Advances in Engineering Software 40 (2009) 935–940

Contents lists available at ScienceDirect

Advances in Engineering Software

journal homepage: www.elsevier.com/locate/advengsoft

Optimization of the implant diameter and length in type B/2 bone for improved biomechanical properties: A three-dimensional finite element analysis

Liang Kong [a,1,*], Zexu Gu [b,1], Kaijin Hu [a], Hongzhi Zhou [a], Yanpu Liu [a], Baolin Liu [a]

[a] *Department of Oral and Maxillofacial Surgery, School of Stomatology, The Fourth Military Medical University, 145 West Changle St., Xi'an 710072, PR China*
[b] *Department of Orthodontics, School of Stomatology, The Fourth Military Medical University, Xi'an 710072, PR China*

ARTICLE INFO

Article history:
Received 10 September 2008
Received in revised form 24 November 2008
Accepted 1 December 2008
Available online 12 January 2009

Keywords:
Three-dimensional finite element analysis
Stress
Implant diameter
Implant length
Optimal selection

ABSTRACT

In this paper, effects of the implant diameter and length on the maximum equivalent stresses were evaluated in jaw bones, and maximum displacements examined in an implant–abutment complex by a finite element method. The implant diameter ranged from 3.0 mm to 5.0 mm, and implant length ranged from 6.0 mm to 16.0 mm. Results suggested that under axial load, the maximum equivalent stresses in cortical and cancellous bones decreased by 77.4% and 68.4% with the increasing of diameter and length respectively. Under buccolingual load, those decreased by 64.9% and 82.8%, respectively. The maximum displacements of implant-abutment complex decreased by 56.9% and 78.2% under axial and buccolingual load respectively. When the diameter exceeded 3.9 mm and the length exceeded 9.5 mm, the minimum stress/displacement was obtained. The evaluating targets were more sensitive to the diameter change than that of the length. Data indicated that the implant diameter affected stress distribution in jaw bone more than length did; and an implant diameter exceeding 3.9 mm and implant length exceeding 9.5 mm was the optimal selection for type B/2 bone in a cylinder implant by biomechanical considerations.

1. Introduction

Though the long-term success rate for mandibular and maxillary implants is high (95% and 90%, respectively) [1], implant failures still pose great frustration and disappointment for both patients and dentists. Therefore, it is crucial to study different strategies to prevent implant failures [2].

Utilizing wide-diameter implants have long been known to be one way to minimize implant failure. Its advantages include improved bone-implant contact [3], immediate placement in failure sites and a reduction in abutment stress and strain [4]. A wide diameter implant can also be used as an alternative to bone grafting in severely resorbed maxillae [5]. Meanwhile, studies showed that shorter implants had statistically lower success rates [6] and it was not recommended due to the belief that the small implant surface area could not dissipate all occlusal forces to prevent excessive stresses at the interface [7]. Long-term studies revealed a dramatic increase in failure of implants when the length was shorter than 7 mm [8]. However, there was no detailed and systematic study on these two parameters.

In this study, a three-dimensional finite element analysis method was designed to continuously and simultaneously study the effects of implant diameter and length. The optimal values for these two implant biomechanical parameters were determined in type B/2 bones.

2. Materials and methods

2.1. 3D Model design

A mandible segment with an implant and a superstructure was modeled on a personal computer, using a 3D program (Pro/E Wildfire, Parametric Technology Corporation, USA). A cross-section image of a mandible in the first premolar region was used to be better compared with existing study of our group [9]. The image was obtained from CT scanning and used as the basis for a solid model. The cross-sectional image was then extruded to create a three-dimensional mandible segment, which contained a thick layer of cortical bone surrounding the dense cancellous bone, i.e. type B/2 bone according to the Lekholm and Zarb classification [10]. The average thickness of the cortical bone in the crestal region, measured by Dicom software, varied from 1.3 mm to 2.0 mm; and the mesial and distal section planes were not covered by the cortical bone. Dimensions of the bone segment are shown in Fig. 1.

The geometry of the ITI® solid implant (Institute Straumann, Basel, Switzerland) was utilized as a reference to model a cylinder screwed implant. A 5-mm-high solid abutment was also modeled, which was simplified to one unit with the implant, as shown in

---

[*] Corresponding author. Tel.: +86 013630231909; fax: +86 29 84776102.
E-mail address: kongliang@fmmu.edu.cn (L. Kong).

[1] These authors contributed equally to this work and should be regarded as co-first authors.

0965-9978/$ - see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.advengsoft.2008.12.010

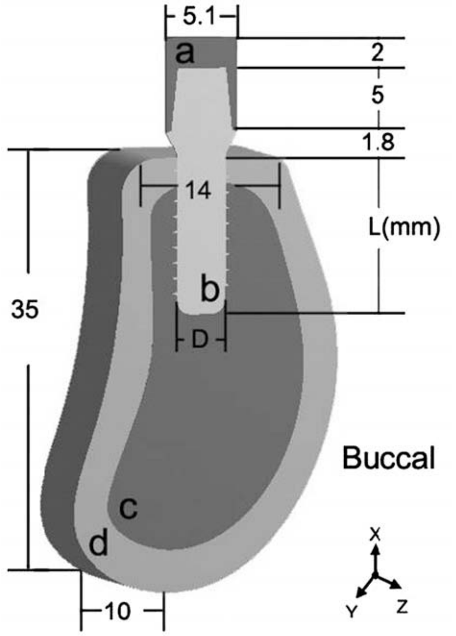

Fig. 1. Cross-sectional view on the symmetry plane of one model. *a* = superstructure; *b* = implant and abutment; *c* = cancellous bone; *d* = cortical bone; *D* = diameter of the implant (ranged from 3.0 mm to 5.0 mm); *L* = length of the implant (ranged from 6.0 mm to 16.0 mm).

Fig. 1. A full ceramic superstructure of 2-mm occlusal thickness was simplified as a cylinder and applied over the titanium abutment (Fig. 1). Diameter (*D*) and length (*L*) of the implant were set as the input variables. *D* ranged from 3.0 mm to 5.0 mm, while *L* ranged from 6.0 mm to 16.0 mm (Fig. 1). All models were meshed and analyzed by Ansys Workbench10.0 (SAS IP, Inc., USA).

2.2. Material properties

Material properties of orthotropy and isotropy were provided by the Ansys Workbench10.0 software. All materials used in the models were considered to be isotropic, homogeneous, and linearly elastic. The elastic properties were taken from the literature, as shown in Table 1.

2.3. Interface conditions

This study simulated an osseointegrated implant with screwed rough surface. For simulation, a “fixed bond” condition was set to its interface with the bone. The same contact was provided at the prosthesis–abutment interface study [11].

2.4. Elements and nodes

Models were meshed with 10-node-tetrahedron and 20-node-hexahedron elements. A finer mesh was generated around the implant (Fig. 2). Models were composed of 45,000 elements and 85,000 nodes on average.

Table 1
Mechanical properties of materials used in the 3D FEM models.

Materials	Young’s modulus (MPa)	Poisson ratio	References
Cortical bone	13,000	0.30	[11]
Cancellous bone	1370	0.30	
Titanium	102,000	0.35	
Ceramic	68,900	0.28	

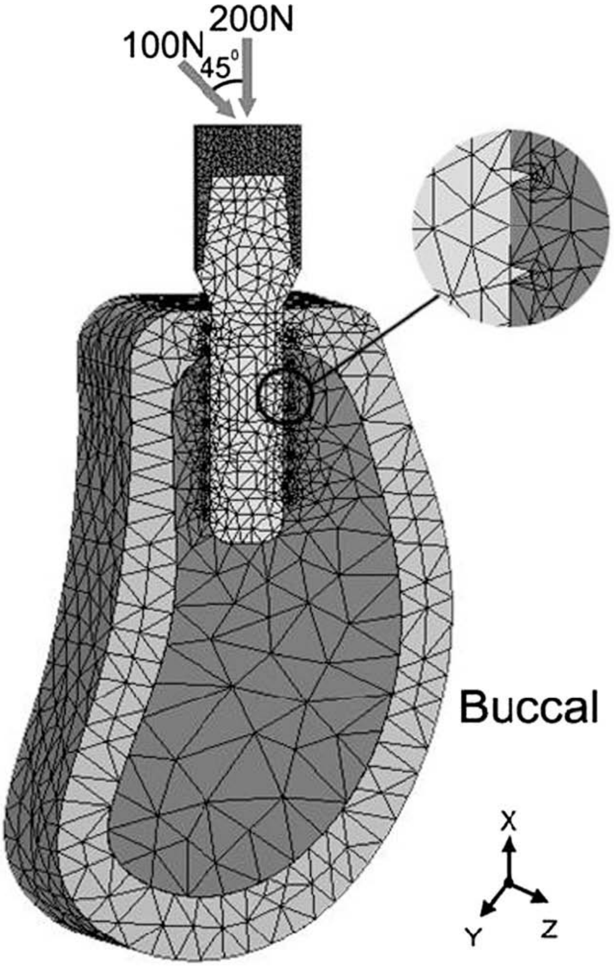

Fig. 2. Cross-sectional view on the symmetry plane of one meshed model and directions of the loads.

2.5. Constraints and loads

Models were constrained in all directions at the nodes on the mesial and distal bone surfaces. Forces of 200 N and 100 N were applied axially (AX) and buccolingually (BL), respectively (Fig. 2), to the middle point at the center of the superstructure [11]. The maximum equivalent Von Mises stress (Max EQV stress) in jaw bones and the maximum displacement (Max displacement) in the implant-abutment complex were set as output variables to evaluate the effect of different implant designs on the jaw bone and implant. Sensitivity of the output variables to input variables were also evaluated.

2.6. Convergence tests

In the current study, convergence tests with mesh refinements were performed. The Max EQV stress in jaw bones were employed for convergence monitoring and a tolerance of 3% was used. Changes less than 3% of the Max EQV stress in cortical and cancellous bones were considered convergence. An adaptive convergence was achieved when the mesh refinement loops was set to two.

2.7. Response surface construction and sensitivity analysis

Nine samples were analyzed for response surface construction. Utilizing the DesignXplorer program, sample values were generated via Latin Hybercube Sampling (LHS), a more advanced and