellow, Department of Prosthodontics, Dentistry Faculty of Gazi University, Ankara, Turkey.
[2]Associate Professor, Department of Prosthodontics, Dentistry Faculty of Gazi University, Ankara, Turkey.
[3]Assistant Professor, Department of Prosthodontics, Gülhane Military Medical Academy, Ankara, Turkey.
[4]Assistant Professor, Department of Prosthodontics, Dentistry Faculty of Gazi University, Ankara, Turkey.

Correspondence to: Dr Alper Çaglar, Gazi Üniversitesi Diş Hekimliği Fakültesi, 82. Sok. 8. Cad Emek, Ankara, Turkey 06510. Fax: +90 312 223 92 26. E-mail: caglaralper@yahoo.com

FEA can be used to evaluate different materials. For example, the nonhomogeneous structure of bone can be subdivided into cortical and trabecular components. Furthermore, this analysis can yield 3-dimensional (3D) images of the displacement and stresses calculated.

In partially edentulous arches, location of the implants is more linear than in complete-arch restorations and is therefore greatly affected by the bending moments. Because of the anatomic position of the maxillary sinus, placement of implants in the desired number and location cannot be established. For this reason the implants are placed in an inclined position.[3,4,7]

In the present study, the effects of mesiodistal inclination of implants on the stress distribution on an implant-supported fixed prosthesis placed in the posterior maxillary region were investigated.

The aim of this study was to evaluate von Mises (VM) stress values and patterns on implants with mesiodistal inclination in implant-supported fixed prostheses applied in maxillary posterior edentulous cases using 3D FEA. For this purpose, cadaveric maxillary bone was used.

MATERIALS AND METHODS

To obtain cross-sectional images of the maxillary bone, Siemens Somatom Volume Zoom computed tomography (Medical Engineering Group, Forcheim, Germany) was used. The apparatus was set to 120 kV and 7 mA, eff. mAs 80. Cross sections were obtained in the frontal plane and from the posterior region in 1-mm section spaces. Organization of the data to obtain a more homogeneous 3D mesh structure was accomplished with the creation of a solid 3D model and the Mark Mentat FEA program (MSC Software, Santa Ana, CA).

In the FEA program, a mesh structure was divided into sets depending on material and region.

Construction of 3 D Finite Element Model

Implants were placed in the left maxillary bone using 2 different configurations. In the first design, 1 implant was placed in the first premolar site (Pm^1), 1 in the second premolar site (Pm^2), and 1 in the second molar (M^2) site; in the second configuration, 1 implant was placed in the Pm^2 region and 1 in the M^2 region. The implants were 11 mm long and 4.3 mm high. Implants were modeled were solid cylinder-type titanium implants.

A minimum of 1 mm of space was preserved between the implants and maxillary sinus. Abutments were modeled as 6.5 mm in height. An inclination of 5 degrees was used. For the implant placed in the M^2 socket, 3 different inclinations were used: 0, 15, and 30 degrees (Fig 1).

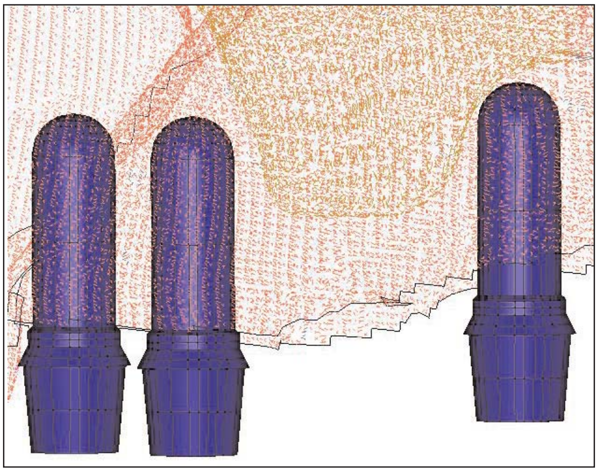

Fig 1 Cylindrical implant design and relationship with the maxillary sinus.

In the study, maxillary bone was considered to be homogeneous type 3 bone. For this reason, a new mesh structure was constituted for the cortical bone of 0.75-mm thickness. Cortical and trabecular bone were assumed to be isotropic, homogeneous, and linearly elastic. The canine set was defined as a separate set and transformed to a solid form as dentin. Modeling of the implant placed into the socket of M^2 at 3 different angles and in 2 different designs required the formation of 6 separate solid models. Other than cortical bone, the rest of the bone structure was transformed to a solid material with trabecular bone properties. A tight bond between bone and the whole implant interface was accepted.

The fixed partial prosthesis was divided into 2 separate sets, metal framework and porcelain. In addition, 2 different designs were prepared. A fixed partial prosthesis with 4 units was constructed to be used with the 3-implant configuration. For the 2-implant configuration, 2 implant supports were used, and a mesial cantilever was extended to the space of Pm^1 tooth. The metal framework was defined as a cobalt-chromium (Co-Cr) alloy and prepared with a minimal thickness of 0.3 mm. For the porcelain set, the mesh structure was detached from the main model. In modeling of the pontic placed in the region where an M^1 tooth was lacking, a mesh structure of the crown of the M^1 tooth derived from cadaver bone was used. The pontic was prepared as a ridge-lap by adding elements and changing the localizations of the nodes.

All of the sets defined in this study can be seen together in Fig 2. The elastic modulus and Poisson ratio are shown in Table 1.

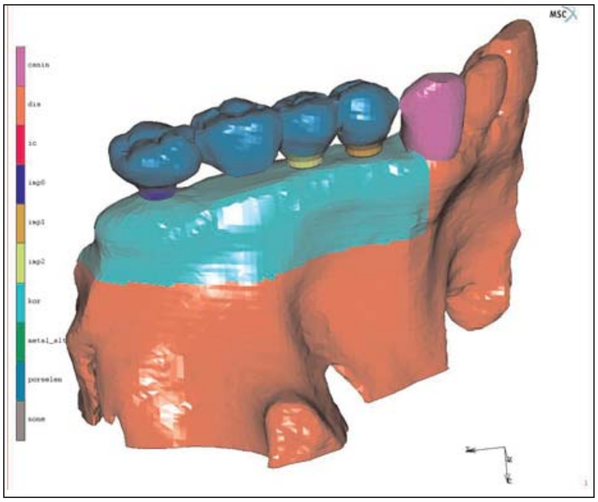

Fig 2 View of all the sets.

Table 1 Elastic Modulus, Poisson's Ratio, and References of the Materials Used

	Elastic modulus (E)	Poisson ratio (Ω)	References
Cortical bone	13.70 GPa	0.30	8, 9, 10
Trabecular bone	1.37 GPa	0.30	8, 10
Dentin	18.60 GPa	0.31	5
Titanium	115.00 GPa	0.35	9
Cr-Co alloy	218.00 GPa	0.33	5
Porcelain	68.90 GPa	0.28	5

Boundary Conditions

The models prepared in the study were fixed in 2 separate planes. The first plane was the medio-axial plane crossing the maxillary bone medially. The second plane was a horizontal plane crossing the maxillary bone at the superior border. These planes acted as support to the model. The model took support from these regions when a force was applied. The supporting planes had to be located far from the areas where stress was to be analyzed to avoid influencing the analysis.[8,9] The planes were fixed at least 20 mm away from the evaluation area.

Loading

Load was applied to each unit of the prosthesis. Loading areas were determined by contact points between the functional cusps of the mandibular teeth and the maxillary teeth in a cusp-marginal ridge relation. Loading was realized at 94 separate nodes (Fig 3).

Loading was applied in 3 different axes, ie, it was either vertical, oblique (at an angle of 30 degrees), or horizontal. Vertical loading was 470 N. Loading was carried out as follows: 100 N to the Pm^1 region, 100 N to the crown at Pm^2 region, 140 N to the pontic, and 130 N to the crown in the M^2 region.[10] Thus, 5-N forces were loaded to each node. Furthermore, a force of 940 N was used for oblique loading; a force of 135 N was applied for horizontal loading. The ratio between the vertical, oblique, and horizontal loads was similar to the ratio reported in the investigation by Koolstra and colleagues.[11] This ratio has also been used by other authors in various studies on this subject.[12–14]

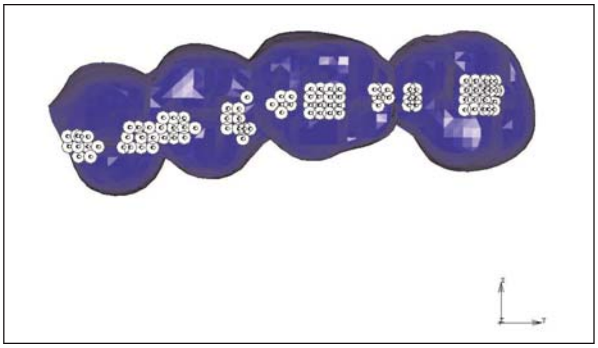

Fig 3 Selected nodes for loading.

RESULTS

VM Stresses Occurring on Implants

Figures were obtained from the buccal and palatinal aspects of implants and abutments to evaluate the changes related to different inclinations and designs. The highest VM stresses on implants and abutments were determined for vertical, oblique, and horizontal loading. Evaluation scales were between 0 and the highest value related to the direction of loading. A valid scale range enables the comparison of figures with each other. The scale was determined to be 0 to 54 MPa for vertical loading, 0 to 242 MPa for oblique loading, and 0 to 68 MPa for horizontal loading. The highest and lowest scale values had to be kept standard so that the same color did not represent different stress intervals in different figures (Figs 4 and 5).

The concentration of VM stresses at the neck region of the implant was observed. For this reason, 4 regions were selected at the buccal, palatinal, mesial, and distal aspects of the implants for determining

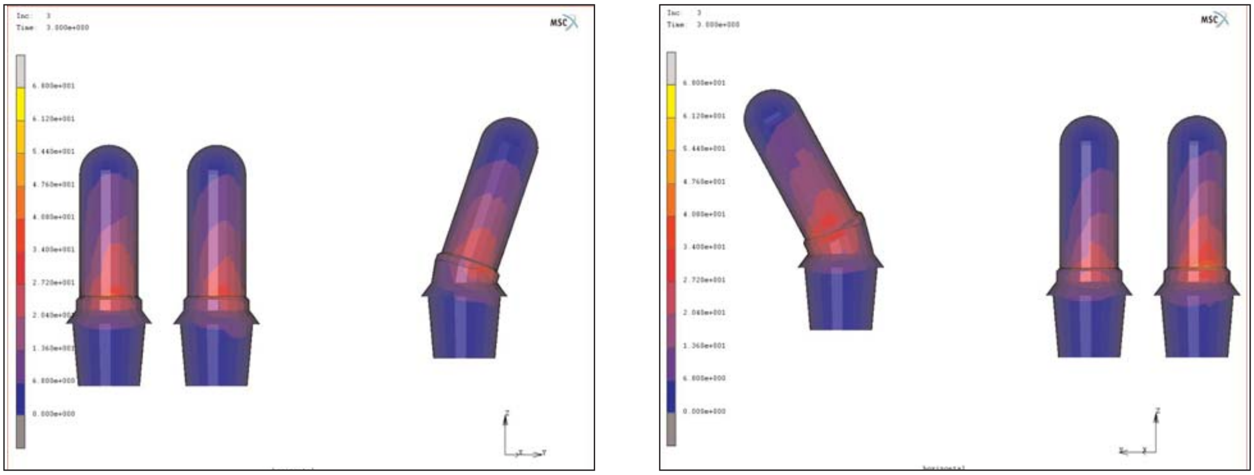

Fig 4 VM stresses on the implants in configuration 1 with horizontal loading (MPa). Molar implant was inclined at 30 degrees.

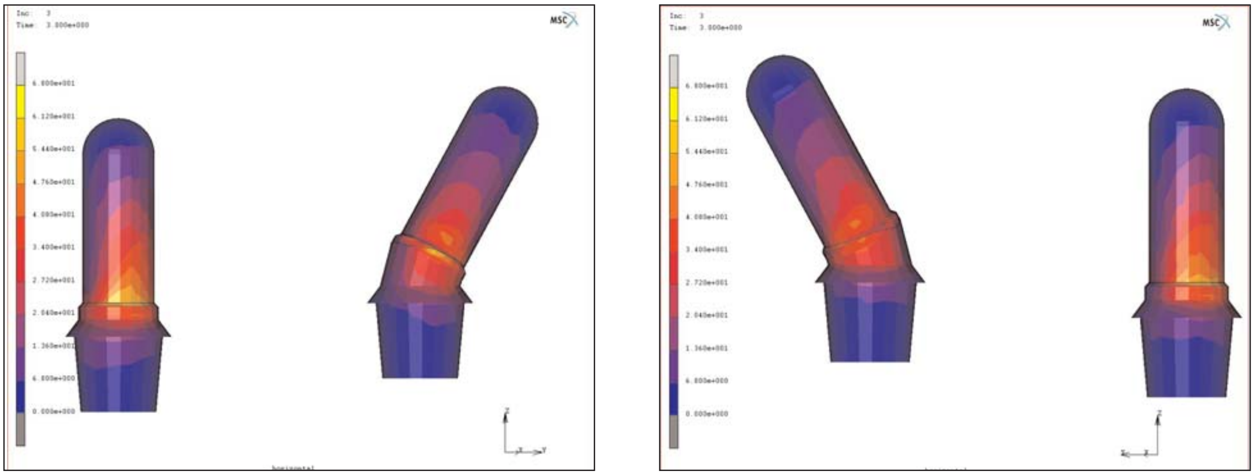

Fig 5 VM stresses on the implants in configuration 2 with horizontal loading (MPa). Molar implant was inclined at 30 degrees.

VM stresses (Figs 6 and 7). Mean VM stress values for 3 nodes in each region are given in Table 2 and Fig 8.

Evaluation of VM Stresses Occurring on Implants with Vertical Loading

VM stresses were concentrated in the neck region of the implant embedded in the molar area for the first design and in the premolar area for the second design. The highest stress values created were in the 36-MPa range and mesial to the neck region of the implant embedded in the premolar area in the second design.

Because of the increase in inclination of the implant, stress at the mesial, buccal, and palatinal points in the neck region of the molar area implant was reduced, while there was an increase in stress distally. VM stresses mesial and palatinal to the premolar area in the second design showed an increase approximately 12 times mesially and 3.5 times palatinally for the first design. Stresses in these areas were at least 1.5 times greater than stresses occurring on whole implant surfaces for each design (Table 2 and Fig 8a).

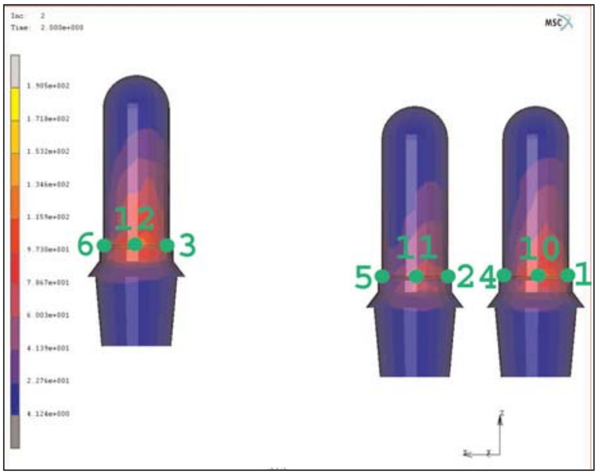

Fig 6 Selected nodes on the mesial, distal, and palatinal aspects of the implants for evaluation of VM stresses.

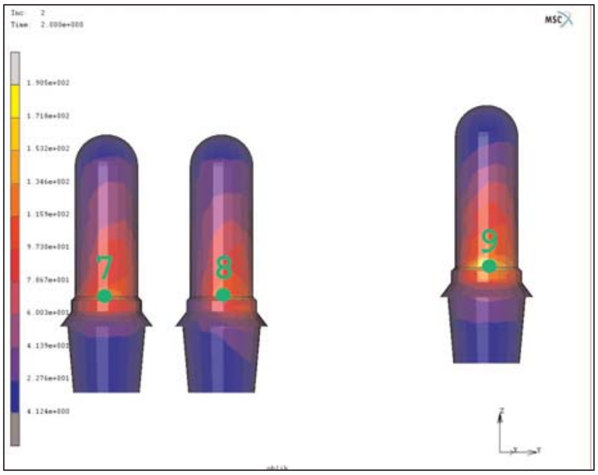

Fig 7 Selected nodes on buccal aspect of the implants for evaluation of VM stresses.

Table 2 VM Stresses on the Implants at Selected Nodes (MPa)

	Design 1			Design 2		
	Vertical	Oblique	Horizontal	Vertical	Oblique	Horizontal
0°						
1	9.33	79.07	21.73	–	–	–
2	2.78	33.90	7.66	36.69	105.83	28.79
3	14.46	47.90	12.90	14.46	58.75	15.30
4	9.08	33.88	11.08	–	–	–
5	18.77	69.77	15.45	7.96	64.00	19.36
6	12.11	67.39	19.20	6.35	65.67	19.30
7	5.72	125.23	33.11	–	–	–
8	8.67	117.18	29.36	9.12	186.78	51.05
9	21.72	155.74	35.00	11.15	184.22	40.55
10	14.04	112.16	38.53	–	–	–
11	9.40	73.04	25.25	32.65	116.95	48.69
12	8.31	117.10	35.31	10.45	151.89	48.13
15°						
1	9.74	80.03	22.01	–	–	–
2	3.06	33.92	9.51	36.64	101.67	28.44
3	9.59	53.77	13.85	9.80	55.04	15.56
4	9.36	37.06	12.21	–	–	–
5	19.29	72.28	15.93	7.31	67.44	20.22
6	16.71	76.88	21.88	9.11	72.77	21.47
7	6.03	125.92	33.18	–	–	–
8	8.97	119.32	29.86	9.35	188.92	51.67
9	17.63	119.38	26.74	7.07	131.81	34.74
10	14.10	113.39	38.83	–	–	–
11	9.35	74.37	25.63	33.21	118.88	40.48
12	4.78	91.25	26.81	7.00	121.58	37.74
30°						
1	10.88	77.66	21.43	–	–	–
2	3.52	35.66	9.89	36.92	103.42	28.95
3	8.77	57.71	14.67	7.30	63.96	17.60
4	9.32	35.51	11.48	–	–	–
5	20.66	77.93	17.08	7.17	71.51	21.37
6	22.78	97.36	23.23	13.50	90.00	25.79
7	6.52	128.62	33.74	–	–	–
8	9.71	123.35	30.74	9.78	194.20	52.45
9	14.25	104.34	23.85	7.37	128.54	33.73
10	14.13	116.28	33.75	–	–	–
11	9.44	76.61	26.31	33.08	122.57	50.20
12	3.97	78.22	22.89	5.53	105.08	32.34

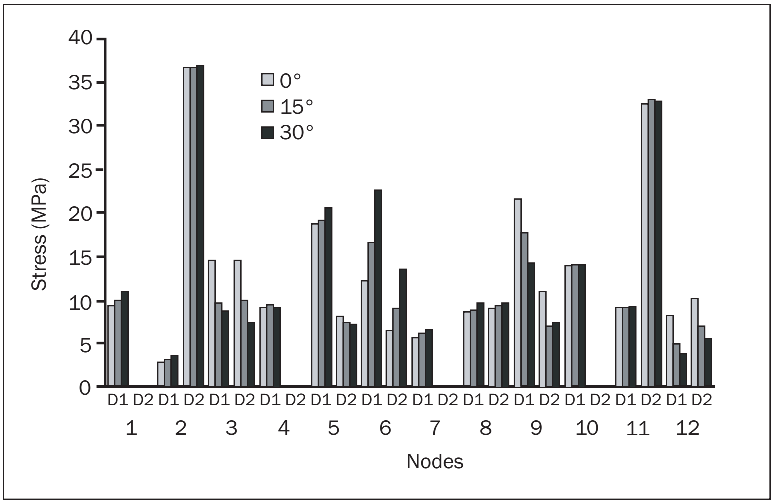

Fig 8a VM stresses on vertical loading at selected nodes (MPa). D1 = design 1; D2 = design 2.

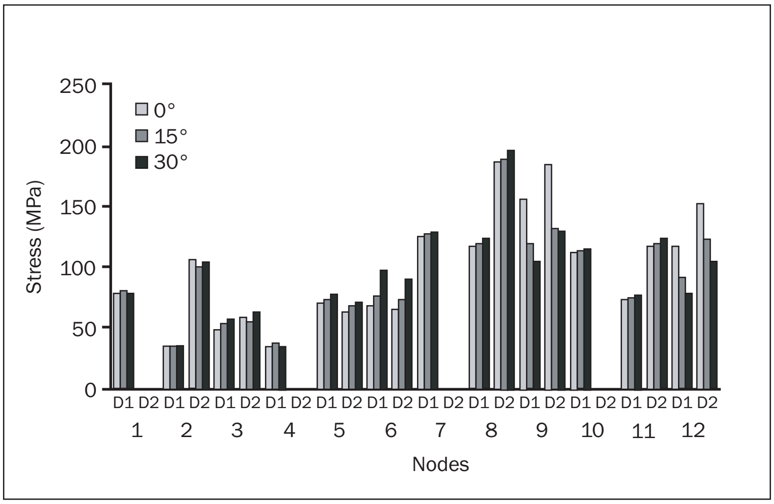

Fig 8b VM stresses on oblique loading at selected nodes (MPa). D1 = design 1; D2 = design 2.

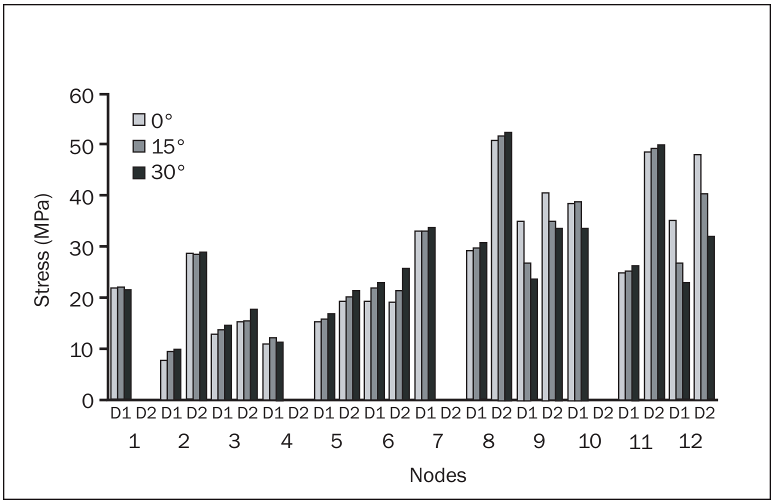

Fig 8c VM stresses on horizontal loading at selected nodes (MPa). D1 = design 1; D2 = design 2.

Evaluation of VM Stresses Occurring on Implants with Oblique Loading

VM stresses were concentrated at the neck region of the implants. The highest stress value observed was around 194.2 MPa on the buccal side of the Pm^2 implant for the second design. VM stresses formed at the buccal and palatinal sides of the implant were much higher compared to those mesial and distal to the implant. While there was an increase in stress because of the inclination distally to the implant in the molar region, a decrease was observed at the buccal and palatinal sides. According to the first design, 3 times more stress was observed mesially at the implant embedded in the premolar area for the second design. At the buccal and palatinal sides of the implant, stress increased 1.5 times (Table 2 and Fig 8b).

Evaluation of VM Stresses Occurring on Implants With Horizontal Loading

For horizontal loading, stress patterns were similar to those seen with oblique loading. Stress values were 3 times less than for oblique loading. The highest stress levels were confirmed buccally and palatinally at the neck of the implant embedded in the Pm^2 area for the second design at about 50 MPa (Table 2 and Fig 8c).

DISCUSSION

In the present study, a 3D model for maxillary bone was formed. Investigations and reports on 3D maxillary models are few. The complexity of the anatomy of maxillary bone compared to mandibular bone is an important factor.

It has been reported that more successful conclusions can be reached with FEA with ideal modeling of the anatomic structure.[5] There should be at least 30,000 to 200,000 elements and nodes. In this case, the size of the element must be 150 to 300 µm. It has been reported that deceptive results can be obtained if the element size is larger than 300 µm.[15,16] A total of 253,212 elements were used in this study.

Lekholm and Zarb classified bone quality in a study performed in 1985.[17] Jaffin and Berman[18] reported that type 3 bone is dominant in the anterior and premolar areas of the maxillary bone in both sexes. They found type 4 bone at the molar site but not bone of types 1 or 2. Lekholm and Zarb's basic system of classification has been used in various investigations.[18–21]

In this study, the implant was embedded in the molar area at a distance of 1 mm from the sinus base. The molar implant apex moves away from the sinus as the angle increases. Smith and Zarb[22] have emphasized the need to avoid harming the maxillary sinus, mandibular canal, and nasal cavity base in implant practice. In their investigation evaluating the clinical success of implant practice, they observed a failure rate of 30% for implants penetrating the sinus and the nasal cavity over 5 years.

In the present study, loading was applied from the regions where functional cusps of the mandibular teeth were in contact with maxillary teeth in a cusp–marginal ridge relation. In this way, loading can be transmitted parallel to the longitudinal axis of the implant. To provide this, reduction of the height of the palatinal contour and palatinal cusp is advised. It has been reported that occlusion can be obtained by a crossbite relationship in the cases of advanced bone resorption.[23–25]

In the present study, VM stresses formed on implants were evaluated. VM stress values are important criteria for ductile materials such as metal. VM stresses are relevant to the distortion energy responsible for plastic deformation. VM stress is used to define the yield point. When the yield point is overcome, material cannot behave elastically, and permanent deformation takes place.[13,26]

In this study, VM stress values and areas on implants were evaluated. The lowest stress values were seen for vertical loading, and the highest values for oblique loading. Stresses were concentrated at the contact areas of implants with cortical bone at each of 3 loading angles. At the same time, stresses formed at the buccal and palatinal sites of the implant in the molar region were concentrated distally because of the inclinations. For vertical loading, stresses at the neck of the implant in the premolar region increased 3.5 times with the second design over the first design and 2 times the oblique and horizontal loading. The highest VM stress formed on titanium with oblique loading was in the level of 194.2 MPa. Stress values obtained did not reach the endurance limit of titanium.

Clelland and associates[27] determined the highest stress levels to be at the neck region of the implant just underneath the bone crest. They have pointed out that the highest stresses at the implant were in the range of titanium's endurance limit (259.9 MPa). They also mentioned that normal occlusal forces did not cause fatigue of the metal. In their study, stress values at sites mesial and distal of the bone surrounding implants were lower than those at buccal and lingual sites.

White and colleagues[28] proposed that the highest stress values for cantilever usage are obtained in cortical bone distal to the distal implant for all cantilever lengths. If only 2 implants are to be used, a cantilever design should not be a choice. Stress doubles on the implant adjacent to the cantilever for a 3-unit prosthesis when 2 implant supports and a cantilever extension are used. Two implants cannot avoid a possible bending moment. Therefore it is important to evaluate the patient's functional habits.[3,29,30]

Van Zyl and coworkers[31] reported an increase in VM stresses for implant lengths exceeding cantilever length by 15 mm. They also emphasized that stresses formed buccally and lingually in the cortical bone surrounding implants were greater than those formed mesially and distally.

İplikçioğlu and Akça[13] have evaluated the effect of buccolingual inclination in implant-supported fixed prostheses applied to the posterior mandibular region by 3D FEA. Inclination increases the stresses on cortical bone, and the highest stress levels are developed with designs involving cantilevers.

In the present study, an increase in stresses on implants was found to be related to implant inclination in the molar region. A significant increase in stress on the implant embedded in the premolar region with the cantilever design was seen compared with the conventional prosthesis design.

CONCLUSION

The following results were obtained:

1. The lowest stress values were found for vertical loading, and the highest values for oblique loading.
2. VM stresses were concentrated at the implant–cortical bone contact areas for each of 3 loading angles. The values obtained at the necks of the implants were greater for buccal and palatinal sites than for mesial and distal sites.
3. Because of inclination, stresses induced at the buccal and palatinal sites of the implant in the molar region were concentrated distally.
4. For vertical loading, stresses at the neck of the implant in the premolar region were increased 3.5 times more with the second design than with the first design and 2 times more than for oblique and horizontal loadings. In the cantilever design, VM stresses increased greatly on the implant adjacent to the cantilever.
5. The highest VM stress value obtained in the study was about 194.2 MPa under oblique loading. This value did not exceed the endurance limit of pure titanium, which is 259.9 MPa.

REFERENCES

1. Gunne J, Jemt T, Linden B. Implant treatment in partially edentulous patients. A report on prostheses after 3 years. Int J Prosthodont 1994;7:143–148.
2. Lekholm U, van Steenberghe D, Herrmann I, et al. Osseointegrated implants in the treatment of partially edentulous jaws. A prospective 5-year multicenter study. Int J Oral Maxillofac Implants 1994;9:627.
3. Palacci P. Optimal Implant Positioning and Soft Tissue Management for the Brånemark System. Chicago: Quintessence, 1995.
4. Rangert B, Jemt T, Jörneus L. Forces and moments on Brånemark implants. Int J Oral Maxillofac Implants 1989;4:241–247.
5. Geng JP, Tan KBC, Liu G.R. Application of finite element analysis in implant dentistry: A review of the literature. J Prosthet Dent 2001; 85:585–598.
6. Iplikçioğlu H, Akça K, Çehreli MC. The use of computerized tomography for diagnosis and treatment planning in implant dentistry. J Oral Implantol 2002;28:29–36.
7. Krekmanov L, Kahn M, Rangert B, Lindstrom H. Tilting of posterior mandibular and maxillary implants for improved prosthesis support. Int J Oral Maxillofac Implants 2000;15:405–414.
8. Barbier L, Vander Sloten J, Krzesinski G, Schepers E, Van Der Perre G. Finite element analysis of non-axial versus axial loading of oral implants in the mandible of the dog. J Oral Rehabil 1998;25:847–858.
9. Teixeira ER, Sato Y, Akagawa Y, Shindoi N. A Comparative evaluation of mandibular finite element models with different lengths and elements for implant biomechanics. J Oral Rehabil 1998;25:299–303.
10. Gross MD, Arbel G, Hershkovitz I. Three-dimensional finite element analysis of the facial skeleton on simulated occlusal loading. J Oral Rehabil 2001;28:684–694.
11. Koolstra JH, van Eijden TMGJ, Weijs WA, Naeije M. A three-dimensional mathematical model of the human masticatory system predicting maximum possible bite forces. J Biomech 1988;21:563–576.
12. Meijer HJA, Starmans FJM, Steen WHA, Bosman F. Location of implants in the interforaminal region of the mandible and the consequences for the design of the superstructure. J Oral Rehabil 1994;21:47–56.
13. Iplikçioğlu H, Akça K. Comparative evaluation of the effect of diameter, length and number of implants supporting three-unit fixed partial prostheses on stress distribution in the bone. J Dent 2002;30:41–46.
14. Meijer HJA, Starmans FJM, Steen WHA, Bosman F. A three-dimensional finite element analysis of bone around dental implants in an edentulous human mandible. Arch Oral Biol 1993;38:491–496.
15. Sato Y, Teixeira ER, Tsuga K, Shindoi N. The effectiveness of a new algorithm on a three-dimensional finite element model construction of bone trabeculae in implant biomechanics. J Oral Rehabil 1999;26:640–643.
16. Sato Y, Wadamoto M, Tsuga K, Teixeira ER. The effectiveness of element downsizing on a three-dimensional finite element model of bone trabeculae in implant biomechanics. J Oral Rehabil 1999;26:640–643.
17. Lekholm U, Zarb GA. Patient selection and preparation. In: Brånemark P-I, Zarb GA, Albrektsson T (eds). Tissue-Integrated Prostheses: Osseointegration in Clinical Dentistry. Chicago: Quintessence, 1985:199–209.
18. Jaffin RA, Berman CL. The excessive loss of Brånemark fixtures in type IV bone: A 5-year analysis. J Periodontol 1991;62:2–4.

19. Holmes DC, Loftus JT. Influence of bone quality on stress distribution for endosseous implants. J Oral Implantol 1997;23:104–111.
20. Truhlar RS, Orenstein IH, Morris HF, Ochi S. Distribution of bone quality in patients receiving endosseous dental implants. J Oral Maxillofac Surg 1997;55:38–45.
21. Ulm C, Kneissel M, Schedle A, Solar P, Matejka M, Schneider B. Characteristic features of trabecular bone in edentulous maxillae. Clin Oral Implants Res 1999; 10: 459–467.
22. Smith DE, Zarb GA. Criteria for success of osseointegrated endosseous implants. J Prosthet Dent. 1989;62:567–572.
23. Misch CE. Contemporary Implant Dentistry. St Louis: Mosby, 1999.
24. Weinberg LA. Therapeutic biomechanics concepts and clinical procedures to reduce implant loading. Part I. J Oral Implantol 2001;27:293–301.
25. Weinberg LA. Therapeutic biomechanics concepts and clinical procedures to reduce implant loading. Part II: Therapeutic differential loading. J Oral Implantol 2001;27:302–310.
26. Lewinstein I, Banks-Sills L, Eliasi R. Finite element analysis of a new system (IL) for supporting an implant-retained cantilever prosthesis. Int J Oral Maxillofac Implants 1995;10:355–366.
27. Clelland NL, Ismail YH, Zaki HS, Pipko D. Three-dimensional finite element stress analysis in and around the Screw-Vent implant. Int J Oral Maxillofac Implants 1991;6:391-398.
28. White SN, Caputo AA, Anderkvist T. Effect of cantilever length on stress transfer by implant-supported prostheses. J Prosthet Dent 1994;71:493-499.
29. Parel SM. Prosthesis design and treatment planning for the partially edentulous implant patient. J Oral Implantol 1996;22:31-33.
30. Renouard F. Risk Factors in Implant Dentistry: Simplified Clinical Analysis for Predictable Treatment. Chicago: Quintessence, 1999.
31. van Zyl PP, Grundling NL, Jooste CH, Terblanche E. 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;10:51-55.