INFLUENCE OF CANCELLOUS BONE RIGIDITY ON STRESS DISTRIBUTION IN BONE AROUND DENTAL IMPLANT: A FINITE ELEMENT STUDY

Hasanen H. Al-Khafagy

BDS, MSc, PhD Conservative Department, College of Dentistry, Ajman University of Science And Technology, United Arab Emirates.

Abstract

Using the finite element method (FEM), this study sought to investigate how the Young's modulus of cancellous jaw bones influenced stress distribution in bone surrounding a dental implant. Six models of bone with osseointegrated dental implant, with different modulus of elasticity values of cancellous bone, were used. The stress values of the six models loaded with vertical force were analyzed. The results have shown that the cancellous bone rigidity has a great effect on the stress distribution. Highly rigid cancellous bone results in even stress distribution and reduction in maximum equivalent stress. The influence of bone rigidity may extend even to cortical bone.

(J Int Dent Med Res 2010; 3: (1), pp. 11-14)

Keywords: Stress analysis, implant, bone, finite element.

Received date: 29 October 2009 **Accept date:** 19 January 2010

Introduction

Finite element analysis is a numerical technique based on the principle of subdividing a structure into a number of finite elements which are interconnected with each other at the nodal points. These nodes are subjected to certain loading conditions, resulting in a behavior of the model similar to that of the structure it represents. The finite element method also allows for the study of stress patterns in two or more dissimilar materials adjacent to each other without affecting their independent behavior[1].

Finite element models are commonly used in implant dentistry to predict the effect of implant geometry, prosthesis design, and type of loading on the stress and strain distribution in the peri-implant region[2,3].

The modulus of elasticity (or Young's modulus) is a measure of the material rigidity; it varies as a function of both the density and microstructure.

Accurate values of modulus of elasticity are needed for finite element modeling of the jaws. The modulus of elasticity is measured in MPa[4].The actual value of Young's modulus for any material is normally determined by carrying out a standard tensile test on a specimen of the material[5].

In the literature, a wide range of elastic properties of bone is reported. The reported values for the Young's modulus are within a range of 7000 to 30000 MPa for mature cortical bone while the value for the cancellous bone are within a range of 1.1 to 9800 MPa[1,4,6,7,8]. These variations result from differences in anatomical position[9,10], loading direction, methods of storage and testing conditions. Also age seems to be a factor influencing these material properties of bone.

For the finite element model, the assignment of accurate values for material properties is an essential step to ensure predictive accuracy. Because of the lack of availability of elastic modulus properties of mandibular cancellous bone, the properties of bone from other parts of the body are frequently used in the finite element models[11]. The modulus of elasicity has been reported for cortical bone from dentate mandible[12]. To date, elastic properties have not been reported for cancellous bone in either the dentate or edentulous mandible[1].

***Corresponding author:**
Dr Hasanen H. Al-Khafagy. BDS, MSc, PhD
Conservative Department, College of Dentistry,
Ajman University of Science And Technology.
United Arab Emirates

Email: dentalhasanen@yahoo.com

Material and Methods

The two-dimensional finite element model includes a section of mandible at the canine area and dental implant. This modeled section of the mandibule was composed of spongy and cortical bones and was 10 mm in thickness buccolingually, surrounded by 2 mm of cortical bone. Cylindrical implant, 4 mm in diameter and 12 mm length was placed in the mandibule model, and superstructure 4 mm in diameter and 8 mm in height, was also modeled[3].

The load which was used in this study was 100 N to the most coronal part of the crown[4]. An ANSYS program, version 5.4 (Swanson Analysis System, Houston, Pennsylvania), was used to perform the two-dimensional finite element modeling and finite element analysis. The total number of the elements used in the finite element model was 4807[13].

Material property values were assigned in this study and assumptions were made on the basis of previously published data. The materials were homogeneous, isotropic and linearly elastic and the bone/implant interface was assumed to be 100% osseointegrated. The properties of the materials which will be used in this study (Cortical bone, cancellous bone and Titanium) are listed in tables 1 and 2.

Six osseointegrated implants with bone models with different value of modulus of elasticity of cancellous bone were used.

Stress levels, according to von Mises criteria, were calculated. Von Mises stresses are commonly reported in finite element analysis studies to summarize the overall stress state.

The material	Young's moduli (in MPa)	Poisson's ratio
Titanium	110000	0.3
Cortical bone	13700	0.3

Table 1. The properties of the cortical bone and titanium.

Model No.	Young's moduli (in MPa)	Poisson's ratio
Model No.I	6850	0.3
Model No.II	2740	0.3
Model No.III	1370	0.3
Model No.IV	685	0.3
Model No.V	457	0.3
Model No.VI	274	0.3

Table 2. The properties of the cancellous bone of the six models.

Results

Numerical equivalent stress values were determined by the von Misses failure theory at the cortical and cancellous bones. The results of the stress are illustrated in the figures 1 and 2.

The stress in the first model doesn't exceed 21.1 MPa while it reaches 84.02 MPa in the sixth model. The stresses in all the six models are higher in the cortical bone than in the cancellous bone.

The stress in the first model, with high modulus of elasticity which is about ½ of cortical bone, has shown even stress distribution. While the models with lower values of modulus of elasticity of the spongy bone stress distribution has appeared with uneven distribution. The stress also concentrated on the cortical bone, especially in the sixth model.

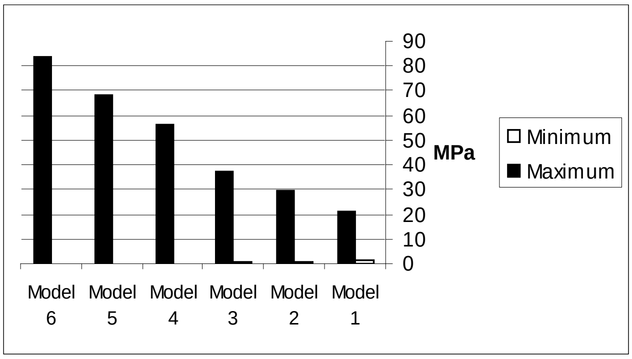

Figure 1. The maximum equivalent stress of six models with dental implant and different values of modulus of elasticity (in MPa).

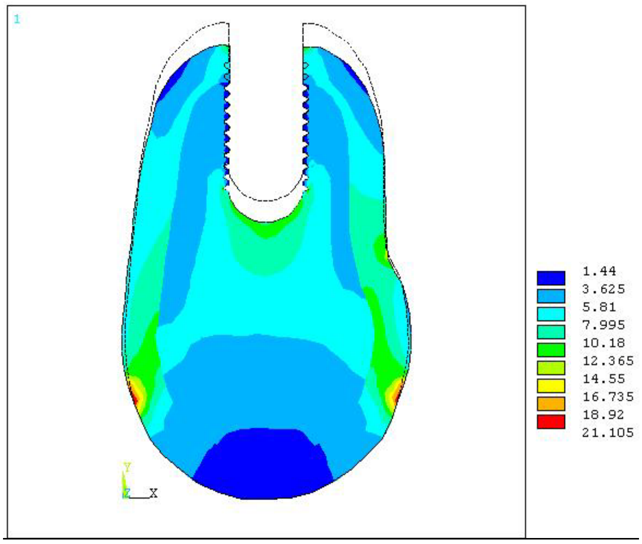

Figure 2a.

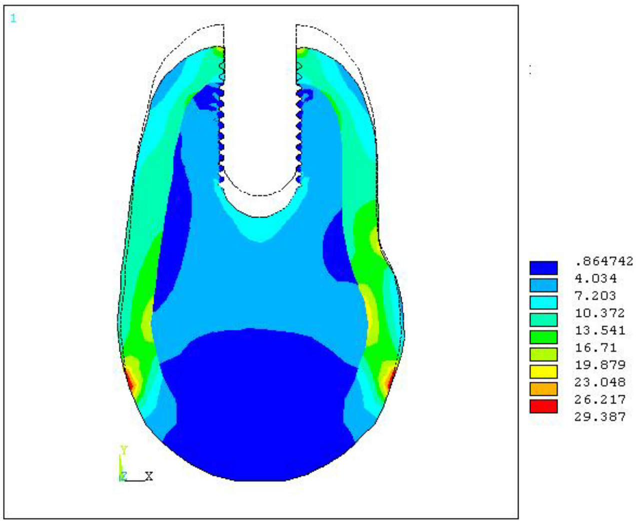

Figure 2b.

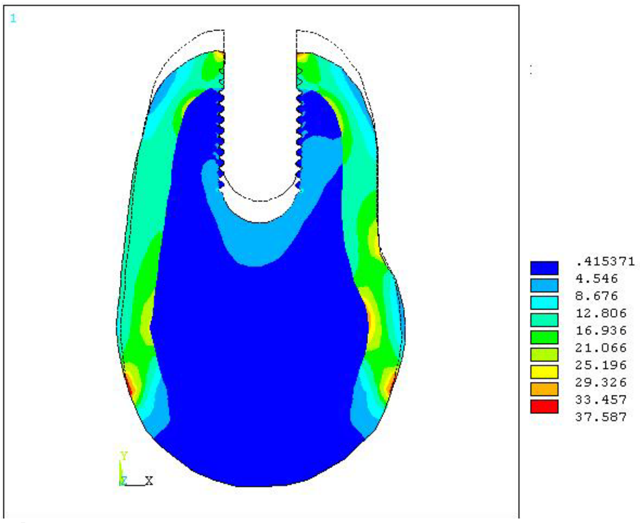

Figure 2c.

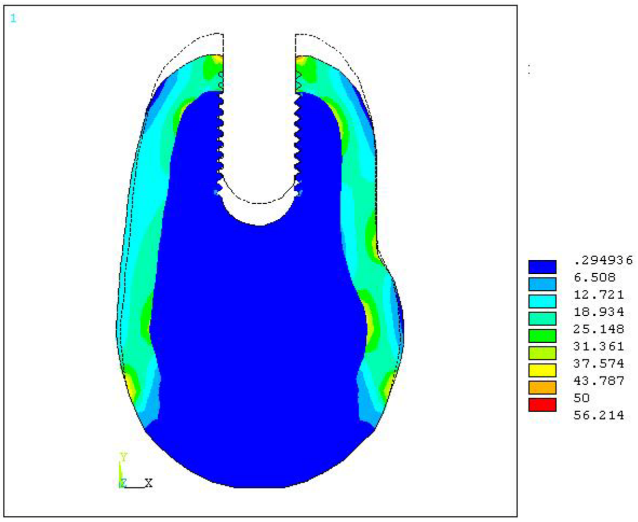

Figure 2d.

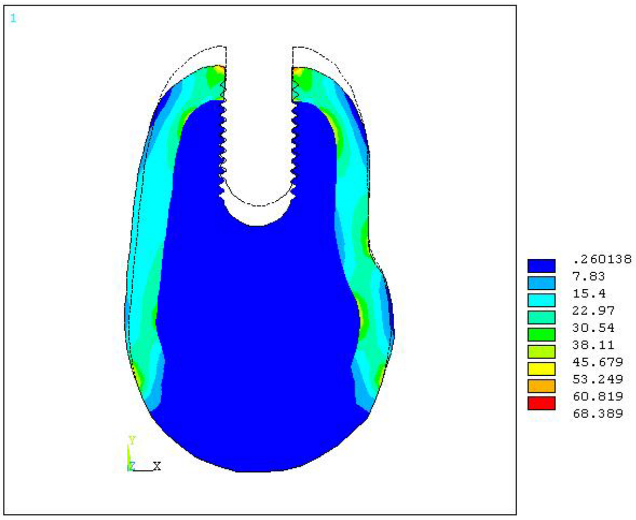

Figure 2e.

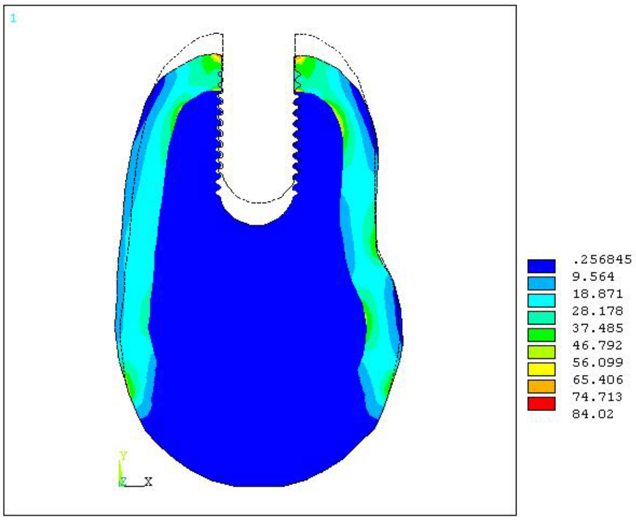

Figure 2f.

Figure 2a, b, c, d, e, f. Stress contour of the six models with dental implant at the cortical and cancellous bone and different value of modulus of elasticity where used. Modulus of elasticity of cancellous bone (A): 6850 MPa, (B): 2740 MPa, (C): 1370 MPa, (D): 685 MPa, (E): 457 MPa, (F): 274 MPa.

Discussion

Several factors appear to affect biomechanical failures of implant-supported prostheses. Resultant stresses, which are the product of occlusal loading, when excessive, are detrimental to the stability and longevity of the implant. Therefore it is important to consider the bone rigidity which result in the lowest levels of stress concentration and produce even distribution of these stresses. In 1992, Meijer, used the finite element method where the model of the mandible was either consisting of only a cortical bone or of a cancellous bone. By his model, he concluded that with an increase of modullous of elasticity the principal stresses in the bone around dental implants will become more extreme[14]. But his model of the mandible showed many shortages of bone representation.

The results that were obtained by the two-dimensional finite element method analysis suggest that the modulus of elasticity of the cancellous bone play a key role in good stress distribution. When a load is applied to the super structure, the results stress transfers to the bone surrounding the implant. When there is a small difference in the values of the modulus of elasticity of both cortical and cancellous bone, the stress will be distributed evenly. Since a dense bone has the ability to bear the stress which is applied on it, while in the case of great difference, (model number six), the stress will be concentrated on the cortical bone.

The relationship between the values of modullous of elasticity of cancellous bone and the maximum equivalent stress is linear. As the values of elasticity decrease, the stress in the bone around the implant increases.

Conclusions

In conclusion, it was confirmed that von Misses equivalent stress was sensitive to the Young's modulus of cancellous jaw bones. Rigid cancellous bone results in even stress distribution and reduction in maximum equivalent stress especially at the cortical bone around dental osseointegrated implants.

Declaration of Interest

The authors report no conflicts of interest. The authors alone are responsible for the content and writing of the article.

References

1. Tuncelli B, Poyrazoglu E, Koyoglu AM, Tezcan S. Comparison of load transfer by implant abutment of various diameters. Eur J Prosth Rest Dent 1997; 5-2:79-83.
2. Assunção WG, Barão VA, Tabata LF, Gomes EA, Delben JA, dos Santos PH. Biomechanics studies in dentistry: bioengineering applied in oral implantology. J Craniofac Surg. 2009 Jul;20 (4) : 1173-7.
3. Akca K. Cehreli MC and Iplikcioglu H. A comparison of three dimensional finite element stress analysis with in vitro strain gauge measurements on dental implants. International J Prosthod 2002; 15, 2,115-121.
4. O'Mahony AM, Williams JL, Katz JO, Spencer P. Anisotropic elastic properties of cancellous bone from a human edentulous mandible. Clin Oral Impl Res 2000.11:415-21.
5. Hearn EJ. Mechanics of Materials. Volume (1) Second Edition, Pregamon Press. 1985. 369-375.
6. Papavasilliou G, Kamposiora P, Bayne SC, Felton DA. Three dimensional finite element analysis of stress-distribution around single tooth implant as a function of bony support, prosthesis type, and loading during function. J Prosth Dent 1996; 76, 6,633-40.
7. Farah JW, Craig RG, Meroueh KA. Finite element analysis of mandibular model. J Oral Rehabil 1988; 15,615-24.
8. Petrie CS, Williams JL. Shape optimization of dental implant designs under oblique using the ρ-version finite element method. J mechanics in medicine and biology. 2002; 2, 3&4; 339-345.
9. Seong WJ, Kim UK, Swift JQ, Heo YC, Hodges JS, Ko CC. Elastic properties and apparent density of human edentulous maxilla and mandible. Int J Oral Maxillofac Surg. 2009 Oct;38(10):1088-93. Epub 2009 Jul 31.
10. Seong WJ, Kim UK, Swift JQ, Hodges JS and Ko CC. Correlation between physical properties of jawbone and dental implant initial stability. J Prosthet Dent. 2009 May;101(5):303-318.
11. Guan H, van Staden R, Loo YC, Johnson N, Ivanovski S, Meredith N.Influence of bone and dental implant parameters on stress distribution in the mandible: a finite element study.Int J Maxillofac Implants.2009 Sep-Oct;24(5):866-876.
12. O'Mahony AM, Williams JL and Spencer P. Anisotropic elasticity of cortical and cancellous bone in the posterior mandible increases peri-implant stress and strain under oblique loading. Clin Oral Impl Res.2001; 12:648-657.
13. Al-Khafagy HH. A three dimensional finite element analysis of the effect of cortical bone thickness on the stress distribution around single0unite osseointegrated implant. International Dentistry SA.2006; 8, 5.
14. Meijer HJA. A biomechanical study on bone around dental implants in an edentulous mandible – a finite element analysis and a design for a radiographic analysis. [Met een samenvatting in het Nederland].1992, 91-103.