Materials Science and Engineering C 33 (2013) 4700–4714

Contents lists available at ScienceDirect

Materials Science and Engineering C

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

Stress analysis in bone tissue around single implants with different diameters and veneering materials: A 3-D finite element study

Joel Ferreira Santiago Junior [a], Eduardo Piza Pellizzer [a,*], Fellippo Ramos Verri [a], Paulo Sérgio Perri de Carvalho [b]

[a] Department of Dental Materials and Prosthodontics, UNESP, Univ Estadual Paulista, 1193 José Bonifácio Street, Araçatuba, São Paulo 16015050, Brazil
[b] Department of Surgery and Integrated Clinic, UNESP, Univ Estadual Paulista, 1193 José Bonifácio Street, Araçatuba, São Paulo 16015050, Brazil

ARTICLE INFO

Article history:
Received 1 March 2013
Received in revised form 15 June 2013
Accepted 20 July 2013
Available online 26 July 2013

Keywords:
Dental implants
Finite element analysis
Alveolar bone loss
Biomechanics

ABSTRACT

The aim of this study was to evaluate the stress distribution on bone tissue with a single prosthesis supported by implants of large and conventional diameter and presenting different veneering materials using the 3-D finite element method. Sixteen models were fabricated to reproduce a bone block with implants, using two diameters (3.75×10 mm and 5.00×10 mm), four different veneering materials (composite resin, acrylic resin, porcelain, and NiCr crown), and two loads (axial (200 N) and oblique (100 N)). For data analysis, the maximum principal stress and von Mises criterion were used. For the axial load, the cortical bone in all models did not exhibit significant differences, and the trabecular bone presented higher tensile stress with reduced implant diameter. For the oblique load, the cortical bone presented a significant increase in tensile stress on the same side as the loading for smaller implant diameters. The trabecular bone showed a similar but more discreet trend. There was no difference in bone tissue with different veneering materials. The veneering material did not influence the stress distribution in the supporting tissues of single implant-supported prostheses. The large-diameter implants improved the transference of occlusal loads to bone tissue and decreased stress mainly under oblique loads. Oblique loading was more detrimental to distribution stresses than axial loading.

1. Introduction

Biomechanical factors are important for longevity of osseointegrated implants [1–5]. Controlling these factors prevents mechanical complications, which include fracture of screws, components, or materials veneering the framework [6].

There is no consensus about the best veneering material for stress distribution [6–9]. Some researchers suggest that modified or acrylic resins reduce the force of impact [10]. On the other hand, some studies demonstrated no significant difference for models with different occlusal materials (e.g., porcelain and resin) [9,11]. Some studies using strain-gauge methodology evaluated different occlusal materials and found no difference for stress distribution between resin and porcelain crowns [11,12].

Furthermore, the diameter of the dental implants is an important factor for stress distribution in implants. A large-diameter implant is indicated for regions with sufficient bone thickness, poor bone quality (bone type IV), and immediate insertion of an implant after removal of a fractured implant [13]. The literature suggests that these implants improve stress distribution [13–18].

Concerning the methodology, the 3-D finite element method (3D-FEM) has been used to evaluate the loading performance of implant-supported prostheses and peri-implant bone [19,20].

Although some researchers have suggested biomechanical analysis in implantology to improve rehabilitation with implants, the literature is scarce regarding evaluation of the veneering material of single prostheses supported by implants with different diameters to guide the planning and development of materials for implant-supported restorations. The hypothesis of this article suggests that different occlusal materials do not influence stress distribution in the bone tissue and that the increase in diameter acts favorably in the distribution of stresses.

2. Material and methods

2.1. Experimental design

The experimental design follows previous studies [19–21]. Sixteen models were proposed with three variation factors: two diameters (3.75×10.00 mm and 5.00×10.00 mm), four occlusal materials (feldspathic porcelain, composite resin, acrylic resin, and alloy NiCr), and two conditions of load (axial and oblique loadings) (Table 1).

2.2. Three-dimensional FE modeling

Three-dimensional models were fabricated representing a section of mandibular bone with implant and crown. The bone block presented 25.46 mm in height, 13.81 mm in width, and 13.25 mm in thickness. It was composed of trabecular bone in the center surrounded by 1 mm of

* Corresponding author. Tel.: +55 1836363297; fax: +55 1836363245.
E-mail address: ed.pl@uol.com.br (E.P. Pellizzer).

0928-4931/$ – see front matter © 2013 Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.msec.2013.07.027

Table 1
Specifications of the models.

Model	Load	Diameter and length (implants)	Crown
1	Axial	3.75 × 10 mm	NiCr
2			Esthetic veneering of feldspathic porcelain
3			Esthetic veneering of composite resin
4			Esthetic veneering of acrylic resin
5		5.00 × 10 mm	NiCr
6			Esthetic veneering of feldspathic porcelain
7			Esthetic veneering of composite resin
8			Esthetic veneering of acrylic resin
9	Oblique	3.75 × 10 mm	NiCr
10			Esthetic veneering of feldspathic porcelain
11			Esthetic veneering of composite resin
12			Esthetic veneering of acrylic resin
13		5.00 × 10 mm	NiCr
14			Esthetic veneering of feldspathic porcelain
15			Esthetic veneering of composite resin
16			Esthetic veneering of acrylic resin

cortical bone in the region of the mandibular second molar, simulating type III bone [22].

The bone (trabecular and cortical) was obtained through computed tomography of the transversal section in the molar region transferred to the InVesalius® software (CTI, Campinas, São Paulo, Brazil). The InVesalius software creates three-dimensional virtual models using a two-dimensional tomographic image. After that, the model was transferred to the Rhinoceros® 3D 4.0 software (NURBS Modeling for Windows, Seattle, WA, U.S.) for final modeling of the surfaces.

The geometry of the threaded external hexagon implant with a size of 3.75 × 10 mm and external hexagon implant with size of 5.00 × 10 mm (Conexão® Master Screw, Sistemas de Próteses, Arujá, São Paulo, Brazil) was used as a reference for fabrication of the implant model, as well as the components of the framework. The implants were drawn and virtually simplified using the SolidWorks® 2006 software (SolidWorks Corp, Waltham, MA, U.S.).

A UCLA-type screwed crown was simulated with four different occlusal veneering materials: a NiCr metallic crown, a crown with a NiCr framework and feldspathic porcelain, a crown with a NiCr framework and composite resin, and a crown with a NiCr framework and acrylic resin. The mounting of the crown with the metallic framework and the implant was carried out using the SolidWorks® software for posterior insertion in the bone structure. The thickness of the veneer materials was, on average, 2.0 mm in the cusp and 1.5 mm in the region of the equator line, narrowing toward the cervical region.

The external surface of the crown was obtained using an artificial mandibular second molar of an experimental dental model (Odonfix®, Ind. Com. Mat. Odont. Ltda. São Paulo, Ribeirão Preto, Brazil) that was scanned (Roland DG, Cotia, São Paulo, Brazil). The three-dimensional image was transferred to the Rhinoceros® 3D 4.0 software for detailing of the surfaces and final mounting of the models, with insertion of the implant/crown assembly in the bone block. After this mounting, the assemblies were transferred to the FEMAP® 10 (Siemens PLM Software Inc., Plano, TX, U.S.) software, which allows pre- and post-processing of finite element models, importation of geometries, mesh generation, configuration of mechanical properties and material models, and simulation of physical performance.

After this stage, the finite element mesh was generated for analysis. Initially, the mechanical properties of each material were incorporated. The Young's modulus and Poisson's ratio were determined according to the literature [23–26] (Table 2). All materials were considered as isotropic, linear, and homogeneous.

After definition of the mechanical properties of the materials, the finite-element mesh for each structure was generated using the standard parabolic tetrahedral solid element of FEMAP 10, given that the simulated structures were solid (Fig. 1).

2.3. Interface conditions and boundary conditions

The loading and boundary conditions are in agreement with the literature [27–29]. The model was defined by establishing the boundary conditions (Fig. 2), restriction, and loading for simulation of real clinical situations. Thus, analysis was linear, considering that the performance observed was within linearity in spite of the complexity of the structure. The bone block was fixed in three planes on the lateral surfaces while the base was maintained as free or suspended (Fig. 2). All matching surfaces between the structures of the study were simulated by direct contact, which means that the contact avoids penetration, sliding, or movement between the surfaces. However, contact between the framework and the implant was juxtaposed to simulate a real clinical situation. The total axial loading of 200 N was divided over four areas (50 N) of the surface [30]. The loading was perpendicular to the chewing surfaces of each cusp. The oblique loading of 100 N was applied by suppression of the loads on the buccal cusps to simulate a real clinical situation (Fig. 2).

2.4. Analysis of the finite element model

The analysis was generated in the FEMAP 10 software and transferred for resolution to the solution nucleus of the finite-element software NEi Nastran® 9.0 (Noran Engineering, Inc., Westminster, CA, U.S.) to obtain the results. The number of nodes and elements for each implant was determined in Table 3.

After the analyses, the results were transferred to the FEMAP 10 software for graphic visualization of the stress and/or displacement maps. The processing analysis of the models was conducted in a Sun work station (Sun Microsystems Inc., São Paulo, Brazil) with the following characteristics: Opteron 64 processor, AMD double nucleus, 4 GB of RAM, and 250 GB of HD.

2.5. Criterion-stress analysis

FEMAP 10 software for graphics was used to visualize the stress and/or displacement maps. The results were visualized through maximum principal stress maps to indicate the levels and standards of stress concentration. This type of analysis is recommended for friable materials such as bone structure [18]. The unit of measurement was megapascals (MPa). Furthermore, the displacements were plotted on maps with values expressed in micrometers (μm). We added an analysis of von Mises (MPa) stresses that allowed an interpretation of the sum total of tensions in the region analyzed [20]. Some structures were individually plotted for better visualization of the results.

3. Results

3.1. Analysis of general displacement maps

Regarding the specific analysis of each crown for maximum displacement, it was observed that there was no difference in displacement (μm) between the diverse occlusal materials and the two diameters. The only difference was between the oblique and axial loads; the increase in the

Table 2
Mechanical properties of materials used in model.

Structures	Elastic modulus (GPa)	Poisson ratio (ν)	References
Trabecular bone	1.37	0.30	[23]
Cortical bone	13.7	0.30	[23]
Implant (titanium)	110.0	0.35	[24]
NiCr alloy	206.0	0.33	[26]
Composite resin	16.6	0.24	[24]
Acrylic resin	2.4	0.35	[24]
Feldspathic porcelain	82.8	0.35	[25]