Universidade Estadual de Campinas
Faculdade de Odontologia de Piracicaba

Jessica Mie Ferreira Koyama Takahashi

Análise tridimensional por elementos finitos da biomecânica de próteses implantossuportadas: influência da carga oclusal, tipo de conexão, angulação dos implantes e simplificações de modelagem

Tese apresentada à Faculdade de Odontologia de Piracicaba, da Universidade Estadual de Campinas para obtenção do Título de Doutor em Clínica Odontológica – Área de Prótese Dental

Piracicaba

– 2011 –

Jessica Mie Ferreira Koyama Takahashi

Cirurgiã Dentista

Análise tridimensional por elementos finitos da biomecânica de próteses implantossuportadas: influência da carga oclusal, tipo de conexão, angulação dos implantes e simplificações de modelagem

Tese apresentada à Faculdade de Odontologia de Piracicaba, da Universidade Estadual de Campinas para obtenção do Título de Doutor em Clínica Odontológica – Área de Prótese Dental

Orientador: Prof. Dr. Marcelo Ferraz Mesquita

Piracicaba

– 2011 –

FICHA CATALOGRÁFICA ELABORADA PELA

BIBLIOTECA DA FACULDADE DE ODONTOLOGIA DE PIRACICABA

Bibliotecária: Elis Regina Alves dos Santos – CRB-8[a] / 8099

T139a	Takahashi, Jessica Mie Ferreira Koyama. Análise tridimensional por elementos finitos da biomecânica de próteses implantossuportadas: influência da carga oclusal, tipo de conexão, angulação dos implantes e simplificações de modelagem / Jessica Mie Ferreira Koyama Takahashi. -- Piracicaba, SP: [s.n.], 2011. Orientador: Marcelo Ferraz Mesquita. Tese (Doutorado) – Universidade Estadual de Campinas, Faculdade de Odontologia de Piracicaba. 1. Implante dentário. 2. Prótese dentária. 3. Biomecânica. I. Mesquita, Marcelo Ferraz. II. Universidade Estadual de Campinas. Faculdade de Odontologia de Piracicaba. III. Título. (eras/fop)

Título em Inglês: Tridimensional finite element analysis of biomechanics of implant-supported prosthesis: influence of occlusal load, implant connection system, implant tilting and modeling simplifications

Palavras-chave em Inglês (Keywords): 1. Dental implantation. 2. Dental prosthesis. 3. Biomechanics

Área de Concentração: Prótese Dental

Titulação: Doutor em Clínica Odontológica

Banca Examinadora: Marcelo Ferraz Mesquita, Rafael Leonardo Xediek Consani, Guilherme Elias Pessanha Henriques, Pedro Yoshito Noritomi, Wirley Gonçalves Assunção

Data da Defesa: 14-02-2011

Programa de Pós-Graduação em Clínica Odontológica

UNIVERSIDADE ESTADUAL DE CAMPINAS
Faculdade de Odontologia de Piracicaba

A Comissão Julgadora dos trabalhos de Defesa de Tese de Doutorado, em sessão pública realizada em 14 de Fevereiro de 2011, considerou a candidata JESSICA MIE FERREIRA KOYAMA TAKAHASHI aprovada.

Prof. Dr. MARCELO FERRAZ MESQUITA

Prof. Dr. WIRLEY GONÇALVES ASSUNÇÃO

Prof. Dr. PEDRO YOSHITO NORITOMI

Prof. Dr. RAFAEL LEONARDO XEDIEK CONSANI

Prof. Dr. GUILHERME ELIAS PESSANHA HENRIQUES

Dedico este trabalho...

Aos meus pais, ***Walter e Marta***, pelo carinho e apoio, pela confiança em mim e em minhas aspirações.

Ao meu namorado e amigo, ***Hugo***, por estar sempre presente. Pela paciência, apoio e compreensão nos momentos difíceis. Pelas alegrias e felicidades compartilhadas.

Amo vocês.

Agradecimentos especiais

Ao meu orientador, ***Prof. Dr. Marcelo Ferraz Mesquita***, Titular da Área de Prótese Total da Faculdade de Odontologia de Piracicaba – UNICAMP, pela oportunidade de realizar este e outros trabalhos, pela seriedade, competência, convivência e ensinamentos transmitidos ao longo de minha Pós-Graduação.

Ao meu pai, ***Engº. Walter Kenkiti Takahashi***, pela paciência em compreender e esclarecer minhas dúvidas. Por me auxiliar na realização deste trabalho, pelos ensinamentos e tempo despendido.

Meus agradecimentos

À ***Faculdade de Odontologia de Piracicaba – UNICAMP,*** na pessoa do seu Diretor ***Prof. Dr. Jacks Jorge Júnior*** e do Diretor Associado ***Prof. Dr. Alexandre Augusto Zaia*** pela oportunidade da realização do Programa de Pós-Graduação em Clínica Odontológica.

À Coordenadora Geral da Pós-Graduação ***Profª. Drª. Renata Cunha Matheus Rodrigues Garcia*** e ao Coordenador do Programa de Pós-Graduação em Clínica Odontológica ***Prof. Dr. Márcio de Moraes.***

À ***Coordenação de Aperfeiçoamento de Pessoal de Nível Superior - CAPES*** pela bolsa de estudos a mim concedida.

À ***Neodent***, pelo apoio ao desenvolvimento deste projeto.

Aos ***funcionários do SMF-INPE***, por me receberem em suas instalações durante a realização deste trabalho.

À ***Profª. Drª. Gláucia Maria Bovi Ambrosano,*** Professora da Área de Bioestatística pelas orientações e pela possibilidade de estagiar sob sua supervisão. Por ampliar meus horizontes e me proporcionar uma formação acadêmica mais completa.

Aos docentes ***Prof. Dr. Guilherme Elias Pessanha Henriques, Prof. Dr. Mauro Antônio de Arruda Nóbilo e Prof. Dr. Rafael Leonardo Xediek Consani*** pelo aprendizado e convívio durante estes anos.

Aos membros das bancas do Exame de Qualificação, ***Profª. Drª. Célia Marisa Rizzatti Barbosa, Prof. Dr. Mauro Antônio de Arruda Nóbilo e Prof. Dr. Wander José da Silva,*** e do Exame de Defesa, ***Dr. Pedro Yoshito Noritomi, Prof. Dr. Wirley Gonçalves Assunção, Prof. Dr. Guilherme Elias Pessanha Henriques, Prof. Dr. Rafael Leonardo Xediek Consani e Prof. Dr. Marcelo Ferraz Mesquita,*** pelas correções, considerações e sugestões para o aprimoramento deste trabalho.

Aos ***docentes do Curso de Pós-Graduação em Clínica Odontológica***, pelos ensinamentos que contribuíram à minha formação acadêmica.

Aos amigos ***Leonardo Luthi, Maria Humel, Maíra Serra e Silva, Andreza Dayrell,*** companheiros em todos os momentos... que tornaram este período muito mais agradável. Amizades verdadeiras são difíceis de encontrar.

À aluna ***Claudia Brilhante*,** pela dedicação, seriedade e competência na realização de seu trabalho de Iniciação Científica. Pela inspiração em ensinar e o orgulho de te ver prosperar.

Aos colegas de Pós-Graduação ***Juliana Nuñez-Pantoja, Brunna Moreira, Mariana Agustinho, Caroline Odo, Izabela Patta, Manoela Capla, Ana Patrícia Macêdo, Ana Paula Farina, Antônio Pedro Ricomini, Sabrina Rodrigues, Ataís Bacchi, Mateus Bertolini, João Paulo Silva e Gabriela Cassaro,*** pelas experiências trocadas e momentos de descontração.

Aos técnicos de laboratório, ***Eduardo Pinez Campos e Maria Elisabete Cardenas*** pela agradável convivência e auxílio, durante estes anos de pós-graduação.

À secretária do Departamento de Prótese e Periodontia, ***Eliete Ap. F. L. Marim*** e às estagiárias ***Mônica L. B. Penzani*** *e* ***Ketney F. Lopes*** pela simpatia, atenção e auxílio durante minha pós-graduação.

A todos que indiretamente contribuíram para a realização deste trabalho.

Muito obrigada.

"Não existem métodos fáceis para resolver problemas difíceis."

René Descartes

"To thrive, to seek, to find and not to yield."

Lord Alfred Tennyson

Resumo

O objetivo nos presentes estudos foi avaliar, por meio de análise tridimensional de elementos finitos, os efeitos nos valores e padrões de distribuição de tensões de modelo virtuais de próteses implantossuportadas na presença de: 1) Simplificações de modelagem das geometrias de superfícies com rosca; 2) Variação do padrão de cargas oclusais e tipo de conexão de implantes; 3) Inclinação do implante e necessidade do uso de componentes angulados. No primeiro estudo foram confeccionados modelos virtuais simulando a interface entre o osso de uma seção posterior de mandíbula e um implante, e a interface entre o componente protético e o parafuso protético. As interfaces foram simuladas com variação da geometria das roscas e tipo de contato entre as estruturas. Os modelos da interface osso/implante receberam carga oblíqua de 180N, e os da interface abutment/parafuso carga de 41N. Foi observado, na interface implante/osso, que os valores de tensão variam de acordo com a geometria modelada e o contato entre as estruturas e que tensões artificialmente elevadas podem ser observadas em regiões pontuais. Na interface abutment/parafuso, a geometria modelada não influenciou os resultados obtidos. No segundo estudo, foram confeccionados modelos virtuais simulando próteses fixas suportadas por implantes de conexão do tipo hexágono externo, hexágono interno, e cone morse; submetidos à aplicação de cargas axiais ou oblíquas sobre a superfície da infraestrutura modelada (180N em primeiro pré-molar e primeiro molar; 280N em segundo pré-molar). Foi observado que mediante aplicação de cargas oblíquas os valores de tensão apresentaram aumento significativo, e que o local de concentração das tensões também foi alterado para regiões mais inferiores nas estruturas avaliadas. Os modelos de montagens com implantes de conexão interna apresentaram padrão de distribuição de tensão mais favorável aos tecidos de suporte. No terceiro estudo, modelos virtuais de próteses fixas implantossuportadas foram simulados com implantes distais posicionados de maneira inclinada. Os modelos foram confeccionados utilizando conexões do tipo hexágono externo e cone morse, sendo utilizados componentes com 17° de angulação. Os modelos foram submetidos às mesmas condições de carregamento citadas no estudo anterior. Foi observado que mediante inclinação dos implantes distais, não há diferença significativa nos valores de tensão nos tecidos de suporte, havendo apenas alteração na área de concentração de tensão. A tensão foi mais concentrada no osso trabecular ao redor do implante posicionado verticalmente. As conexões avaliadas apresentaram valores de tensão semelhantes nos tecidos de suporte, enquanto os modelos de conexão cone morse apresentaram maior tensão nos componentes protéticos. Desta maneira, concluiu-se que: 1) a simplificação da geometria de roscas para análise utilizando o método de elementos finitos gera valores e padrão de distribuição de tensão confiáveis; 2) a aplicação de cargas oblíquas gera aumento significativo na tensão nas estruturas ósseas; 3) o tipo de conexão do implante pode interferir na tensão em componentes protéticos e osso; 4) a inclinação de implantes não gera maior tensão nos tecidos de suporte.

Palavras-chave: implante dental, análise de tensões, elementos finitos, abutment angulado, implante inclinado

Abstract

The aim of the following studies was to evaluate with finite element analysis, the effect on stress values and distribution pattern of virtual models of implant-supported prostheses in the presence of: 1) Modeling simplification of threaded surfaces geometry; 2) Variation of occlusal load pattern and implant connection type; 3) Implant inclination and the use of angled abutments. On the first study, virtual three-dimensional models were obtained simulating the interface between bone of a posterior section of the lower jaw and a dental implant, and the interface between an abutment and a prosthetic screw.

The screwed interfaces were simulated with variation of thread geometry and contact between the structures. Bone/implant models were loaded with 180N oblique force, while abutment/screw models were loaded with 41N force. It was observed, on the implant/bone interface that stress values depend of the modeled geometry and the amount of contact between the surfaces; and that artificially high stress might be observed in located areas. On the abutment/screw interface, modeled geometry did not influence the results. For the second study, virtual models were obtained simulating three-element fixed partial dentures supported by two implants of external-hexagon, internal-hexagon or morse-taper connection. The models were submitted to axial or oblique loading (180N on first premolar and first molar, 280N on second premolar). It was observed that oblique loading significantly increased stress values, and changed stress concentration areas. Also, the models with internal connection implants presented stress distribution pattern more favorable to the supporting structure. On the third and final study, virtual models of implant-supported frameworks were obtained with inclined distal implants with external-hexagon or morse-taper connection and 17^{o} angled abutment. All models were submitted to the same loading conditions previously described. It was observed that inclined implants do not significantly increase stress on supporting tissue, with changes only on stress concentration area. Higher stress was concentrated at trabecular bone around the straight implant. The evaluated connection systems presented similar stress values on the supporting tissue, while morse-taper models presented higher stress on the prosthetic components than the external-hexagon. Thus, it was concluded that: 1) finite element models, with simplified representation of the thread interface present reliable stress values and distribution patterns; 2) oblique loading of the framework increases stress values on the supporting tissues; 3) implant connection system may interfere on stress on prosthetic components and bone; 4) implant inclination does not increase stress on supporting tissue.

Key-words: dental implants, stress analysis, finite element, angled abutment, inclined implant

Sumário

Página

1. Introdução

O conceito de reabilitação dental e as possibilidades de tratamento foram modificados com o desenvolvimento dos implantes osseointegrados. A partir da década de 70, quando Brånemark descreveu a osseointegração (Branemark *et al.*, 1977) foram realizados diversos estudos longitudinais que demonstraram a confiabilidade e longevidade deste tratamento, apresentando taxas de sucesso de 97,7% (Jung *et al.*, 2008; Jemt, 2008). Existe uma diferença clara entre a taxa de sucesso dos implantes e a das reabilitações protéticas (Jemt, 2008); entretanto foi relatada taxa de sucesso de 96,6% para reabilitações parciais implantossuportadas (Halg *et al.*, 2008).

De modo geral, as falhas das reabilitações sobre implantes podem ocorrer em duas fases. A primeira, após a colocação do implante, principalmente em decorrência de inflamações e a segunda, devido à remodelação óssea ao redor do implante (Sethi *et al.*, 2000).

Desta maneira, fatores biomecânicos devem ser amplamente considerados durante o planejamento de reabilitações sobre implantes, pois podem ser responsáveis pela reabsorção óssea e fratura de componentes e infraestruturas protéticas (Sahin *et al.*, 2002). A transferência das forças oclusais para a interface osso/implante pode depender do tipo de carga oclusal à que a reabilitação é submetida, do material de confecção da prótese, da natureza da interface osso/implante, da quantidade e qualidade óssea e da geometria do implante utilizado (Geng *et al.*, 2001; Eskitascioglu *et al.*, 2004).

As cargas oclusais às quais as reabilitações protéticas são submetidas são originárias dos movimentos mastigatórios que induzem forças axiais e oblíquas. Estas forças aplicadas sobre a superfície oclusal das próteses afetam diretamente as tensões geradas no implante e nas estruturas ósseas (Geng *et al.*, 2001), considerando que o carregamento oclusal excessivo pode induzir reabsorção óssea e comprometer a longevidade do implante e da reabilitação. A tensão induzida pelas cargas oclusais é decorrente da associação entre o tipo e distribuição destas cargas, das propriedades mecânicas das estruturas envolvidas e do *design* do implante, da conexão protética e da infraestrutura (Barbier *et al.*, 1998).

Com relação ao *design* do implante, o perfil das roscas do implante (Geng *et al.*, 2004; Kong *et al.*, 2008) e a forma do implante (Degerliyurt *et al.*, 2010) tem sido relatados como fatores que podem influenciar a distribuição das forças oclusais para os tecidos ósseos de suporte. Além destas variações geométricas, o tipo de conexão entre o implante e o componente protético pode também influenciar a distribuição de tensões no osso e componentes protéticos (Maeda *et al.*, 2006; Chun *et al.*, 2006; Quaresma *et al.*, 2008).

Quando se consideram as reabilitações de dentes posteriores, muitas vezes o posicionamento dos implantes pode ser dificultado devido à presença de irregularidades ósseas decorrentes de reabsorção alveolar ou discrepância esquelética (Canay *et al.*, 1996). Como alternativa, podem ser utilizados implantes curtos (Fugazzotto, 2008), ou implantes de comprimento maior podem ser inseridos no osso na posição inclinada. Esta inclinação dos implantes faz com que seja necessário o uso de componentes protéticos angulados, para correção de posicionamento nos sentidos vestíbulo-lingual ou mésio-distal, fornecendo condições adequadas para confecção e inserção da prótese. Desta maneira, a possibilidade do uso de componentes angulados permite que os implantes sejam posicionados em regiões mais favoráveis do rebordo alveolar, com relação à quantidade e qualidade óssea (Canay *et al.*, 1996).

Entretanto, a utilização de componentes angulados pode gerar aumento da incidência de cargas oblíquas sobre os implantes e tecido ósseo de suporte (Lin *et al.*, 2008), que pode ser afetado de diferentes maneiras e apresentar diferentes padrões de distribuição de tensões em decorrência de sua densidade (Holmes & Loftus, 1997). A incidência de cargas mastigatórias sobre a reabilitação ocasiona momentos fletores nos implantes de suporte. Quando o implante é posicionado de maneira inclinada, a intensidade destes momentos fletores pode aumentar (Kao *et al.*, 2008), gerando maior tensão nos tecidos de suporte.

A avaliação do comportamento biomecânico de reabilitações implantossuportadas em pacientes é limitada, o que torna interessante o uso de metodologias alternativas que possibilitem a realização desta avaliação. A análise do comportamento biomecânico de reabilitações implantossuportadas está diretamente associada à análise das tensões sobre os implantes, osso (cortical e medular) e componentes protéticos. As metodologias mais comumente utilizadas para a análise de tensões são a fotoelasticidade, a extensometria e os elementos finitos.

Por meio da fotoelasticidade, é possível observar o padrão de distribuição de tensão em um material com propriedades fotoelásticas, que simula os tecidos de suporte. O padrão de distribuição das tensões no material fotoelástico é determinado pela presença e concentração de franjas isocromáticas na região ao redor dos implantes, que são formadas em função do fluxo de tensões na região avaliada (Cehreli *et al.*, 2004). O número de franjas indica a magnitude da tensão/*stress* e a proximidade entre elas indica sua concentração na região. No entanto, apesar da possibilidade de utilização de implantes e componentes protéticos geometricamente fiéis aos de reabilitações protéticas *in vivo*, a presença da resina para simular a estrutura óssea de suporte promove diferenças com relação à situação clínica real. A resina fotoelástica possui propriedade de isotropia (Caputo, 1993), na qual um material apresenta as mesmas propriedades em todas as direções (Geng *et al.*, 2001), o que não ocorre nos tecidos ósseos. Além disso, os modelos fotoelásticos apresentam estruturas de suporte homogêneas e padrões de contato perfeito entre a superfície do implante e a resina fotoelástica, situações que também diferem da realidade clínica.

A técnica da extensometria por sua vez é uma maneira eficiente de quantificar tensões (Abduo *et al.*, 2010). A análise extensométrica é realizada por meio da utilização de resistores elétricos, extensômetros (*strain gauges*), associados a equipamentos analisadores que fornecem informações de carregamento estático ou dinâmico, podendo ser utilizada *in vivo* ou *in vitro* (Akca *et al.*, 2002; Assuncao *et al.*, 2009). Por meio da extensometria, podem-se obter informações quantitativas a respeito da situação avaliada. Entretanto, os *strain gauges* aferem deformações em apenas uma direção (Clelland *et al.*, 1993), não se sabendo ao certo a influência do tamanho do extensômetro nos resultados apresentados pelo teste (Tanino *et al.*, 2007), desta maneira a detecção de tensões poderá ser aleatória e altamente dependente do local onde o extensômetro é fixado (Sahin *et al.*, 2002; Karl *et al.*, 2004)

Já no caso da análise utilizando o método dos elementos finitos, é possível avaliar a dinâmica das tensões por meio da simulação de um modelo numérico, desenvolvido com auxílio de computador, que permite a visualização da distribuição das tensões e seus valores em todas as superfícies e estruturas do modelo virtual, sendo amplamente aplicada em estudos na área de Odontologia (Tanino *et al.*, 2007). As simulações por meio do método dos elementos finitos permitem predizer a distribuição de tensão na região de contato do implante com a cortical óssea e em torno do ápice no osso trabecular (Sutpideler *et al.*, 2004), além de permitir prever problemas na conexão prótese-implante e falhas no parafuso de retenção e demais componentes protéticos (Kano *et al.*, 2006).

A semelhança entre os modelos de elementos finitos e as situações clínicas reais são atrativas devido à possibilidade de simulação de estruturas de osso cortical e medular, bem como as geometrias dos implantes e componentes protéticos, e a aplicação de carga dinâmica. Entretanto, para a confecção de modelos de elementos finitos, algumas pressuposições devem ser realizadas e podem influenciar os resultados obtidos após a análise. Algumas destas pressuposições são referentes ao detalhamento da geometria óssea e do implante, a serem modeladas, às propriedades dos materiais, condições de contorno (Korioth & Versluis, 1997) e interface entre osso e implante (Van Oosterwyck *et al.*, 1998).

Desta maneira, o presente estudo foi desenvolvido com o objetivo de verificar o efeito das simplificações da geometria de componentes modelados nos resultados apresentados em estudos utilizando o método de elementos finitos e, avaliar por meio do método de elementos finitos, o comportamento biomecânico de reabilitações suportadas por implantes de diferentes conexões, posicionados paralelos entre si ou inclinados, e submetidos a cargas oclusais variadas.

2. *Capítulos*

A apresentação desta tese está baseada na Resolução CCPG/002/06UNICAMP que regulamenta o formato alternativo para dissertações de Mestrado e teses de Doutorado. Três capítulos contendo artigos científicos compõem este estudo, conforme descrito abaixo:

***Capítulo 1* –** 3D modeling of implant-supported prostheses for finite element analyses: effect of thread representation

Artigo nas normas do periódico científico – ***International Journal of Oral and Maxillofacial Implants***

***Capítulo 2* –** Effect on stress of implant–abutment connection under different loading conditions: A 3D-FEA study

Artigo nas normas do periódico científico – ***Journal of Oral Rehabilitation***

***Capítulo 3* –** Rehabilitation with angled inserted dental implants: tridimensional finite element study of the influence of stress distribution of the rehabilitation

Artigo nas normas do periódico científico – ***Clinical Oral Implants Research***

Capítulo 1

3D modeling of implant-supported prostheses for finite element analyses: effect of thread representation

Jessica Mie Ferreira Koyama Takahashi, DDS, MS[a]

Rafael Leonardo Xediek Consani, DDS, MS, PhD[b]

Marcelo Ferraz Mesquita, DDS, MS, PhD[c]

a – PhD Student

b – Associated Professor

c – Chairman and Professor

Piracicaba Dental School – Campinas State University – Piracicaba, São Paulo, Brazil

Limeira Avenue, 901 - 13414-903, Piracicaba-SP, Brazil

Corresponding author:
Jessica Mie Ferreira Koyama Takahashi
Limeira Avenue, 901
13414-903
Piracicaba-SP, Brazil
E-mail:jemfkt@yahoo.com.br
Phone: 55 19 91022215
Fax: 55 19 21065211

3D modeling of implant-supported prostheses for finite element analyses: effect of thread representation

Abstract

Purpose: This study aimed to evaluate the effect on maximum stress values and the stress distribution patterns of threaded interface simplifications by means of a finite element (FE) analysis.

Materials and Methods: Three-dimensional solid models were obtained of the interface between a lower posterior mandibular bone section and an implant, and between an abutment and a prosthetic screw. The implant/bone interface was designed with a triangular profile thread representation and as a non-threaded cylinder. Thread thickness and contact area were evaluated on the models. The abutment/screw joint was modeled as a triangular threaded interface, and as a cylinder, with or without forged structures. Implant/bone models was loaded with 180N oblique force, while the abutment/prosthetic screw models were arbitrarily loaded with 36N axial and 20N lateral.

Results: Changes in absolute values and the distribution patterns of stress were observed on the implant/bone FE models, regardless of thread representation. Maximum stress values on bone ranged from 135.89MPa to 178.12MPa on the threaded implant/bone interface. Artificially high stresses and strains were observed on the threaded models, concentrated on the first implant threads. Unthreaded models presented maximum stresses of 126.42MPa and 158.84MPa. On the abutment/screw joints, von Mises maximum stress values presented minimal changes with model simplification, with values around 404.23MPa and 413.76MPa. Stress distribution was similar on both models, with stress concentrated at the neck of the prosthetic screw.

Conclusions: For the assumed linear analysis with elastic materials, models with lower extreme stress values are considered to be the most realistic. Therefore, a simplified thread profile might be used for both implant/bone and abutment/prosthetic screw modeling, generating similar stress distribution.

Keywords: biomechanics, stress, finite element analysis, dental implants, threads

Introduction

Since their early development, implant-supported rehabilitations have faced several challenges to their success. These challenges have ranged from physiological or anatomical compromise[1] to unfavorable biomechanical outcomes.[2] Therefore, in order to provide the safest and most reliable rehabilitative treatment for any patient, researchers have always looked for treatments or implant/prosthetic alternatives to overcome these difficulties.

Concerning the biomechanical outcomes of implant-supported rehabilitations, many scientific methodologies are used to verify strain/stress development in supporting bone and prosthetic components. Experimental stress analyses, such as fotoelasticity[3] and extensometry (using strain gauges)[4] or numerical simulation (using the finite element method),[4,5] are techniques that allow researchers to study the biomechanical outcomes of different rehabilitation designs and loading conditions.

The finite element method has been suggested for stress analysis involving complex geometries. This method is based on finding a solution to a mechanical problem by dividing the complex geometry domain into several smaller domains, called elements, and finding a solution to each element so that, when all solutions are combined properly, they produce a solution for the entire body. Despite the possibility of simulating the cortical and trabecular bones, implant and prosthetic components' geometry, and different loading conditions, the finite element method application requires some special care regarding bones' mechanical properties and implant/bone or cortical/trabecular bone interface.[5]

In addition, many studies have presented difficulties in modeling the actual implant interface with the correct representation of implant threads. Implant threads have been presented as completely absent[6,7] or in a simplified model, such as in concentric rings[8] instead of the helix pattern, due to difficulties in modeling the threads' design or due to computational limitations.[9] Nonetheless, there is little information regarding the effect of these modeling simplifications on the stress patterns of implant-supported analyses.

Thus, the aim of the present study was to evaluate the effect of implant thread and prosthetic screw thread simplification on the tridimensional finite element analysis of stress development patterns in implant-supported rehabilitations. A complete assembly was used to evaluate the feasibility of managing large models with the available computer resources.

Materials and Methods

Three-dimensional, solid elements-based finite element (FE) models were built, reproducing an assembly with a dental implant, an intermediary abutment, and a prosthetic screw for lower posterior rehabilitation with the aim of evaluating the stress distribution in the joints between the implant and the supporting bone, and between the abutment and the prosthetic screw. The assemblies consisted of a 4.1mm X 11mm external hexagon implant, a conical abutment and a prosthetic screw (Neodent – Curitiba, PR, Brazil). The mandibular section was 21.5mm high X 8.9mm wide X 9mm long with 1.5mm thick cortical bone layer.

For the analyses of the implant/bone interface, two main 3D FE models were constructed, one with a triangular profile thread representation and the other as a cylinder with no threads. From the initial threaded FE model, five variations were obtained, based on thread thickness and bone/implant contact area (Figure 1). Threaded model-1 (TM-1) presented a triangular thread profile of both implant and bone with cortical bone contact up to the top portion of the uppermost thread until the implant neck; TM-2 presented the same implant profile with reduced bone contact between the implant threads; TM-3 presented the same profile as TM-2 without cortical bone contact on the implant neck; TM-4 presented increased contact area between bone/implant with a changed profile between the implant threads; TM-5 presented similar implant profile as TM-3, with reduced bone contact between the implant threads.

From the initial unthreaded FE model, two variations were obtained (Figure 2). Unthreaded model-1 (UM-1) presented cortical bone contact up to the implant neck, while UM-2 did not.

The abutment/prosthetic screw (AS) interface simplifications were evaluated in three 3D FE models (Figure 3). The first model (AS-1) presented a triangular threaded design of the screwed interface; AS-2 presented absent threads, and AS-3 presented a forged interface of the prosthetic components.

The FE models were obtained by automatically meshing a SolidWorks Assembly (Dassault Systèmes, SolidWorks Corporation, Concord, MA, USA) using the CosmosWorks FEM feature. Three-dimensional Parabolic Tetrahedral elements were used to generate compatible meshes between the models' parts. Each model was generated by setting the same element size of 0.35mm, giving an equivalent total number of elements and nodes. Boundary conditions for implant/bone models (TM and UM) were set on both sides of the bone section, while for the abutment/prosthetic screw models (AS), they were set around the abutment neck. For the implant/bone models, a total of 180N[10] load was applied, consisting of two components: a 161N downward axial force combined with an 80.5N lateral force. For the abutment/prosthetic screw models, the loading conditions were arbitrarily set on 36N axial and 20N lateral.

All materials were considered to be isotropic, homogeneous, and linearly elastic. The relevant materials' mechanical properties (Poisson ratio and Young Modulus) are presented in Table 1.[5,11,12] As mentioned above, all CAD and CAE activities for constructing the solid geometry representation of the parts, the assembly, the further finite element analysis, and the post-processing were performed using the SolidWorks 3D CAD Design Software (Dassault Systèmes, SolidWorks Corporation, Concord, MA, USA). Maximum von Mises stress values were determined, as well as the stress distribution pattern for each model for the purpose of mechanical performance evaluation.

Results

The 3D stress-state effect is rated according to the associated von Mises stress, which provided the maximum stress values and stress distribution pattern for each model that was analyzed. The data obtained from the FE analyses are presented in colored diagrams, allowing visualization of the stress distribution characteristics.

Changes in absolute values and the distribution pattern of stress were observed on the implant/bone FE models, regardless of thread representation. The FE model with triangular profile thread representation and complete bone/implant contact (TM-1) presented higher maximum von Mises stress (178.12MPa), concentrated on the extremity of the first bone threads. The reduction of bone/implant contact (TM-2) generated decreased stress (154.72MPa), which also decreased after removal of cortical bone around the implant neck (TM-3; 135.89MPa) and was concentrated on the bone surface. In the presence of increased implant/bone contact area, with a changed profile between the implant threads (TM-4), the stress concentration was along the first threads, and the maximum von Mises stress obtained was 164.84MPa, which decreased after a reduction of implant/bone contact by changing the bone thread profile (TM-5; 142.00MPa) and presented the stress concentration on the bone surface (Figure 4). Among the non-threaded implant FE models, a difference was also observed with the cortical bone positioning over the implant (Figure 5). The model with cortical bone around the implant neck (UM-1) presented a maximum von Mises stress of 158.84MPa, while the model without cortical bone around the implant neck (UM-2) presented a maximum value of 126.42MPa. Nonetheless, stress distribution was similar for both models.

On the abutment/prosthetic screw FE models (Figure 6), von Mises maximum stress values presented minimal changes in the presence of model simplification. When the triangular thread profile of the screwed interface was modeled (AS-1), maximum stress was 412.55MPa. In the absence of the threads, von Mises maximum stress decreased to 404.23MPa, whereas when the abutment and prosthetic screw were forged to each other, maximum stress was 413.76MPa. Stress distribution was similar for both models, with stress concentrated at the neck of the prosthetic screw.

Discussion

Finite element studies have often presented simplified implant thread configurations, presenting the threads as completely absent[6] or modeled as concentric rings.[8] Nonetheless, there is little information regarding the effect of these modeling simplifications on the stress patterns of implant-supported analyses. Thus, the present study was developed to verify the effect of thread simplification on FE models of implant-supported rehabilitations. For this purpose, and knowing that screwed joints are present on both implant/bone and abutment/prosthetic screw interfaces, separate analyses were made for each screwed joint.

Previous studies[12-15] have been performed to evaluate the effect of implant thread configuration on stress distribution on the supporting alveolar bone. Some studies have found that different thread configurations do not affect the von Mises distribution at the supporting bone[15] or only at the cortical bone structure.[12] In contrast, it has been suggested that thread design might affect the stress distribution and intensity.[13,14] Nonetheless, the main goal of the present study was not to compare implant thread designs directly. Due to the great difference in rigidity between the alveolar bone and the dental implant, this study aimed to discover a modeling technique for a dental implant/alveolar bone interface that would better indicate the stress distribution pattern and would present reliable von Mises maximum stress values. However, because of the assumptions that need to be made in order to perform a finite element analysis, it has been suggested that the data obtained by means of FE analyses be evaluated qualitatively rather than quantitatively.[16]

The FE model representing the implants with a triangular threaded profile (TM-1) presented the highest maximum stress values, which were mainly concentrated in the extremity of the first threads. This stress distribution pattern was observed for most threaded models, regardless of the simplifications that were made. It was noticed that when the bone threads were altered (TM-2 and TM-3), reducing the contacting interfaces between bone and implant threads, the stress distribution pattern changed along with a reduction on maximum stress values. However, it might have been expected that with a reduction in the contact area between the surfaces, the maximum stress values would increase, which was not observed. When the interface between implant and bone threads was reestablished with the changed thread profile (TM-4), the maximum stress increased, even though the contact surface area had increased. Nonetheless, when the profile was altered to an attenuated thread, maximum stress decreased and became more concentrated on the cortical bones' surface.

Despite the fact that the purpose of the present study was not to compare different implant thread designs directly, the results obtained corroborate with previous studies that evaluated the effect of thread design on bone tissue.[12,14] It has been demonstrated that different implant and thread designs might affect stress values, and that truncated V-threaded (with a profile similar to TM-3), 0.36mm width square-threaded,[12] and threads with 0.34-0.50mm height and 0.18-0.30mm width[14] might be favorable for stress distribution pattern.

The obtained maximum stress values on the interface between implant and alveolar bone were demonstrated to be highly dependent on the geometric considerations of the joint,[13,14] probably due to the great difference between the rigidity properties (Young modulus) of these structures. In addition, when FE analyses of biological systems are performed, some artificially high stresses and strains might be portrayed due to kinematic constraints, point loads, or sharp corners.[17] The appearance of these artificially high stresses results from the theory of elasticity, to which the FE solutions are approximated. According to the theory of elasticity, stress and strain singularities are indicative that some modeling idealizations are physically impossible.[17] Saint Venant's principle states that these highly concentrated modeling artifacts do not affect stress values in other regions of the model.[18] Nonetheless, the presence of these artificially high stress values makes it difficult to extract accurate maximum von Mises stress values from the FE analysis.[17]

Considering that this was a linear analysis with elastic materials and forged structures, one might expect the materials to settle and minimize stress concentration, which would make the stress distribution pattern with the lower extreme maximum stress value the most realistic (TM-3 model). Therefore, when the maximum stress values obtained for the lower stress threaded FE model (TM-3) and the equivalent non-threaded FE model (UM-2) are compared, the absolute values difference is around 7%. Besides, both TM-3 and UM-2 models presented similar stress distribution patterns, which agrees with a previously reported study that evaluated the stress distribution patterns of threaded and non-threaded dental implants as geometric simplification for finite element analysis.[19]

Regarding the evaluation of the simplification of the interface between the abutment and the prosthetic screw, it may be noticed that maximum stress values ranged from 404.23-413.76Mpa, with a variation of only 2% between FE models. Along with the minimal variation on maximum stress values, all models presented very similar stress distribution patterns. The obtained results of the geometrically simplified models suggest that either one of these geometries might be suitable for the analysis of implant-supported prosthetic assemblies when the overall mechanical behavior of the parts is the main goal.

The present study was developed in the search for assembly simplifications due to difficulties in obtaining compatible mesh within the representation of complex geometries and limited computational resources (software and hardware). As an example, to accomplish the TM-4 modeling, the required computer resources increased just because a trapezoidal male-female contact area was included. The simulated models aimed to discover a model configuration in which there is a balance between the stress distribution pattern and the cross-sectional contacting interfaces of the evaluated joint structures.

Conclusion

Within the limitations of this FE study, it can be concluded that simplified thread profiles might be used for implant/bone and abutment/prosthetic screw modeling, generating similar stress distribution patterns. In addition, simplifying the most complex geometrical feature of the FE model allows a compatible mesh of the entire assembly to be obtained with fewer computer resources.

Acknowledgements

The authors would like to thank CAPES (Coordenação de Aperfeiçoamento de Pessoal de Nível Superior) and Neodent (Curitiba-PR, Brazil) for support in the development of this research.

References

1. Sethi A, Kaus T, Sochor P. The use of angulated abutments in implant dentistry: five-year clinical results of an ongoing prospective study. Int J Oral Maxillofac Implants 2000;15:801-810.

2. Sahin S, Cehreli MC, Yalcin E. The influence of functional forces on the biomechanics of implant-supported prostheses--a review. J Dent 2002;30:271-282.

3. Turcio KH, Goiato MC, Gennari Filho H, dos Santos DM. Photoelastic analysis of stress distribution in oral rehabilitation. J Craniofac Surg 2009;20:471-474.

4. Akca K, Cehreli MC, Iplikcioglu H. A comparison of three-dimensional finite element stress analysis with in vitro strain gauge measurements on dental implants. Int J Prosthodont 2002;15:115-121.

5. Geng JP, Tan KB, Liu GR. Application of finite element analysis in implant dentistry: a review of the literature. J Prosthet Dent 2001;85:585-598.

6. Kao HC, Gung YW, Chung TF, Hsu ML. The influence of abutment angulation on micromotion level for immediately loaded dental implants: a 3-D finite element analysis. Int J Oral Maxillofac Implants 2008;23:623-630.

7. Kitagawa T, Tanimoto Y, Odaki M, Nemoto K, Aida M. Influence of implant/abutment joint designs on abutment screw loosening in a dental implant system. J Biomed Mater Res B Appl Biomater 2005;75:457-463.

8. Chun HJ, Shin HS, Han CH, Lee SH. Influence of implant abutment type on stress distribution in bone under various loading conditions using finite element analysis. Int J Oral Maxillofac Implants 2006;21:195-202.

9. 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.

10. Mericske-Stern R, Assal P, Mericske E, Burgin W. Occlusal force and oral tactile sensibility measured in partially edentulous patients with ITI implants. Int J Oral Maxillofac Implants 1995;10:345-353.

11. Sakaguchi RL, Borgersen SE. Nonlinear finite element contact analysis of dental implant components. Int J Oral Maxillofac Implants 1993;8:655-661.

12. Geng JP, Ma QS, Xu W, Tan KB, Liu GR. Finite element analysis of four thread-form configurations in a stepped screw implant. J Oral Rehabil 2004;31:233-239.

13. Hansson S, Werke M. The implant thread as a retention element in cortical bone: the effect of thread size and thread profile: a finite element study. J Biomech 2003;36:1247-1258.

14. Kong L, Hu K, Li D, Song Y, Yang J, Wu Z, et al. Evaluation of the cylinder implant thread height and width: a 3-dimensional finite element analysis. Int J Oral Maxillofac Implants 2008;23:65-74.

15. Eraslan O, Inan O. The effect of thread design on stress distribution in a solid screw implant: a 3D finite element analysis. Clin Oral Investig 2010;14:411-416.

16. Iplikcioglu H, Akca 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.

17. Dumont ER, Grosse IR, Slater GJ. Requirements for comparing the performance of finite element models of biological structures. J Theor Biol 2009;256:96-103.

18. Horgan CO. Recent developments concerning Saint-Venant's principle: An update. Appl Mech Rev 1989;42:295-303.

19. Assunção WG, Gomes EA, Barão VA, de Sousa EA. Stress analysis in simulation models with or without implant threads representation. Int J Oral Maxillofac Implants 2009;24:1040-1044.

Table 1 - Materials' properties.

Structure	Material	Poisson ratio	Young modulus (GPa)
Bone	Cortical bone	0.3	13.7
	Trabecular bone	0.3	1.37
Abutment/Prosthetic screw	Ti-6Al-4V	0.31	110
Dental implant	cp Ti	0.3	117

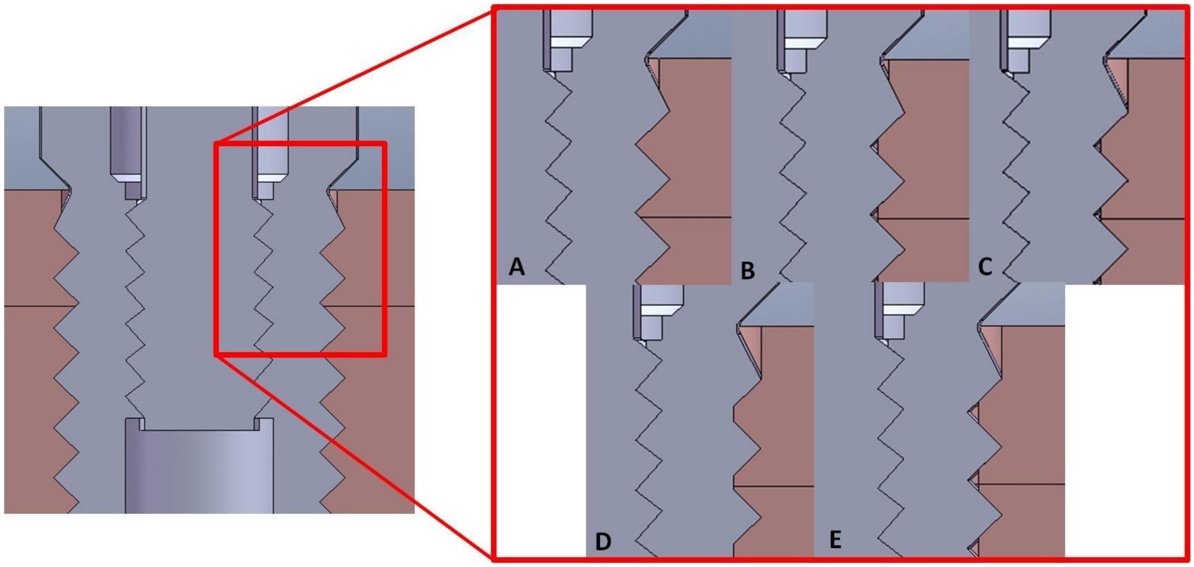

Figure 1 – Bone/implant (pink/grey) interface - Implant threaded models - TM-1 (a); TM-2 (b); TM-3 (c); TM-4 (d); TM-5 (e).

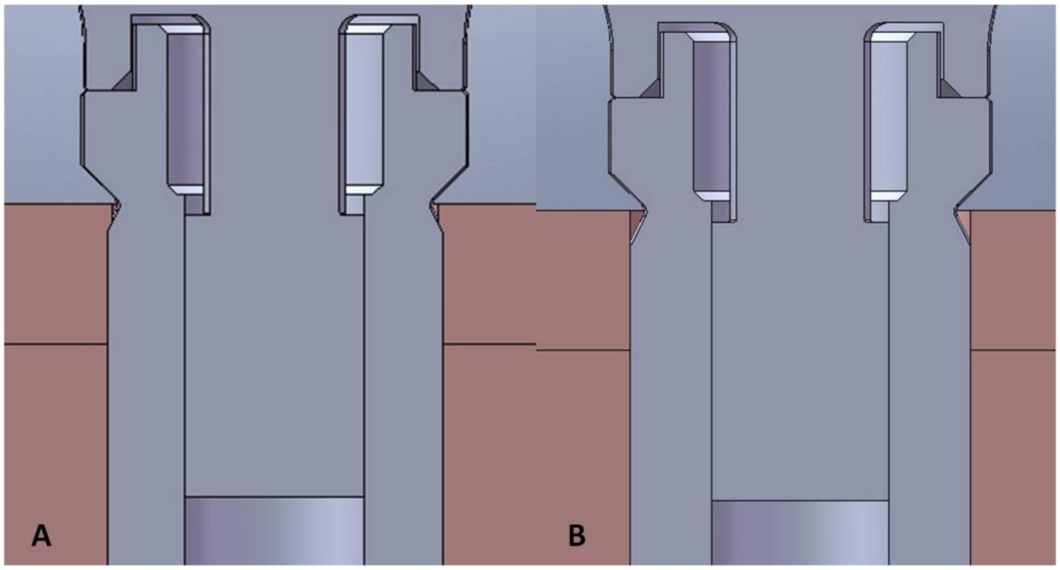

Figure 2 – Bone/implant (pink/grey) interface - Implant unthreaded models - UM-1 (a); UM-2 (b).

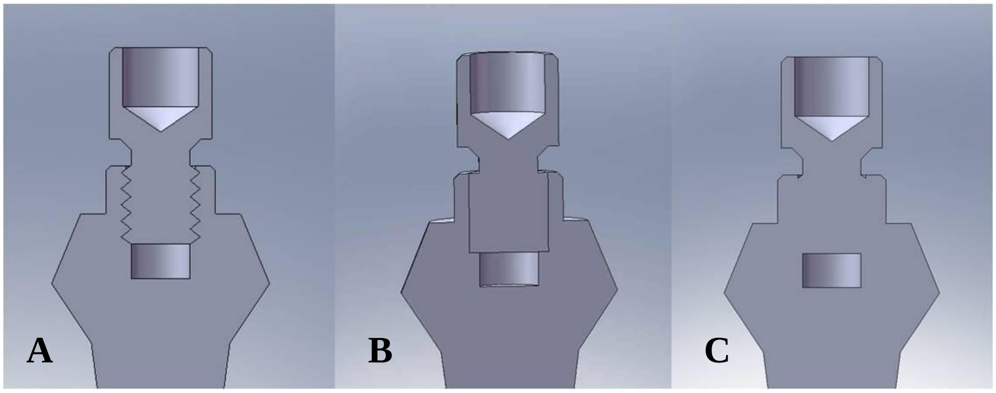

Figure 3 – Abutment/prosthetic screw models – AS-1 (a); AS-2 (b); AS-3 (c).

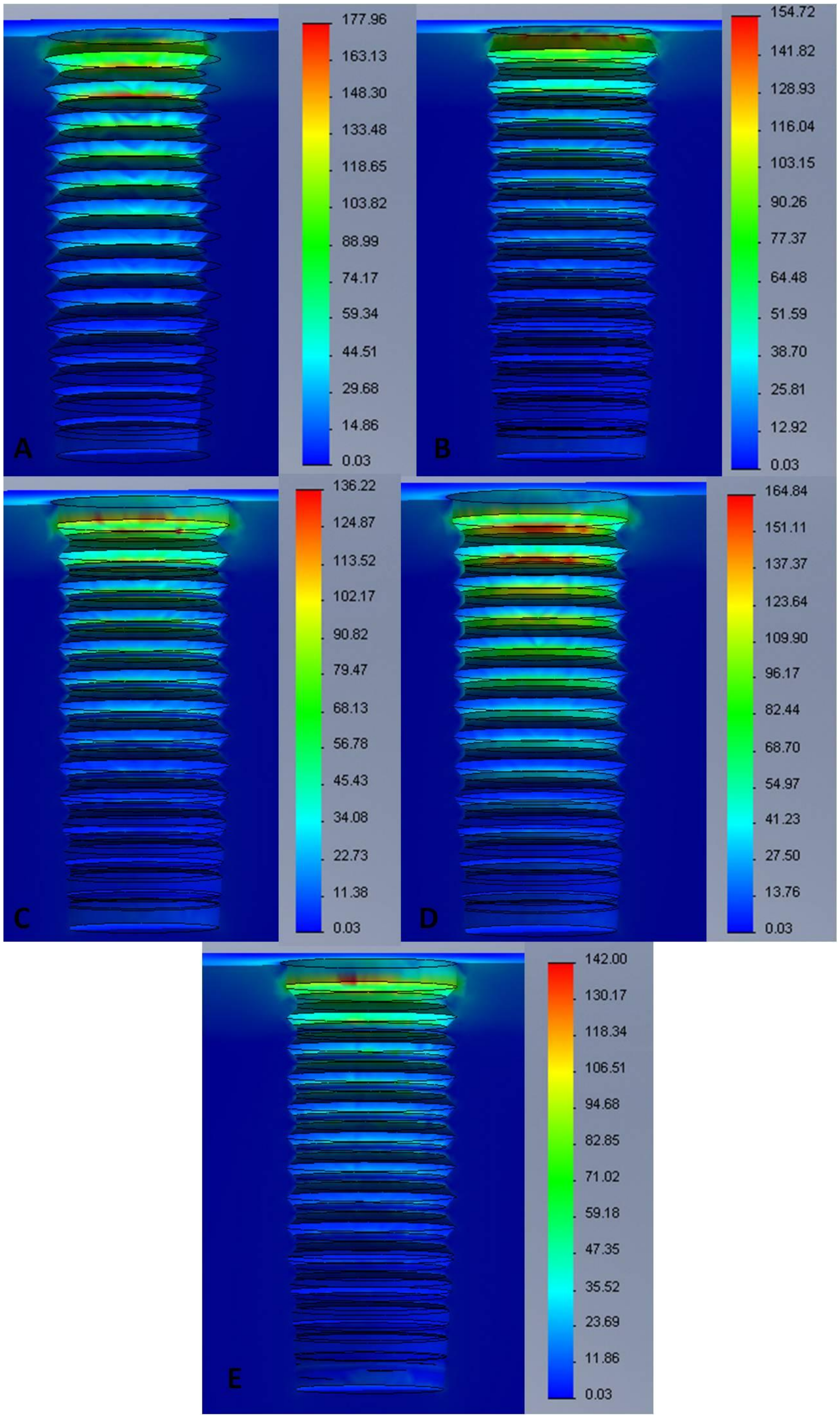

Figure 4 – Bone – Implant threaded models – von Mises stress (MPa) - TM-1 (a); TM-2 (b); TM-3 (c); TM-4 (d); TM-5 (e).

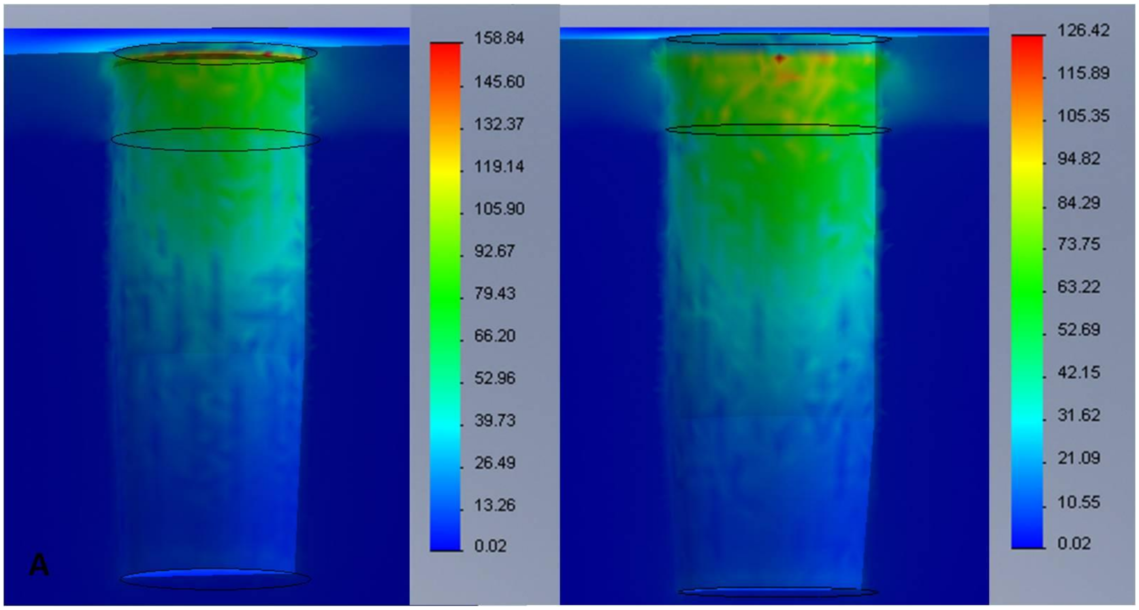

Figure 5 – Bone – Implant unthreaded models – von Mises stress (MPa) - UM-1 (a); UM-2 (b).

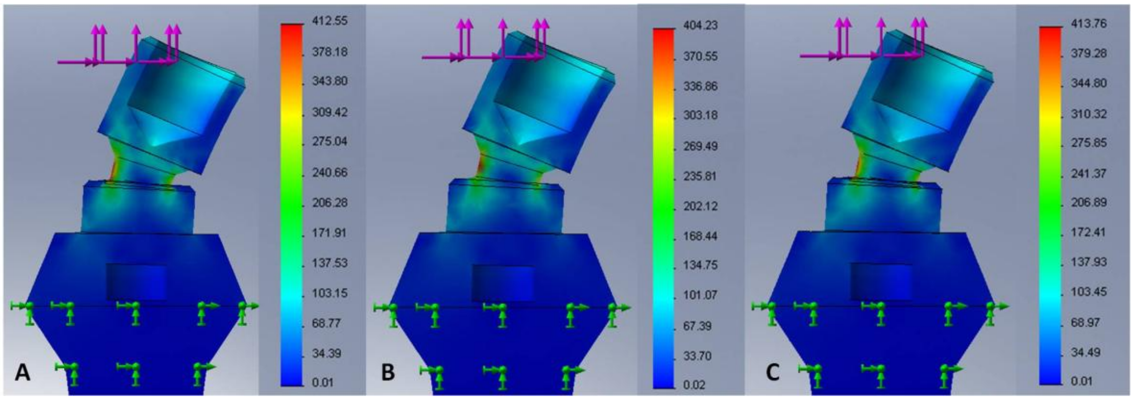

Figure 6 - Abutment/prosthetic screw models – von Mises stress (MPa) - AS-1 (a); AS-2 (b); AS-3 (c).

Capítulo 2

Effect on stress of implant–abutment connection under different loading conditions: A 3D-FEA study

Stress distribution of implant-abutment interfaces

Jessica Mie Ferreira Koyama Takahashi
Rafael Leonardo Xediek Consani
Mauro Antônio de Arruda Nóbilo
Marcelo Ferraz Mesquita

Piracicaba Dental School – Campinas State University – Piracicaba, São Paulo, Brazil

Corresponding author:
Jessica Mie Ferreira Koyama Takahashi
Limeira Avenue, 901
13414-903
Piracicaba-SP, Brazil
E-mail:jemfkt@yahoo.com.br
Phone: 55 19 91022215
Fax: 55 19 21065211

Submission date: 01/06/2011

Effect on stress of implant–abutment connection under different loading conditions: A 3D-FEA study

Stress distribution of implant-abutment interfaces

Abstract

This study aimed to evaluate, by means of three dimensional finite element analyses, stress values and distribution pattern of axial and oblique occlusal loading of three-element fixed implant-supported prostheses, manufactured with different implant connection systems. External-hexagon, internal-hexagon and morse-taper connection systems were chosen to obtain posterior lower partial denture models, with conical abutments and screwed titanium frameworks. For modeling purposes, all materials were considered to be homogeneous, isotropic and linearly elastic. Implant was considered to be completely osseointegrated and prosthetic structures were forged together. Both oblique and axial loads were standardized at 180 N for first premolar and molar, and 280 N for second premolar. Under exclusively axial loading, similar values and stress distribution patterns were observed for all connection systems. However, under oblique loading, stress values presented significant increase and stress distribution pattern changed for some of the internal-connection models' structures. Despite the absence of information regarding the values of stress that actually induce bone resorption and remodeling, it can be concluded that oblique loads generate increased stress on bone structures and also on prosthetic components. Based on the results presented in this study, it can be suggested that internal connection implant systems may present more favorable stress distribution pattern than external-connection system implants.

Key words: finite element analysis, dental implants, occlusal loading, connection system

Introduction

Overall, implant supported rehabilitations may fail under two situations. The first, immediately after the implant has been placed, mostly due to tissue inflammation. The second, due to bone resorption around the dental implant (1). Therefore, biomechanical factors must be widely considered during planning of implant supported rehabilitations due to its main responsibility in bone resorption and fracture of prosthetic components and frameworks (2). Occlusal load transfer between bone/implant interface depends not only of the occlusal load, but also of the material in which the prosthesis is manufactured, the nature of the bone/implant interface, bone quantity and quality and the geometry of the implant that is selected for the rehabilitation (length, diameter and shape) (3, 4).

Researches have reported that when it comes to implant design, the implant thread profile (5, 6) and the implants' shape (7) might influence the stress distribution of occlusal load over the supporting bone tissue. The implant-abutment connection system has also been reported as potentially affecting stress distribution on bone and prosthetic components (8-10).

Several abutment connection systems have been idealized, each one presenting its pros and cons for patient rehabilitation. The conventional external-hexagon connection (butt-joint) has been reported advantageous for its anti-rotational mechanism, easiness on prosthesis unscrewing and compatibility between different implant systems. Nonetheless, external-hex connections might be considered slightly unstable due to the height of the hexagon, allowing prosthesis micromovement and rotation. Inasmuch, internal-hexagon connection systems are advantageous for producing antirotational, stable and more resistant rehabilitations with better force distribution. However, in internal-hex rehabilitations adjustment of divergent implant angles might be difficult. Still within the internal connection types, the taper joint system with a conical seal or Morse taper presents some of the advantages of the internal-hex connection added to better sealing of the joints (9). Taper joint connections are more stable and resistant than the other connections and are also more difficult to be released upon need.

Due to the wide variation of implant and prosthetic systems to be chosen, implant-supported rehabilitations must be carefully planned, so that optimal biomechanical scenario is obtained. Nevertheless, clinical trials are extremely difficult to be performed when it aims to evaluate stress distribution on implant-supported prostheses. Thus, finite element (FE) studies have been performed (3-8, 10), simulating possible clinical conditions and aiming to verify stress distribution on bone structures and prosthetic components. Nonetheless, few FE studies have been performed that have considered both axial and lateral components of occlusal loading.

It is known that vertical loads from masticatory movements induce both axial forces and bending moments that will affect the stress in the implant and bone structures (3). Excessive occlusal loading might be responsible for bone resorption and osseointegration loss, compromising the longevity of the implant-supported rehabilitation. The stress generated due to the occlusal loads is directly related to the type and distribution of these loads, associated with the mechanical properties and design of the implant assembly and the prosthetic framework (11).

Thus, the aim of the present study was to simulate with the finite element method a clinical situation of a three-element fixed partial implant-supported prosthesis, and evaluate the stress distribution pattern of different implant-abutment connection systems (external-hexagon, internal-hexagon or morse taper) loaded with only axial or a combination of both axial and lateral forces.

Materials and Methods

Three-dimensional solid elements based finite element (FE) models were built reproducing the clinical situation of a lower implant-supported rehabilitation. A three-element fixed partial denture framework was designed to restore first and second premolars and first molar, and was supported by two dental implants. Three assemblies were modeled, consisting in two 4.1 mm X 11 mm implant, two conical abutments, two prosthetic screws (Neodent – Curitiba, PR, Brazil) and a framework (25 mm length X 8 mm height). Implant-abutment connection system varied on each assembly, one assembly was modeled with external-hex connection implants and abutments (EH), the other with internal-hex connection (IH) and the last with morse taper connection (MT). A posterior section of the mandible was modeled, with 1.5 mm thick cortical bone layer.

All assemblies were obtained using SolidWorks Assembly (Dassault Systèmes, SolidWorks Corporation, Concord, MA, USA). For the finite element models, the CosmosWorks FEM feature of the same computational program was used to automatically obtain the meshes of the models. Three-dimensional parabolic tetrahedral elements were used to generate compatible meshes between all the models' parts. Each model was generated setting the same element size of 0.35 mm. Boundary conditions for all models were set on both sides of the bone section.

Compressive load was applied over the entire occlusal surface of the prosthetic framework, following two loading conditions. For the exclusively axial loading condition (AX), 180 N were applied over the first premolar and first molar, and 280 N were applied over the second premolar (12). For the oblique loading condition (OB), associated axial and lateral loads were applied, which resulted in a sixty three degree oblique loading. On the first premolar and molar, the 180N load was applied consisting of two components: 161 N downward axial force combined with an 80.5 N lateral force. On the second premolar, the 280 N load was applied with 250.44 N downward axial force combined with 125.22 N lateral force.

All materials were considered to be isotropic, homogeneous and linearly elastic. The materials' mechanical properties (Poisson ratio and Young Modulus) were inserted as follows (3, 5, 13): Implants and prosthetic framework – Commercially pure titanium (0.3; 117 GPa); Abutment and prosthetic screws – Ti-6Al-4V (0.31; 110 GPa); Cortical bone (0.3; 13.7 GPa); Trabecular bone (0.3; 1.37 GPa). All CAD and CAE activities constructing the solid geometry representation of the parts and the assembly and the further finite element analysis and post-processing were performed using the SolidWorks 3D CAD Design Software (Dassault Systèmes, SolidWorks Corporation, Concord, MA, USA).

Maximum von Mises stress values were determined as well as the stress' distribution pattern for each model. Colored diagrams were obtained using FEMAP version 8.3 (Siemens PLM Software, Plano, TX, USA) and are presented for the visualization of the models' stress distribution characteristics. The models are labeled according to the implant-abutment connection system (EH, IH or MT) and the loading condition (AX or OB).

Results

All FE models presented similar number of nodes and elements. The external-hexagon model presented 1 490 816 nodes and 1 073 719 elements, the internal-hexagon model presented 1 492 945 nodes and 1 073 671 elements, and the morse-taper model presented 1 472 904 nodes and 1 062 819 elements. The data obtained from the FE analyses are presented as the associated von Mises stress, providing maximum stress values and stress distribution pattern for each model. Maximum stress values obtained for the evaluated structures are presented on Table 1.

Under axial loading (Table 1 – AX models), the internal-hexagon model (IH-AX) presented higher maximum stress value on the cortical bone structure than the other connections. However, on trabecular bone, IH-AX and EH-AX presented similar maximum stress values, while MT-AX presented lower stress value. Higher stress values were observed around the implant neck on the cortical bone (Figure 3) and at the bottom of the implants on the trabecular bone (Figure 4). The implants cylinders also presented different maximum stress values between the EH implant and the IH implant. Higher stress values were observed at the implants' platform (Figure 5). For the MT model, maximum stress values could only be obtained for the assembled implant and abutment.

On the prosthetic abutments, obtained maximum stress values differed between EH-AX and IH-AX. Maximum stress values at the IH abutment screw was 28.40 MPa. On the abutments, maximum stress was located at the abutment platform, at the interface between the framework and the abutment (Figure 6). Prostheses frameworks presented similar maximum stress values for all models. Higher stress values were located at the bottom of the framework, at the contacting surface with the abutment (Figure 7). Nevertheless, the prosthetic screws presented different maximum stress values between the models. EH-AX model presented the lowest maximum stress, followed by MT-AX and IH-AX. Despite the difference on maximum stress values, stress location was similar for all models, at the prosthetic screws' neck (Figure 8).

Maximum stress values of all evaluated structures presented increase when the models were submitted to the oblique loading conditions (Table 1 – OB models). EH-OB model presented maximum stress on cortical bone similar to MT-OB. On trabecular bone, MT-OB presented the lowest maximum stress value and EH-OB presented the highest. Distribution patterns of maximum stress on cortical bone were similar to those of the axial loading condition (Figure 3). For the trabecular bone, high stress values were observed on the surface of the trabecular bone, close to the implants' neck and at the bottom of the implant (Figure 4). On the implants, maximum stress values were located at the neck of the EH and IH implants (Figure 5). The EH implant presented higher maximum stress value.

The IH prosthetic abutment presented lower maximum stress value. However, stress concentration was similar for all models. Higher stress values were located at the lower portion of the abutment, at the contacting surface with the implant (Figure 6). Maximum stress at the IH abutment screw increased to 157 MPa. Maximum stress values and stress distribution on the frameworks remained similar for all models, with higher stress concentration at the bottom of the framework (Figure 7). The same was observed on the prosthetic screws (Figure 8), with exception of the IH-OB prosthetic screw, which presented reduced maximum stress (Table 1).

Discussion

Stress distribution pattern of implant-supported rehabilitations has often been evaluated by means of strain gauge, fotoelasticity and finite element methods. It has been suggested that some features of the implant-supported rehabilitations might affect the stress magnitude and distribution pattern to the supporting bone (5-7). Implant connection system has also been reported as potentially affecting stress distribution on bone and prosthetic components (8-10). Nevertheless, there is no agreement on the actual effect of this parameter on stress of implant-supported rehabilitations (8, 14).

Three commonly used implant connection systems were evaluated by means of FE modeling in the present study. Finite element models were obtained simulating three-element implant-supported fixed partial rehabilitations, using implants with external-hexagon, internal-hexagon or morse-taper connection system. The models were submitted to two loading conditions, with exclusively axial loading or with the combination of both axial and lateral loading. Loading forces were obtained based on a previous report (12) and applied over the entire occlusal surface of each teeth modeled in the framework to distribute stress more equally to the other modeled structures (4, 15).

Previous studies have diverged on rather the implant connection type affects or not stress values and distribution on implants and prosthetic components. In the present study the evaluated implant connection systems presented slight changes on maximum von Mises stress values. Nonetheless, stress distribution pattern was similar for all models under exclusively axial loading. These obtained results corroborate with previous studies (8, 16) in which under axial loading similar stress distribution patterns and stress values were obtained for EH, IH and MT systems.

However, occluding masticatory forces induce both axial and lateral forces, originating bending moments that will affect the stress in the implant and bone structures (3, 17). It is known that vertical components of occlusal forces are much higher than the oblique and horizontal forces (7), thus, the 2:1 ratio that was determined for the axial and lateral forces used in the present study. From the combination of the loads determined for both loading axis, a resulting force at 63^{o} angulation to the horizontal plane was obtained and applied over the occlusal surface of the frameworks.

Upon oblique loading, maximum von Mises stress increased at all the evaluated structures (18, 19). Stress at cortical bone ranged around four fold the stress at axial loading. Trabecular bone and prosthetic framework presented close to twice higher stress under oblique loading. Implants, abutments and prosthetic screws presented even higher increased stress values. The increased stress when oblique loading is applied was expected as the presence of lateral forces during occlusion generates a bending moment within the prosthesis, prosthetic components and the supporting implants that influences principal stresses and von Mises stress values of the rehabilitation structures (20).

Besides increasing the maximum stress values of the structures, oblique loading promoted change on some of the stress concentration areas. Trabecular bone presented stress concentration areas at both the apex of the implant and close to the implants' platform. In the presence of oblique loading, stress distribution area may be greater mainly at the opposite direction to which the load was applied (21). Higher stress was also concentrated on a lower portion of the implants cylinders and the abutments when compared to the location on the axially loaded models, which occurred probably as a result of the bending moment originated by the oblique loading of the rehabilitation (3, 17).

The different connection systems that were evaluated presented minor differences on stress intensity. However, internal connection systems presented a tendency for lower stress values. Upon oblique loading, morse-taper connection presented lower stress at trabecular bone, while internal-hexagon connection presented lower stress at implants, abutments and prosthetic screw. These findings are in agreement with other studies, on which internal connection systems presented lower stress concentration than external connection systems (16, 20, 22). Internal connection systems' lower stress values might have been a consequence of the greater contact area between the abutments and the implants. Increased contact area between these structures might have reduced the effect of bending caused by the horizontal component of the applied load (8). In addition, internal connection systems present greater stability than external-hex systems (23), which may contribute to stress values and distribution. When comparing both internal connection systems, it was observed that internal-hex model presented higher stress on trabecular bone and lower on prosthetic structures, while morse-taper model presented higher stress on the prosthesis structures and lower on cortical and trabecular bone (10). Nonetheless, both internal connection models presented lower stress on either trabecular bone or prosthetic components.

Excessive load on implant-supported rehabilitations are known to be responsible for bone resorption around the dental implant (1). Cortical bone structure is known to present higher stress concentration than trabecular bone, hence the resorption pattern frequently observed. Ultimate tensile and compressive strength of cortical bone has been reported around 100-121 MPa and 167-173 MPa respectively (24, 25), however, stress values that actually cause biological changes such as resorption and remodeling in the bone are not presently known (26).Therefore, while planning an implant-supported rehabilitation reduced stress values and more equal stress distribution are desired.

Thus, within the limitations of this study and taking in consideration the assumptions made for these FE models, it can be concluded that upon oblique loading, stress values present significant increase. Also, the slight differences between the stress values obtained in this study might suggested that internal connection systems may present more favorable stress pattern than external-hexagon connection system.

Acknowledgements

The authors would like to thank CAPES (Coordenação de Aperfeiçoamento de Pessoal de Nível Superior) and Neodent (Curitiba-PR, Brazil) for the support to the development of this research.

References

1. Sethi A, Kaus T, Sochor P. The use of angulated abutments in implant dentistry: five-year clinical results of an ongoing prospective study. Int J Oral Maxillofac Implants. 2000; 15: 801-810.

2. Sahin S, Cehreli MC, Yalcin E. The influence of functional forces on the biomechanics of implant-supported prostheses--a review. J Dent. 2002; 30: 271-282.

3. Geng JP, Tan KB, Liu GR. Application of finite element analysis in implant dentistry: a review of the literature. J Prosthet Dent. 2001; 85: 585-598.

4. Eskitascioglu G, Usumez A, Sevimay M, Soykan E, Unsal E. The influence of occlusal loading location on stresses transferred to implant-supported prostheses and supporting bone: A three-dimensional finite element study. J Prosthet Dent. 2004; 91: 144-150.

5. Geng JP, Ma QS, Xu W, Tan KB, Liu GR. Finite element analysis of four thread-form configurations in a stepped screw implant. J Oral Rehabil. 2004; 31: 233-239.

6. Kong L, Hu K, Li D, Song Y, Yang J, Wu Z, et al. Evaluation of the cylinder implant thread height and width: a 3-dimensional finite element analysis. Int J Oral Maxillofac Implants. 2008; 23: 65-74.

7. Degerliyurt K, Simsek B, Erkmen E, Eser A. Effects of different fixture geometries on the stress distribution in mandibular peri-implant structures: a 3-dimensional finite element analysis. Oral Surg Oral Med Oral Pathol Oral Radiol Endod. 2010; 110: e1-11.

8. Chun HJ, Shin HS, Han CH, Lee SH. Influence of implant abutment type on stress distribution in bone under various loading conditions using finite element analysis. Int J Oral Maxillofac Implants. 2006; 21: 195-202.

9. Maeda Y, Satoh T, Sogo M. In vitro differences of stress concentrations for internal and external hex implant-abutment connections: a short communication. J Oral Rehabil. 2006; 33: 75-78.

10. Quaresma SE, Cury PR, Sendyk WR, Sendyk C. A finite element analysis of two different dental implants: stress distribution in the prosthesis, abutment, implant, and supporting bone. J Oral Implantol. 2008; 34: 1-6.

11. 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.

12. Mericske-Stern R, Assal P, Mericske E, Burgin W. Occlusal force and oral tactile sensibility measured in partially edentulous patients with ITI implants. Int J Oral Maxillofac Implants. 1995; 10: 345-353.

13. Sakaguchi RL, Borgersen SE. Nonlinear finite element contact analysis of dental implant components. Int J Oral Maxillofac Implants. 1993; 8: 655-661.

14. Akca K, Cehreli MC. A photoelastic and strain-gauge analysis of interface force transmission of internal-cone implants. Int J Periodontics Restorative Dent. 2008; 28: 391-399.

15. Dittmer MP, Kohorst P, Borchers L, Schwestka-Polly R, Stiesch M. Stress analysis of an all-ceramic FDP loaded according to different occlusal concepts. J Oral Rehabil. 2010 (*epub* ahead of print);

16. Bernardes SR, de Araujo CA, Neto AJ, Simamoto Junior P, das Neves FD. Photoelastic analysis of stress patterns from different implant-abutment interfaces. Int J Oral Maxillofac Implants. 2009; 24: 781-789.

17. Cehreli M, Duyck J, De Cooman M, Puers R, Naert I. Implant design and interface force transfer. A photoelastic and strain-gauge analysis. Clin Oral Implants Res. 2004; 15: 249-257.

18. Huang HL, Lin CL, Ko CC, Chang CH, Hsu JT, Huang JS. Stress analysis of implant-supported partial prostheses in anisotropic mandibular bone: in-line versus offset placements of implants. J Oral Rehabil. 2006; 33: 501-508.

19. Lin CL, Wang JC, Ramp LC, Liu PR. Biomechanical response of implant systems placed in the maxillary posterior region under various conditions of angulation, bone density, and loading. Int J Oral Maxillofac Implants. 2008; 23: 57-64.

20. Hansson S. Implant-abutment interface: biomechanical study of flat top versus conical. Clin Implant Dent Relat Res. 2000; 2: 33-41.

21. Falcon-Antenucci RM, Pellizzer EP, de Carvalho PS, Goiato MC, Noritomi PY. Influence of cusp inclination on stress distribution in implant-supported prostheses. A three-dimensional finite element analysis. J Prosthodont. 2010; 19: 381-386.

22. Pessoa RS, Muraru L, Junior EM, Vaz LG, Sloten JV, Duyck J, et al. Influence of implant connection type on the biomechanical environment of immediately placed implants - CT-

based nonlinear, three-dimensional finite element analysis. Clin Implant Dent Relat Res. 2010; 12: 219-234.

23. Kitagawa T, Tanimoto Y, Odaki M, Nemoto K, Aida M. Influence of implant/abutment joint designs on abutment screw loosening in a dental implant system. J Biomed Mater Res B Appl Biomater. 2005; 75: 457-463.

24. Reilly DT, Burstein AH. The elastic and ultimate properties of compact bone tissue. J Biomech. 1975; 8: 393-405.

25. 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: 238-245.

26. Akca K, Iplikcioglu H. Finite element stress analysis of the effect of short implant usage in place of cantilever extensions in mandibular posterior edentulism. J Oral Rehabil. 2002; 29: 350-356.

Table 1 – Maximum von Mises stress values (MPa) presented by the evaluated models.

Model		Cortical bone	Trabecular bone	Implants	Abutments	Prosthetic screw	Framework
EH	**AX**	31.30	7.02	90	114	49.20	110
	OB	123	13.30	605	565	127	260
IH	**AX**	35.30	7.10	83.30	105	71.90	102
	OB	139	12.90	399	416	66.20	264
MT	**AX**	33.20	5.58	132*		67.60	112
	OB	125	9.32	882*		210	279

* On the MT model, implant and abutment were modeled as one piece, therefore, maximum stress values were obtained for this combined piece.

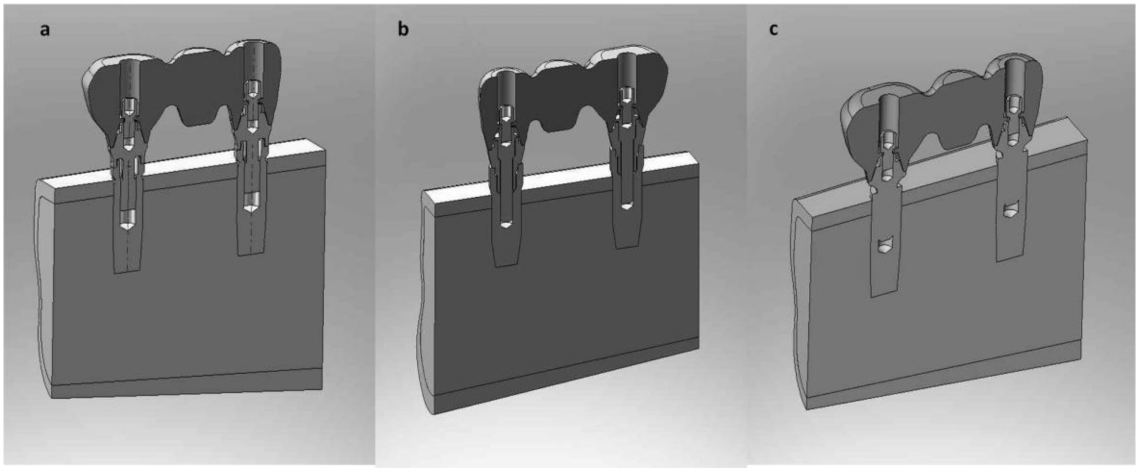

Figure 1 – Simulated three-element implant supported rehabilitations – sectioned view – (a) external-hexagon implants, (b) internal-hexagon implants, (c) morse-taper implants.

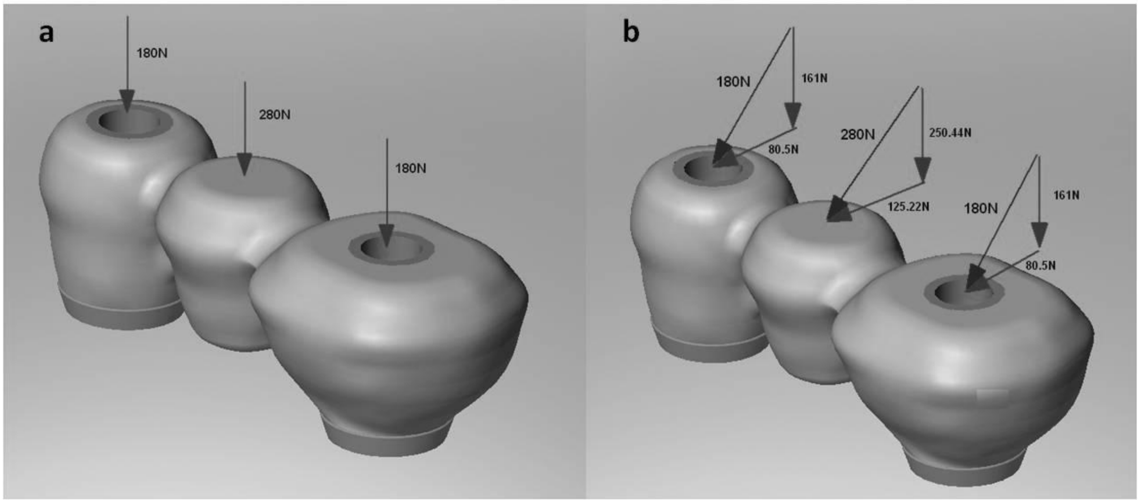

Figure 2 – Occlusal loading schemes. Single arrows only ilustrate loading direction. Occlusal loads were applied over the entire occlusal surface of the frameworks – (a) axial loading, (b) oblique loading.

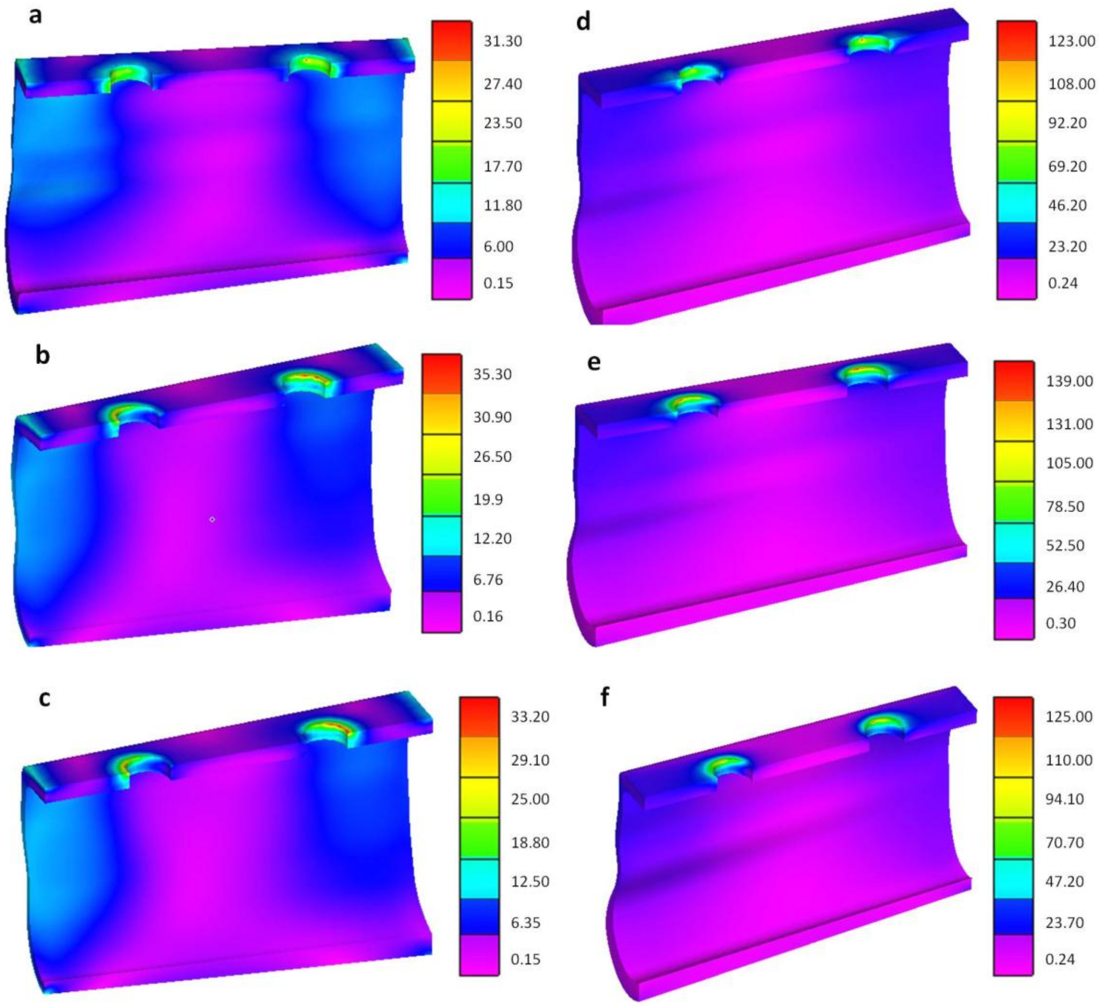

Figure 3 – Cortical bone – von Mises stress (MPa). Maximum stress located at the cortical bone surrounding the implant – (a) axial load, external-hexagon implants, (b) axial load, internal-hexagon implants, (c) axial load, morse-taper implants, (d) oblique load, external-hexagon implants, (e) oblique load, internal-hexagon implants, (f) oblique load, morse-taper implants.

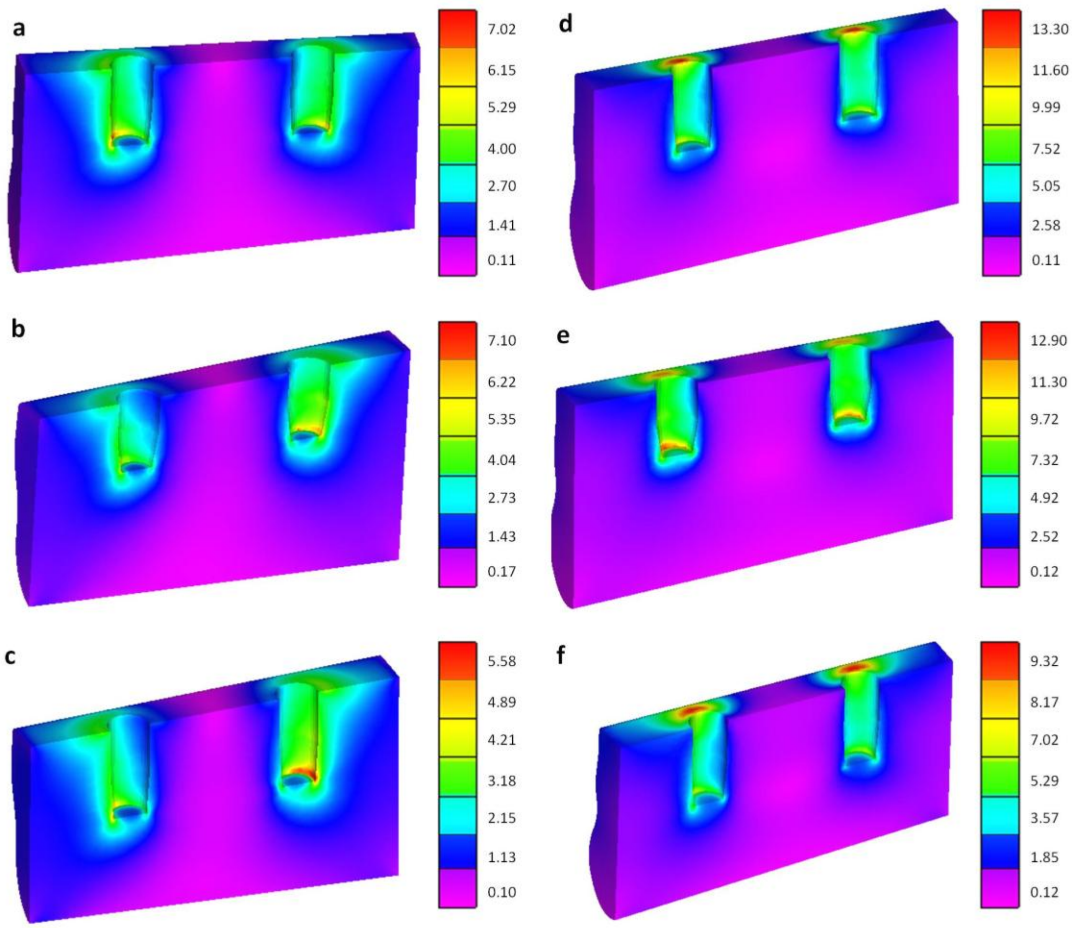

Figure 4 – Trabecular bone – von Mises stress (MPa). On AX models, maximum stress is located at the bottom of the implants, while on the OB models maximum stress also locates at the surface of the trabecular bone – (a) axial load, external-hexagon implants, (b) axial load, internal-hexagon implants, (c) axial load, morse-taper implants, (d) oblique load, external-hexagon implants, (e) oblique load, internal-hexagon implants, (f) oblique load, morse-taper implants.

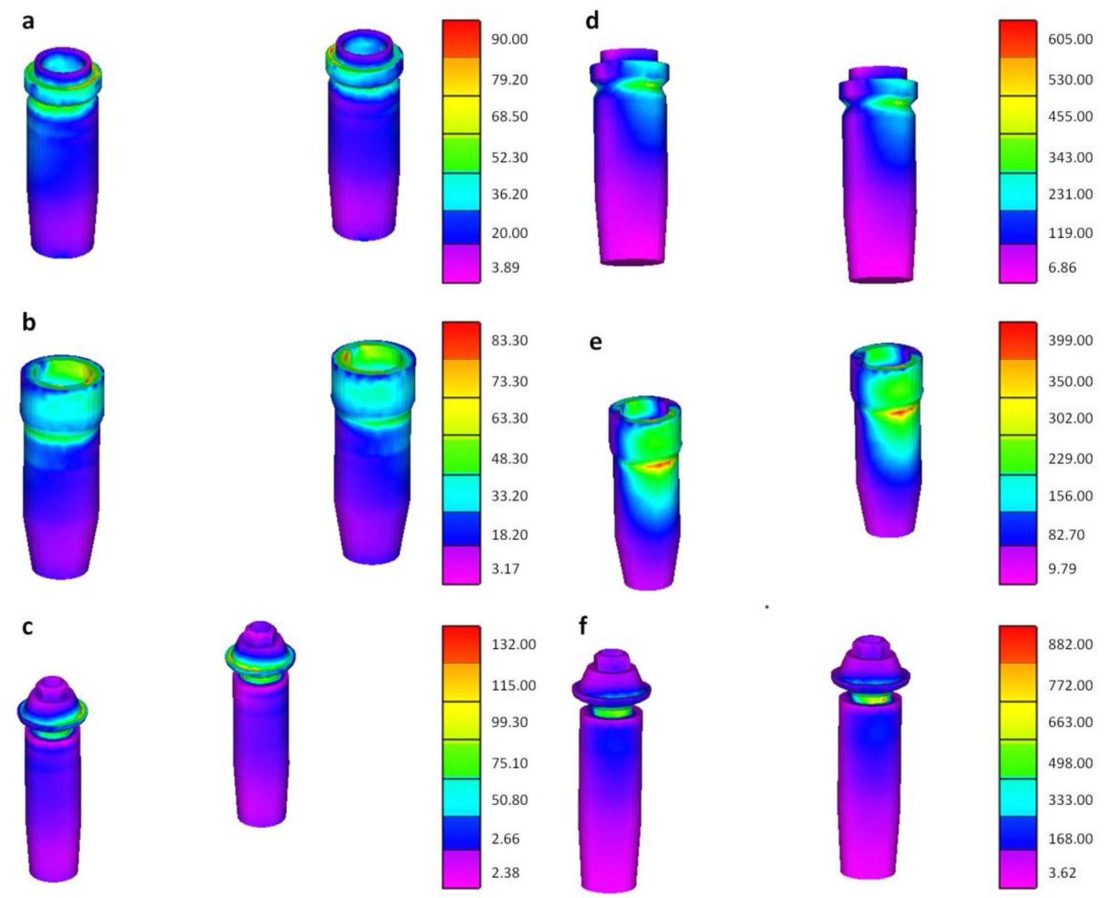

Figure 5 – Implants – von Mises stress (MPa). Maximum stress location changed from the implants' platform (AX models) to the implants' neck (OB model) – (a) axial load, external-hexagon implants, (b) axial load, internal-hexagon implants, (c) axial load, morse-taper implants, (d) oblique load, external-hexagon implants, (e) oblique load, internal-hexagon implants, (f) oblique load, morse-taper implants.

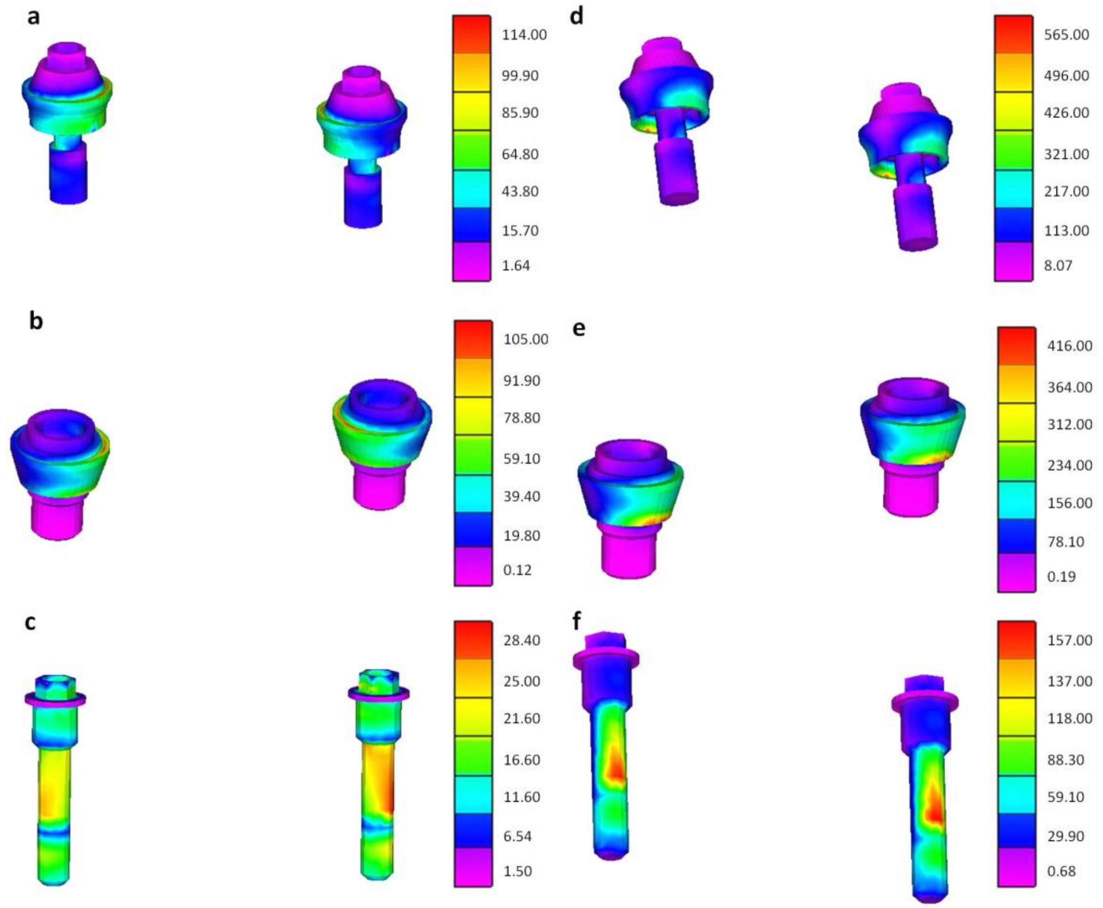

Figure 6 – Abutments – von Mises stress (MPa). Under AX loading, maximum stress on the abutments was located at the contact area between the abutments and the framework. Under oblique loading, maximum stress was located at the contact area between the abutments and the implants – (a) axial load, external-hexagon implants, (b) axial load, internal-hexagon implants, (c) axial load, internal-hexagon abutment screw, (d) oblique load, external-hexagon implants, (e) oblique load, internal-hexagon implants, (f) oblique load, internal-hexagon abutment screw.

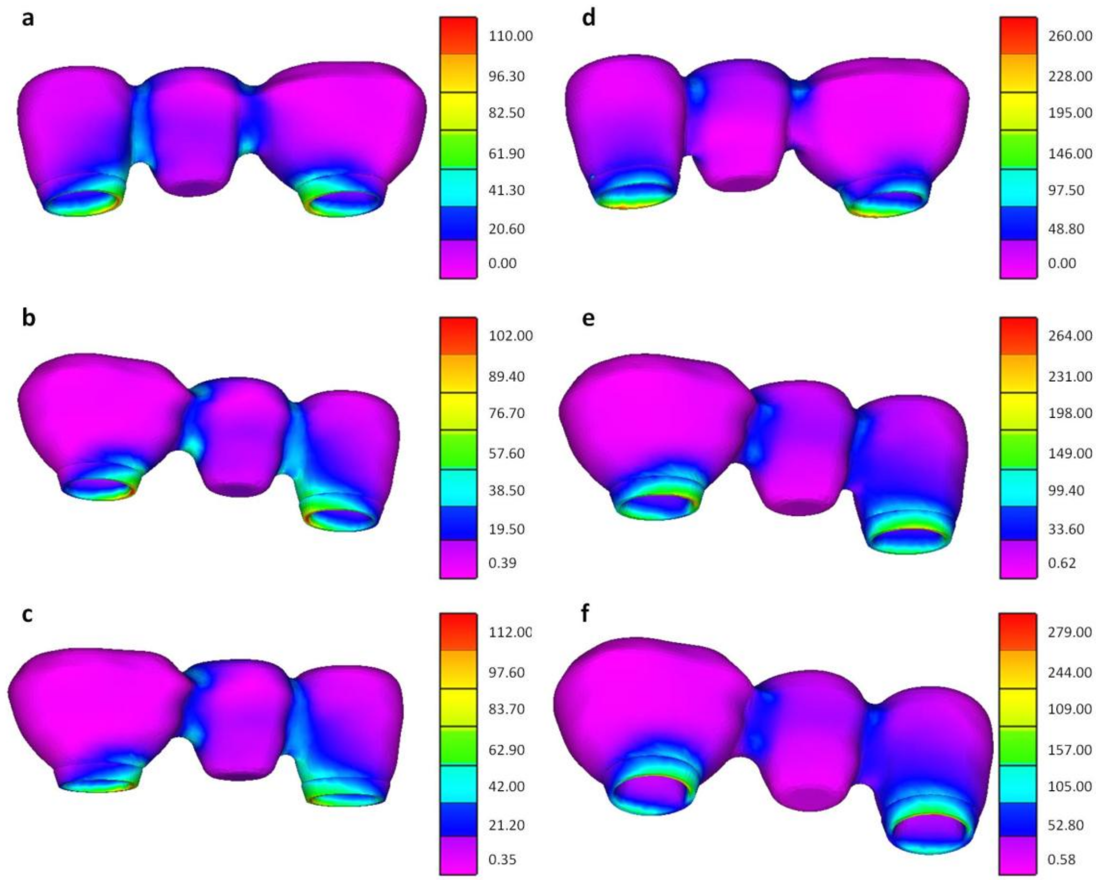

Figure 7 – Frameworks – von Mises stress (MPa). Stress distribution was similar for both loading conditions, with increased maximum stress value upon oblique loading – (a) axial load, external-hexagon implants, (b) axial load, internal-hexagon implants, (c) axial load, morse-taper implants, (d) oblique load, external-hexagon implants, (e) oblique load, internal-hexagon implants, (f) oblique load, morse-taper implants.

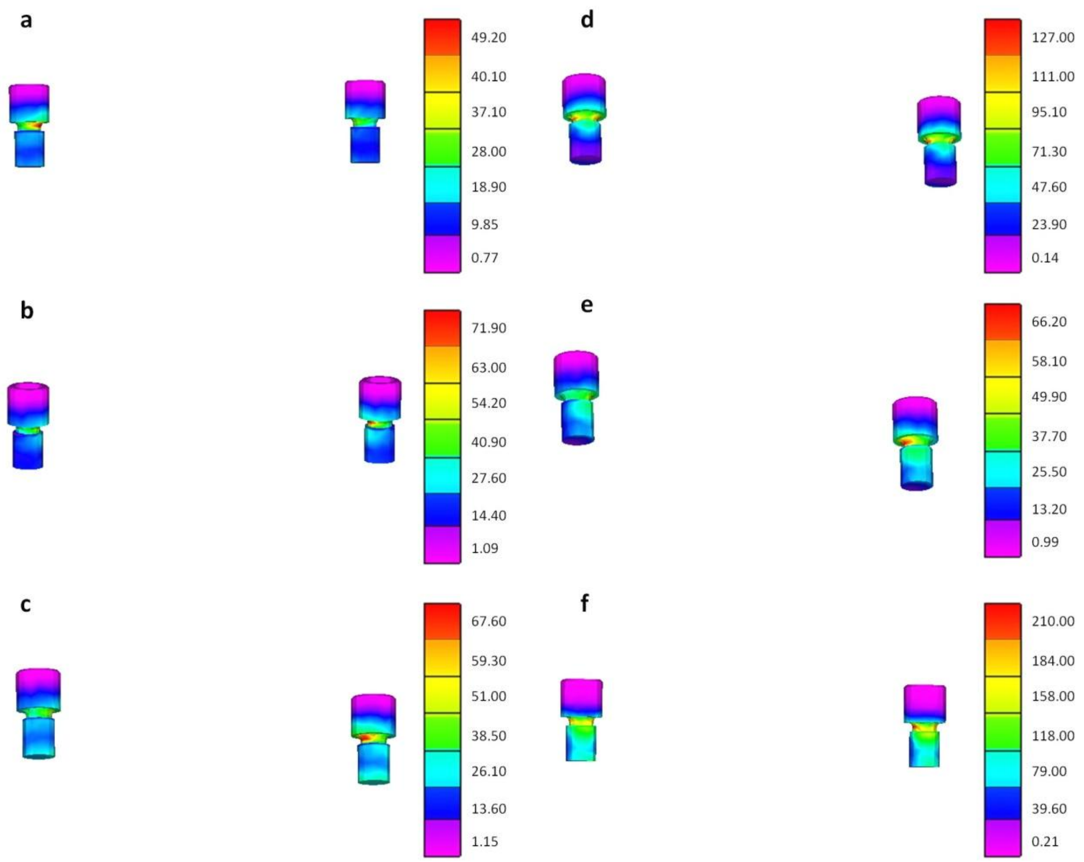

Figure 8 – Prosthetic screws – von Mises stress (MPa). Both loading conditions generated maximum stress around the screws' neck. Oblique loading generated increased maximum stress values – (a) axial load, external-hexagon implants, (b) axial load, internal-hexagon implants, (c) axial load, morse-taper implants, (d) oblique load, external-hexagon implants, (e) oblique load, internal-hexagon implants, (f) oblique load, morse-taper implants.

Capítulo 3

Rehabilitation with angled inserted dental implants: tridimensional finite element study of the influence of stress distribution of the rehabilitation

Stress on rehabilitations with angled implants

Jessica Mie Ferreira Koyama Takahashi

Rafael Leonardo Xediek Consani

Guilherme Elias Pessanha Henriques

Marcelo Ferraz Mesquita

Piracicaba Dental School – Campinas State University

Corresponding author:
Jessica Mie F. K. Takahashi
Limeira Avenue, 901
13414-903
Piracicaba – SP, Brazil
E-mail: jemfkt@yahoo.com.br
Phone: 55 199102-2215
Fax: 55 19 2106-5211

Key-words: dental implants, stress, finite element method, angled abutments, tilted implants

Abstract

Objectives: To evaluate the effect on stress on bone structures and prosthetic components of posterior lower jaw three-element fixed partial dentures supported by inclined distal implants.

Material and Methods: Three-dimensional finite element models were built simulating a three-element fixed partial denture supported by two implants with external-hexagon or morse-taper connection system. Models were built with parallel implants and straight abutments or with mesial tilted distal implant and 17^{o} angled abutment. Screw retained frameworks were loaded with oblique force of 180N on first premolar and first molar and 280N on second premolar. Stress distribution pattern and von Mises maximum stress were obtained for each model and structure.

Results: Stress distribution on cortical bone was similar on all models. Maximum stress values on cortical bone were around 124MPa, presenting little variation. The same pattern was observed on trabecular bone, nonetheless, mesial tilting of the distal implant increased stress concentration on trabecular bone around the straight implant. Morse-taper prosthetic components presented higher maximum stress on abutment and prosthetic screw. Angled abutments presented stress distribution and maximum stress values similar to those presented by the straight abutments. Prosthetic screws of the same connection system model were not affected by the presence of angled abutments.

Conclusion: Both external-hexagon and morse-taper connection systems presented similar stress distribution on the supporting bone, and maximum stress values within the ultimate compressive and tensile strength of bone. Prosthetic components of the morse-taper connection present higher stress than the external-hexagon components.

Introduction

Dental rehabilitation concepts and treatment options are in constant evolution. Every since the osseointegration was first described (Branemark et al. 1977), longitudinal studies have been performed and have demonstrated the reliability and longevity of dental implant treatment, with reported success rates of up to 97.7% (Jemt 2008, Jung et al. 2008). A clear difference between implants' and prostheses' success rates (Jemt 2008), nonetheless, a success rate of 96.6% has been reported for implant-supported fixed partial dentures (FPD) (Halg et al. 2008). As implant success goes, failure may occur in two stages. The first stage being after the implant is placed and the second due to bone loss around the implant after its initial osseointegration (Sethi et al. 2000).

Therefore, biomechanical risk factors must be broadly considered during planning of implant placement and prosthetic rehabilitation for being responsible for both bone resorption and framework and prosthetic components' fracture (Sahin et al. 2002). Biomechanical failures are mainly related to the rehabilitations' overloading. Inasmuch, load transference to the implant-bone interface depends on the intensity and direction of the occlusal load, the material of which the prosthesis is manufactured, the nature of the implant-bone interface, bone quantity and quality and also the geometry of the inserted dental implant (Eskitascioglu et al. 2004, Geng et al. 2001).

When it comes to posterior teeth rehabilitation, implant positioning might be difficult due to the presence of bone irregularities as consequence of bone resorption or skeletal discrepancy (Canay et al. 1996). Less quantity of dense bone is often observed in the posterior region of partially edentulous lower jaws if compared to the anterior interforaminal region of completely edentulous jaws (Akca & Iplikcioglu 2001). In addition, the presence of anatomical landmarks, such as the mandibular canal and the mentual foramen might difficult implant positioning on the jaw and restrict the length of the implant to be placed (Akca & Iplikcioglu 2002). In such situations, short implants' placing (Fugazzotto 2008) or placing of longer implants on inclined position is often performed.

In the presence of inclined positioned dental implants for posterior partial rehabilitations, angled abutments are needed in order to reduce the divergence between the implants' platform and obtain proper insertion axis for the prosthesis. However, the use of angled abutments, and the inclined position of the supporting implants might increase the susceptibility of the rehabilitation system to oblique occlusal loads, mainly over the implants and bone structures (Lin et al. 2008). Posterior teeth rehabilitations are also exposed to more intense loading condition, hence, maximum bite force of patients with FPD supported by dental implants has been demonstrated to be similar of those patients with natural dentition (Mericske-Stern et al. 1995).

Because osseointegrated implants present an ankylosed interface with the surrounding bone, the crestal bone around the implants might act as a fulcrum point for lever action when a force (bending moment) is applied and, therefore, the crestal bone would be more susceptible to this force (Kim et al. 2005). If an implant is placed inclined in the residual bone, this bending moment might be increased upon loading (Kao et al. 2008) and could eventually compromise the osseointegration of the implant and the longevity of the rehabilitation. Nevertheless, in some situations the assessment of the biomechanical behavior of prosthetic rehabilitations *in vivo* is somehow limited, which makes interesting the use of *in vitro* simulations of the rehabilitations.

Several in vitro methods have been reported for the assessment of stress and strain of implant-supported rehabilitations (Assunção et al. 2009). Among these methods, the finite element method (FEM) has been widely used to evaluate problems involving complex geometries, such as the implant-supported rehabilitations by allowing the determination of stress and strain on surfaces of virtual models (Tanino et al. 2007) with the assistance of a computer device. This reliable method is advantageous for it is possible to verify stress distribution pattern in contacting interfaces of implant and surrounding cortical and trabecular bones (Sutpideler et al. 2004), besides providing information regarding prosthesis-implant interface and prosthetic components (Kano et al. 2006).

Thus, the aim of this study was to assess by means of tridimensional finite element method the effect on stress on bone structures and prosthetic components of posterior lower jaw three-element fixed partial dentures supported by inclined distal implants.

Material and Methods

Three-dimensional finite element models were built using solid tetrahedral elements, simulating a three-element fixed partial denture supported by two implants. The FPD framework was modeled to simulate the rehabilitation of three missing teeth: first premolar, second premolar (pontic) and first molar. Four assemblies were modeled, consisting each of two dental implants (4.1mm X 11mm), two conical abutments, two prosthetic screws (Neodent – Curitiba, PR, Brazil) and a prosthetic framework. Models were built using different implant-abutment connection systems. The assemblies were modeled supported by external hexagon (EH) or morse-taper (MT) connection implants.

A posterior section of the mandible was modeled, with 1.5mm thick cortical bone layer. Implants were placed on the mandible section positioned either parallel (P) to each other or with the distal implants' platform mesially inclined (I). For each implant positioning situation, proper abutments were modeled in order to obtain parallel prosthetic platforms for both implants, regardless of implant inclination. Therefore, for the models with parallel implants, conventional straight abutments were modeled; while for the inclined implants' models 17^{o} angled abutment was placed on the inclined implant, so that implant inclination would be corrected for adequate prosthetic rehabilitation.

All CAD and CAE activities constructing the solid geometry representation of the parts and the assembly and the further finite element analysis and post-processing were performed using the SolidWorks 3D CAD Design Software (Dassault Systèmes, SolidWorks Corporation, Concord, MA, USA).

As a first step, based on *.SAT files of the components, SolidWorks parts were generated. The assembled models were then obtained as SolidWorks Assembly, and finally, the finite element models were obtained with the SolidWorks Software included

CosmosWorks FEM feature. The models were automatically meshed with compatible meshing generated between the parts. Parabolic tetrahedral elements were used, with standardized size of 0.35mm. Boundary conditions were set on both sides of the bone section on all models.

Loading conditions were set on the entire occlusal surface of the modeled prosthetic framework. Occlusal loads were applied on a combination of axial and lateral forces, which resulted in a sixty three degree oblique loading over each one of the restored teeth. Loading forces varied for each tooth (Mericske-Stern et al. 1995). First premolar and first molar received 180N load, with 161N downward axial force combined with 80.5N lateral force. Second premolar received 280N load with 250.44N downward axial force combined with 125.22N lateral force.

For this study, all materials were considered to be isotropic, homogeneous and linearly elastic. Mechanical properties (Poisson ratio and Young Modulus) of the materials were obtained from previous reports (Geng et al. 2004, Geng et al. 2001, Sakaguchi & Borgersen 1993): Implants and prosthetic framework – Commercially pure titanium (0.3; 117GPa); Abutment and prosthetic screws – Ti-6Al-4V (0.31; 110GPa); Cortical bone (0.3; 13.7GPa); Trabecular bone (0.3; 1.37GPa). All materials were considered to be isotropic, homogeneous and linearly elastic. Maximum von Mises stress values were determined as well as the stress' distribution pattern and strain for each model. The models are labeled according to the implant-abutment connection system (EH or MT) and the distal implant position (P or I).

Results

All FE models presented similar number of nodes and elements. The external-hexagon parallel implants' model (EH-P) presented 1490816 nodes and 1073719 elements, the external-hexagon distal inclined implant model (EH-I) presented 1494400 nodes and 1075913 elements, the morse-taper parallel implants' model (MT-P) presented 1472904 nodes and 1062819 elements and the morse-taper distal inclined implant model

(MT-I) presented 1431540 nodes and 1030013 elements. The data from the FE analyses are presented as the associated von Mises stress. Colored diagrams were obtained using FEMAP FEM Post Processor (Version 8.3 - Siemens PLM Software, Plano, TX, USA) and are presented for the visualization of the models' stress distribution characteristics. Maximum stress values are presented on Table 1.

Cortical bones' stress distribution pattern was similar on all the models. Higher stress was concentrated at the interface between cortical bone and the implants' neck (Figure 1). Regardless of the position of the implants, maximum von Mises stress on cortical bone was very similar on all models, ranging from 123-126MPa. On trabecular bone, MT parallel implants' models presented lower maximum stress value (9.32MPa) than EH-P (13.30MPa). Once again, little difference on stress values could be observed between the evaluated connection systems in the presence of inclined distal implant (EH-I – 12.80MPa, MT-I – 12.60MPa). Maximum stress was concentrated at the cervical portion of the trabecular bone, for both implant positions. On the inclined implants' models, higher stress presented to be more concentrated around the straight placed implant (Figure 2). Strain concentration on bone was similar to the stress distribution patterns. Strain obtained on the evaluated models is presented on Table 2.

For the implant structure and the prosthetic components, overall, MT models presented higher maximal von Mises stress values. External-hexagon implants presented maximum stress values ranging from 605-705MPa, while morse-taper implants-abutment models presented values ranging from 882-988MPa. On the implants maximum stress was concentrated on the neck of the external-hexagon implants and on the interface between abutment and implant on the morse-taper models (Figure 3). Morse-taper angled abutment presented two-fold higher maximum stress (892MPa) than the external-hexagon angled abutment (432MPa), similar behavior was observed for the prosthetic screws on both parallel and inclined implant positioning. Maximum stress was concentrated at the abutments' platform on the contacting surface with the implants' platform of the external-hexagon models (Figure 4). On the morse-taper connection systems, stress was concentrated at the abutments' cylinder (Figure 4). Despite the difference of maximum stress values on the prosthetic screws, stress was located at the same site for all models, which was around the screws' neck (Figure 5). All frameworks presented concentrated stress at the bottom of the framework, on the contacting area between the framework and the abutment (Figure 6).

Discussion

Patients presenting missing teeth are often interested in a replacement method that will offer them the most natural and long-lasting rehabilitation, manufactured aiming the maximum preservation of healthy structures, minimal surgical risk, cost-effectiveness and low maintenance (Belser et al. 2000). Therefore, implant-supported fixed partial dentures have become a valid option to restore occlusal function of partially edentulous patients. However, when it comes to posterior edentulous areas, implant placing might be difficult, due to bone irregularities or the presence of anatomical landmarks that compromise optimal positioning of the dental implants (Akca & Iplikcioglu 2001, Akca & Iplikcioglu 2002, Canay et al. 1996).

As a rehabilitation alternative, implants have been placed inclined, instead of parallel to each other. Tilted implants have been described as an option to reducing distal cantilever of free-end prosthesis, by means of distal inclination of the implants' platform (Zampelis et al. 2007) aiming to better distribute occlusal loads (Del Fabbro et al. 2010). In the present study, distal implant was mesially inclined, simulating a situation where the mandibular canal would compromise the vertical positioning of the implant. The results obtained suggest that tilting of the distal implant does not promote extreme changes on maximum stress values on bone structures, both trabecular and cortical bone. Similar findings have been reported regarding distal (Zampelis et al. 2007) and mesial (Lan et al. 2010) tilted dental implants. Also, stress distribution patterns on supporting bone were similar for all models. Higher stress was concentrated on surface areas of cortical and trabecular bone, which contradicts a photoelastic analysis (Markarian et al. 2007) on which the isochromatic fringes suggested uneven stress distribution pattern in the presence of tilted implant on a fixed partial rehabilitation, but is in accordance with the known bone resorption pattern around dental implants (Brosh et al. 1998, Canay et al. 1996).

Maximum stress values of cortical bone ranged from 123 to 126MPa, which is bellow the ultimate tensile and compressive strength of cortical bone that has been reported to be around 100-121MPa and 167-173MPa respectively (Akca & Iplikcioglu 2001, Reilly & Burstein 1975). This findings could suggest that cortical bone stress on three-element FPD on mandible would be within the acceptable limits of bone tissue, however, stress values that actually cause biological changes such as resorption and remodeling in the bone are not presently known (Akca & Iplikcioglu 2002). Inasmuch, strains exceeding 3,000με are considered to be above the physiological tolerance threshold of bone and could favor bone resorption due to pathological loading (Duyck et al. 2001). Nonetheless, strain values obtained in the present study were found to be within this limit, also suggesting that the current loading conditions would not represent harm to the supporting bone structures. These assumptions might be sustained by clinical studies, on which survival rates of loaded prosthesis with angled abutments were of 98.2% (Sethi et al. 2000), similar to those with straight abutments or tooth-supported (Pjetursson et al. 2007). The same may be stated for immediately loaded tilted implants, that have also presented high mid-term survival rates (Del Fabbro et al. 2010).

When it comes to the evaluation of prosthetic components, biomechanical studies have been performed to evaluate stress on both prosthetic screws and abutment structures (Alkan et al. 2004, Lin et al. 2008, Pessoa et al. 2010, Wu et al. 2010). In the prosthetic structures, overall stress was higher on the abutment and prosthetic screw of the morse-taper model. Nevertheless, conical connection components usually present greater contact surface, greater extent of contact pressure and frictional resistance (Wu et al. 2010), which could be the reason for, despite higher stress concentration, screw loosening of morse-taper connection prosthesis is less frequent (Lin et al. 2008).

Maximum stress was located at the bottom of the external-hexagon abutments, while for the morse-taper abutments, stress was concentrated at the cylindrical surface that contacts with the inner surface of the implants (Pessoa et al. 2010), regardless of the abutments' angulation. Maximum stress values obtained presented to be higher on implants and abutments for both implant designs, which contradicts previous findings (Pessoa et al. 2010) that demonstrated that on morse-taper connection systems, minor stress would be observed on implants and abutment screw, while external-hexagon systems would present major stress concentration in these components.

Connection type has been reported to mainly affect stress values on the abutment screw (Pessoa et al. 2010), which was not observed in the present study. The comparison of stress values presented by the components of the evaluated implant connection systems suggests that connection type would affect stress on both the prosthetic screw and the abutments. Nevertheless, the presence of angled abutments did not seem to increase stress values within the same connection system models. Stress concentration sites on the abutment screws diverged for the connection systems. Abutment screws of external-hexagon system presented higher stress located close to the screws' threads, while morse-taper abutment screws presented higher stress concentrated at the screws' neck, near the higher stress concentration area of the abutment. Higher stress concentration has been reported to be around the shank of the screws, both prosthetic and abutment screw, regardless of implant connection system (Alkan et al. 2004). This stress distribution pattern was observed on the prosthetic screws of all models evaluated.

Finite element modeling for the evaluation of biomechanical behavior of implant-supported rehabilitations is indeed considered reliable. However, some assumptions that were made in the present study need to be evidenced prior to any conclusion. All materials simulated in this study were considered to be linearly elastic, and bone structures were considered to be homogeneous, isotropic and completely osseointegrated to the implants' surface. Thus, within the limitations of a finite element study, it can be concluded that both external-hexagon and morse-taper connection systems presented similar stress distribution on the supporting bone, and maximum stress values within the ultimate compressive and tensile strength of bone. Nonetheless, abutment and prosthetic screw of morse-taper connection present higher stress than external-hexagon components.

References

Akca, K. & Iplikcioglu, H. (2001) 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 Dentistry* **10**: 238-245.

Akca, K. & Iplikcioglu, H. (2002) Finite element stress analysis of the effect of short implant usage in place of cantilever extensions in mandibular posterior edentulism. *Journal of Oral Rehabilitation* **29**: 350-356.

Alkan, I., Sertgoz, A. & Ekici, B. (2004) Influence of occlusal forces on stress distribution in preloaded dental implant screws. *Journal of Prosthetic Dentistry* **91**: 319-325.

Assunção, W. G., Barão, V. A., Tabata, L. F., Gomes, E. A., Delben, J. A. & dos Santos, P. H. (2009) Biomechanics studies in dentistry: Bioengineering applied in oral implantology. *Journal of Craniofacial Surgery* **20**: 1173-1177.

Belser, U. C., Mericske-Stern, R., Bernard, J. P. & Taylor, T. D. (2000) Prosthetic management of the partially dentate patient with fixed implant restorations. *Clinical Oral Implants Research* **11 Suppl 1**: 126-145.

Branemark, P. I., Hansson, B. O., Adell, R., Breine, U., Lindstrom, J., Hallen, O. & Ohman, A. (1977) Osseointegrated implants in the treatment of the edentulous jaw. Experience from a 10-year period. *Scandinavian Journal of Plastic and Reconstructive Surgery Suppl* **16**: 1-132.

Brosh, T., Pilo, R. & Sudai, D. (1998) The influence of abutment angulation on strains and stresses along the implant/bone interface: Comparison between two experimental techniques. *Journal of Prosthetic Dentistry* **79**: 328-334.

Canay, S., Hersek, N., Akpinar, I. & Asik, Z. (1996) Comparison of stress distribution around vertical and angled implants with finite-element analysis. *Quintessence International* **27**: 591-598.

Del Fabbro, M., Bellini, C. M., Romeo, D. & Francetti, L. (2010) Tilted implants for the rehabilitation of edentulous jaws: A systematic review. *Clinical Implant Dentistry and Related Research*

Duyck, J., Ronold, H. J., Van Oosterwyck, H., Naert, I., Vander Sloten, J. & Ellingsen, J. E. (2001) The influence of static and dynamic loading on marginal bone reactions around osseointegrated implants: An animal experimental study. *Clinical Oral Implants Research* **12**: 207-218.

Eskitascioglu, G., Usumez, A., Sevimay, M., Soykan, E. & Unsal, E. (2004) The influence of occlusal loading location on stresses transferred to implant-supported prostheses and supporting bone: A three-dimensional finite element study. *Journal of Prosthetic Dentistry* **91**: 144-150.

Fugazzotto, P. A. (2008) Shorter implants in clinical practice: Rationale and treatment results. *International Journal of Oral and Maxillofacial Implants* **23**: 487-496.

Geng, J. P., Ma, Q. S., Xu, W., Tan, K. B. & Liu, G. R. (2004) Finite element analysis of four thread-form configurations in a stepped screw implant. *Journal of Oral Rehabilitation* **31**: 233-239.

Geng, J. P., Tan, K. B. & Liu, G. R. (2001) Application of finite element analysis in implant dentistry: A review of the literature. *Journal of Prosthetic Dentistry* **85**: 585-598.

Halg, G. A., Schmid, J. & Hammerle, C. H. (2008) Bone level changes at implants supporting crowns or fixed partial dentures with or without cantilevers. *Clinical Oral Implants Research* **19**: 983-990.

Jemt, T. (2008) Single implants in the anterior maxilla after 15 years of follow-up: Comparison with central implants in the edentulous maxilla. *International Journal of Prosthodontics* **21**: 400-408.

Jung, U. W., Choi, J. Y., Kim, C. S., Cho, K. S., Chai, J. K., Kim, C. K. & Choi, S. H. (2008) Evaluation of mandibular posterior single implants with two different surfaces: A 5-year comparative study. *Journal of Periodontology* **79**: 1857-1863.

Kano, S. C., Binon, P., Bonfante, G. & Curtis, D. A. (2006) Effect of casting procedures on screw loosening in ucla-type abutments. *Journal of Prosthodontics* **15**: 77-81.

Kao, H. C., Gung, Y. W., Chung, T. F. & Hsu, M. L. (2008) The influence of abutment angulation on micromotion level for immediately loaded dental implants: A 3-d finite element analysis. *International Journal of Oral and Maxillofacial Implants* **23**: 623-630.

Kim, Y., Oh, T. J., Misch, C. E. & Wang, H. L. (2005) Occlusal considerations in implant therapy: Clinical guidelines with biomechanical rationale. *Clinical Oral Implants Research* **16**: 26-35.

Lan, T. H., Pan, C. Y., Lee, H. E., Huang, H. L. & Wang, C. H. (2010) Bone stress analysis of various angulations of mesiodistal implants with splinted crowns in the posterior mandible: A three-dimensional finite element study. *International Journal of Oral and Maxillofacial Implants* **25**: 763-770.

Lin, C. L., Wang, J. C., Ramp, L. C. & Liu, P. R. (2008) Biomechanical response of implant systems placed in the maxillary posterior region under various conditions of angulation, bone density, and loading. *International Journal of Oral and Maxillofacial Implants* **23**: 57-64.

Markarian, R. A., Ueda, C., Sendyk, C. L., Lagana, D. C. & Souza, R. M. (2007) Stress distribution after installation of fixed frameworks with marginal gaps over angled and parallel implants: A photoelastic analysis. *Journal of Prosthodontics* **16**: 117-122.

Mericske-Stern, R., Assal, P., Mericske, E. & Burgin, W. (1995) Occlusal force and oral tactile sensibility measured in partially edentulous patients with iti implants. *International Journal of Oral and Maxillofacial Implants* **10**: 345-353.

Pessoa, R. S., Muraru, L., Junior, E. M., Vaz, L. G., Sloten, J. V., Duyck, J. & Jaecques, S. V. (2010) Influence of implant connection type on the biomechanical environment of immediately placed implants - ct-based nonlinear, three-dimensional finite element analysis. *Clinical Implant Dentistry and Related Research* **12**: 219-234.

Pjetursson, B. E., Bragger, U., Lang, N. P. & Zwahlen, M. (2007) Comparison of survival and complication rates of tooth-supported fixed dental prostheses (fdps) and implant-supported fdps and single crowns (scs). *Clinical Oral Implants Research* **18 Suppl 3**: 97-113.

Reilly, D. T. & Burstein, A. H. (1975) The elastic and ultimate properties of compact bone tissue. *Journal of Biomechanics* **8**: 393-405.

Sahin, S., Cehreli, M. C. & Yalcin, E. (2002) The influence of functional forces on the biomechanics of implant-supported prostheses--a review. *Journal of Dentistry* **30**: 271-282.

Sakaguchi, R. L. & Borgersen, S. E. (1993) Nonlinear finite element contact analysis of dental implant components. *International Journal of Oral and Maxillofacial Implants* **8**: 655-661.

Sethi, A., Kaus, T. & Sochor, P. (2000) The use of angulated abutments in implant dentistry: Five-year clinical results of an ongoing prospective study. *International Journal of Oral and Maxillofacial Implants* **15**: 801-810.

Sutpideler, M., Eckert, S. E., Zobitz, M. & An, K. N. (2004) Finite element analysis of effect of prosthesis height, angle of force application, and implant offset on supporting bone. *International Journal of Oral and Maxillofacial Implants* **19**: 819-825.

Tanino, F., Hayakawa, I., Hirano, S. & Minakuchi, S. (2007) Finite element analysis of stress-breaking attachments on maxillary implant-retained overdentures. *International Journal of Prosthodontics* **20**: 193-198.

Wu, T., Liao, W., Dai, N. & Tang, C. (2010) Design of a custom angled abutment for dental implants using computer-aided design and nonlinear finite element analysis. *Journal of Biomechanics* **43**: 1941-1946.

Zampelis, A., Rangert, B. & Heijl, L. (2007) Tilting of splinted implants for improved prosthodontic support: A two-dimensional finite element analysis. *Journal of Prosthetic Dentistry* **97**: S35-43.

Table 1 – Maximum von Mises stress values (MPa) presented by the evaluated models.

Model		Bone		Implants	Abutments		Screw		Framework
		Cortical	Trabecular		Straight	Angled	Abutment	Prosthesis	
EH	P	123	13.30	605	565	-	-	127	260
	I	126	12.80	705	518	432	213	137	290
MT	P	125	9.32	882*		-	-	210	279
	I	123	12.6	988*		892	242	228	352

* On the MT model, implant and straight abutment were modeled as one piece, therefore, maximum stress values were obtained for this combined piece.

Table 2 – Strain on bone (με) presented on the evaluated models.

Model	Cortical bone	Trabecular bone
EH-P	777	840
EH-I	796	809
MT-P	793	589
MT-I	776	799

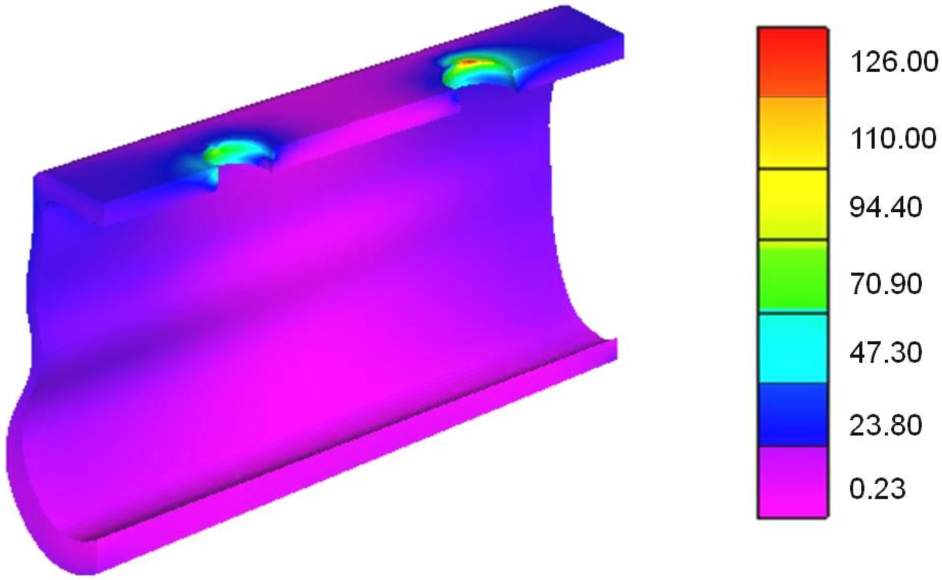

Figure 1 – von Mises stress distribution pattern on cortical bone (MPa). Maximum stress located at the cortical bone surrounding the implant.

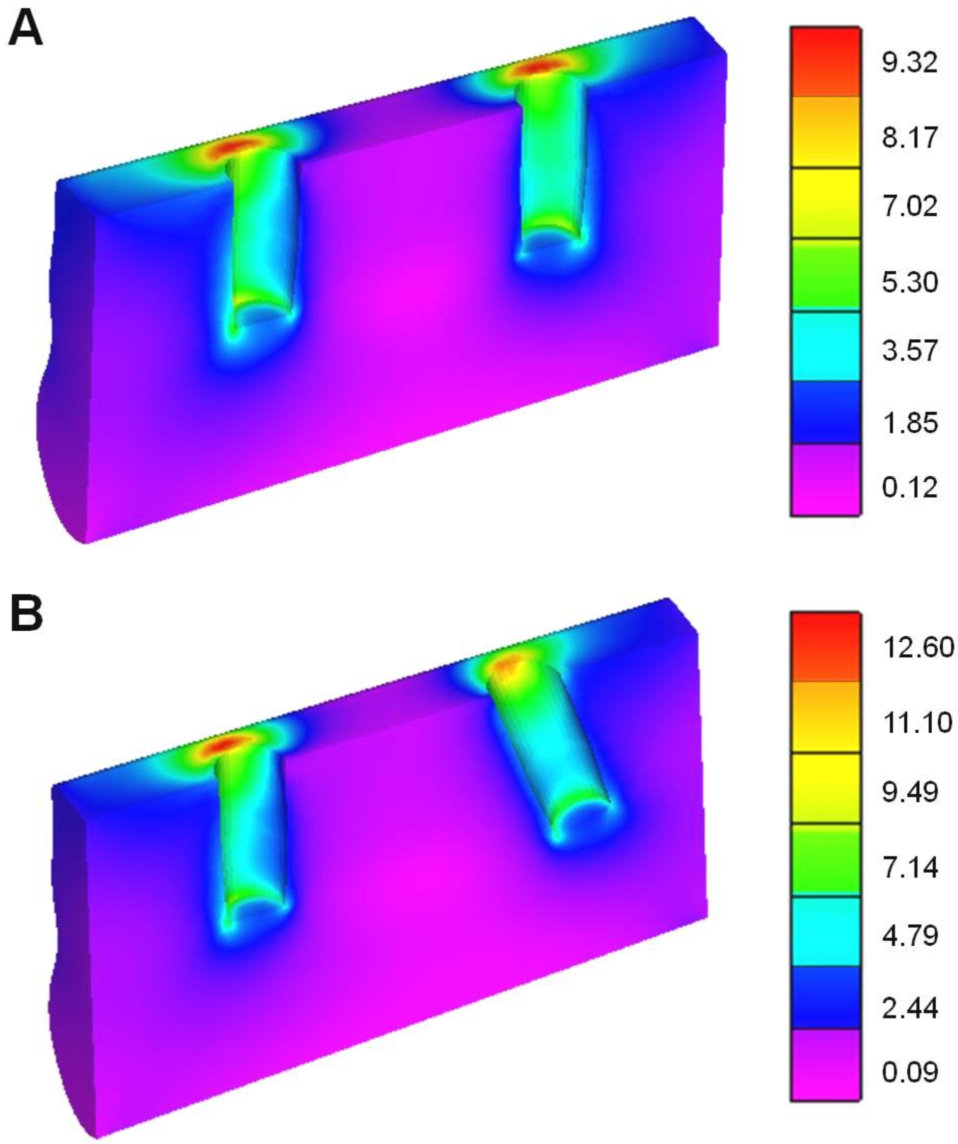

Figure 2 – von Mises stress distribution pattern on trabecular bone (MPa) – A (parallel implants), B (inclined distal implant). Maximum stress is located at the surface of the trabecular bone. On inclined implants' models, higher stress concentration is seen on the straight placed implant.

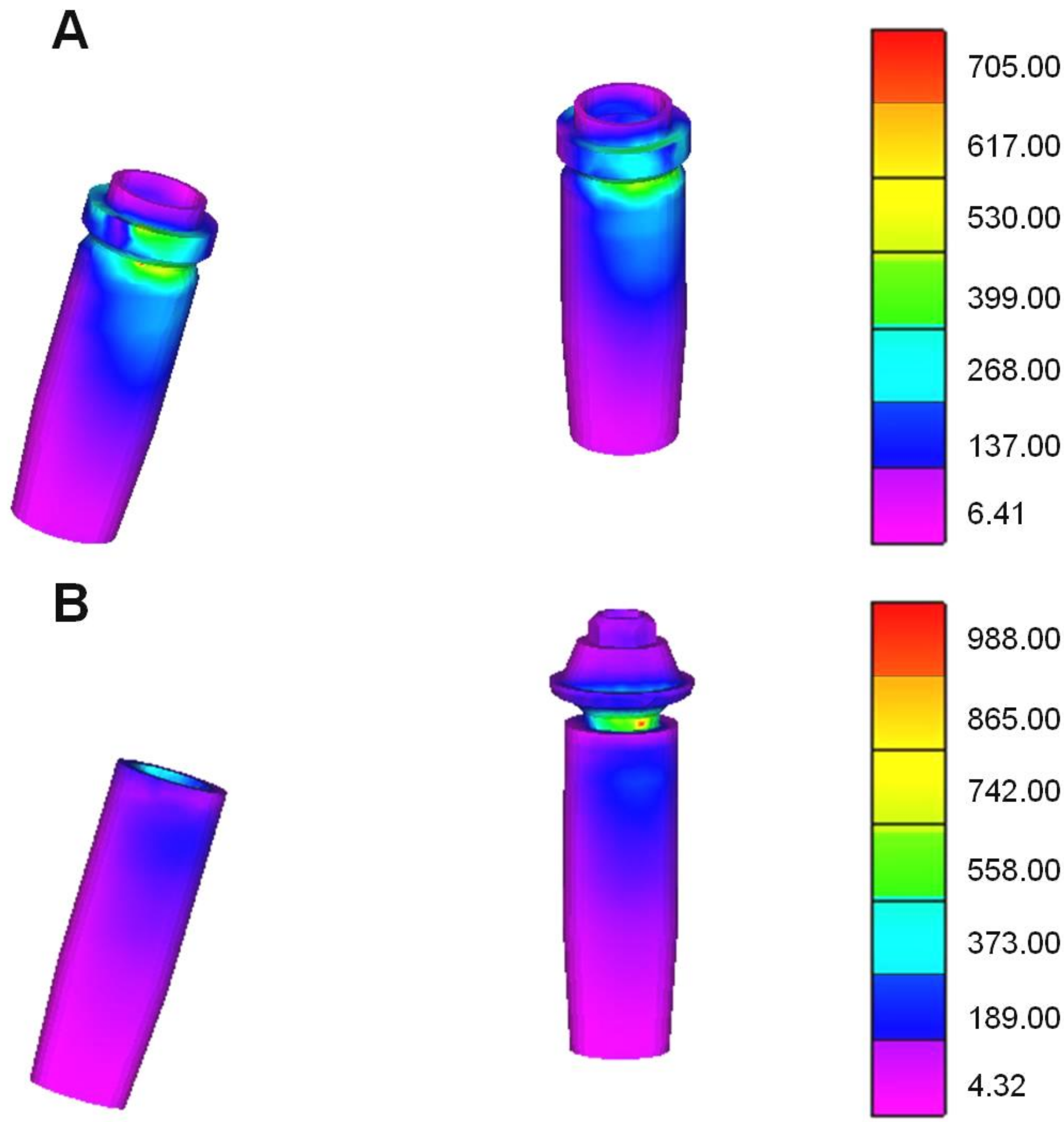

Figure 3 – von Mises stress distribution pattern on the dental implants (MPa). Maximum stress at the implants' neck o HE models (A) and on the contacting surface between the abutment and implant on the MT models (B).

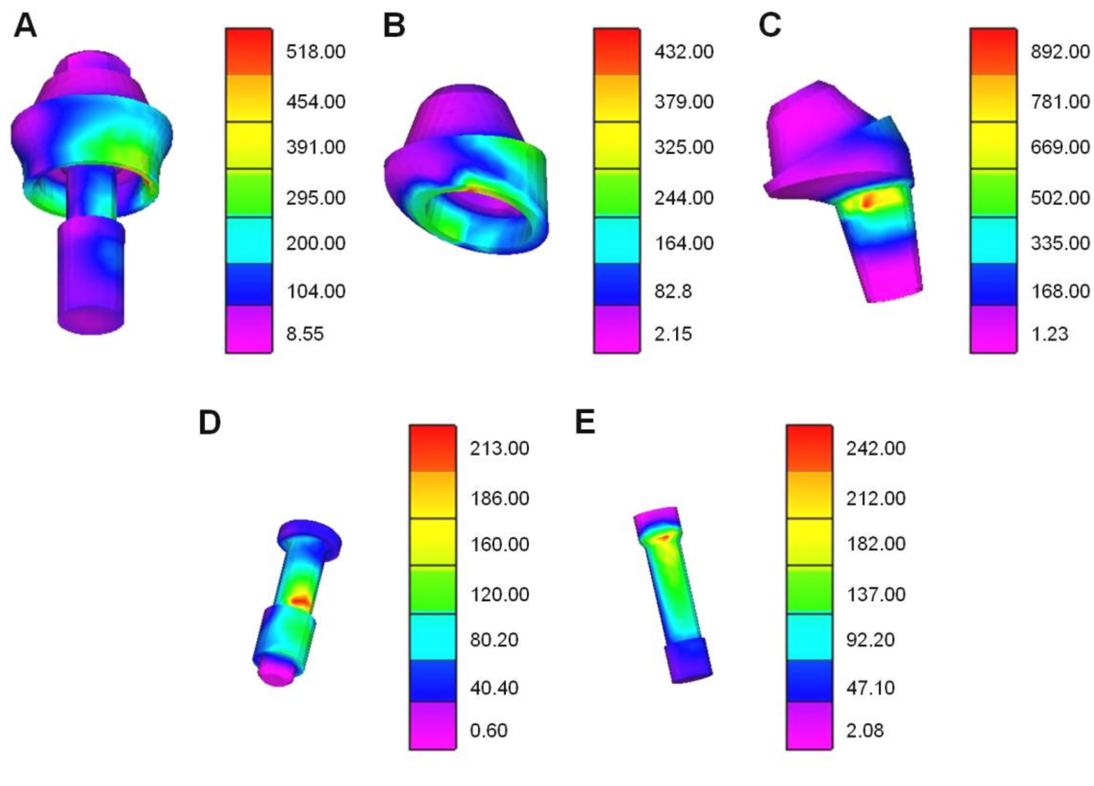

Figure 4 – von Mises stress distribution pattern on abutment components (MPa). Both straight and angled HE abutments presented maximum stress concentrated at the bottom of the abutment (A and B). Morse-taper angled abutment presented higher stress on the abutments cylinder (C). Angled abutment screw presented stress close to the screws' threads (D – HE-I) or close to the screws' neck (E – MT-I).

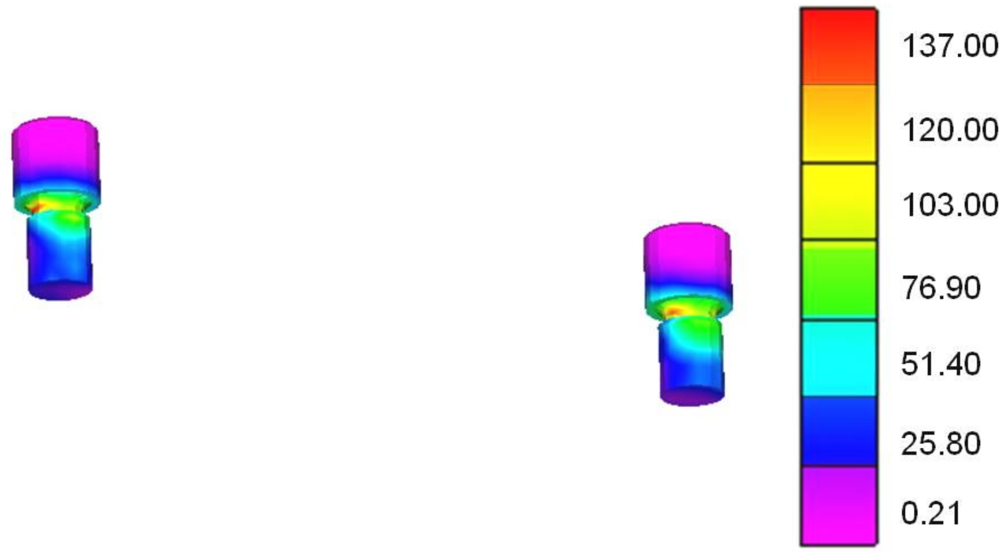

Figure 5 – von Mises stress distribution pattern of the prosthetic screws (MPa). Maximum stress was concentrated around the screws' neck.

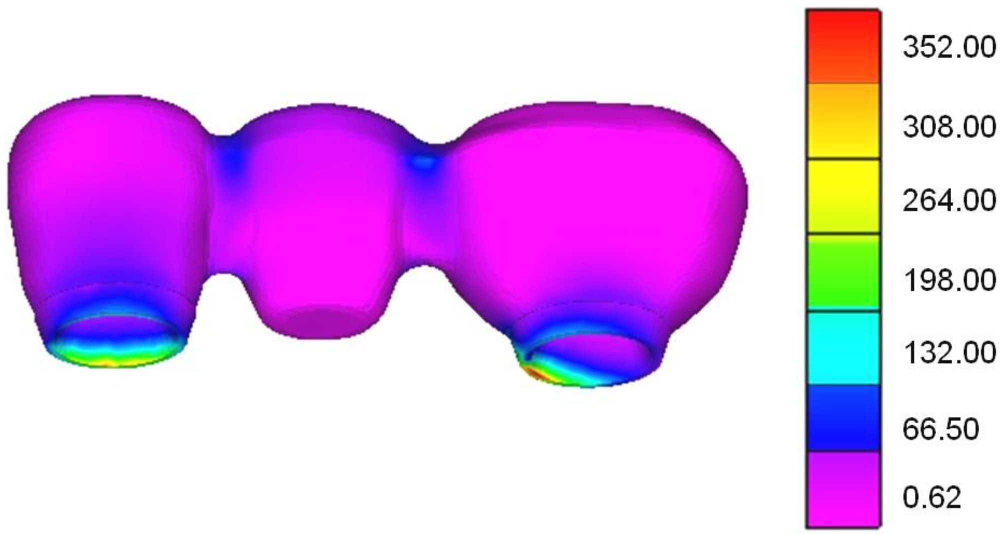

Figure 6 – von Mises stress distribution pattern (MPa). Maximum stress was concentrated at the bottom of the frameworks.

3. Considerações Gerais

Os capítulos previamente apresentados foram idealizados com o objetivo de retratar os questionamentos encontrados no decorrer da realização desta Tese, do desenvolvimento de modelos de elementos finitos e da avaliação da biomecânica de diversas condições clínicas, de carregamento oclusal, seleção e posicionamento de implantes e uso de componentes protéticos angulados.

A avaliação do efeito de simplificações de geometria na obtenção de modelos virtuais que simulam situações clínicas foi realizada com o intuito melhor compreender o comportamento das estruturas modeladas mediante a análise por elementos finitos, além de verificar se a realização destas simplificações poderia comprometer a acuidade dos resultados obtidos, uma vez que estudos utilizando a metodologia de elementos finitos frequentemente são realizados com representação simplificada das roscas dos implantes, sejam estas representadas como anéis concênticos (Chun *et al.*, 2006) ou totalmente ausentes (Kao *et al.*, 2008). Tais simplificações são realizadas principalmente devido à dificuldade de se obter malhas compatíveis entre as estruturas componentes do modelo de elementos finitos e devido às limitações de recursos computacionais para a solução de modelos de maior complexidade, e com número de elementos elevado.

Apesar da alta frequência em que modelos simplificados são apresentados na literatura científica, apenas um artigo científico foi realizado com o intuito de averiguar se estas simplificações comprometem os resultados obtidos. Assunção *et al.* (2009) verificaram por meio de análise bidimensional de elementos finitos, que a não representação das roscas dos implantes não compromete o padrão de distribuição de tensões no osso, mas demonstrou que os valores obtidos apresentam grande variação.

No presente estudo, as variações na modelagem da interface entre implante e osso demonstraram que os resultados obtidos podem ser dependentes da geometria modelada (Hansson & Werke, 2003; Kong *et al.*, 2008), sendo que os maiores valores de tensão foram observados nas regiões de extremidade de rosca. Por sua vez, o padrão de distribuição de tensões, sendo estas concentradas na região cervical dos implantes, com maior concentração no osso cortical é condizente com achados clínicos. A solução de problemas mecânicos por meio da análise de elementos finitos pode gerar tensões e deformações artificialmente elevadas em decorrência de restrições cinemáticas, pontos de aplicação de cargas e regiões de extremidades anguladas. Estes fenômenos são exacerbados quando o problema mecânico avaliado por meio de análises de elementos finitos envolve estruturas biológicas (Dumont *et al.*, 2009). O desenvolvimento destas tensões pontuais elevadas ocorre em função da teoria da elasticidade, à qual tendem os resultados obtidos pelo método de elementos finitos.

Entretanto, de acordo com o princípio de Saint Venant, artefatos de modelagem que geram valores pontuais de tensão excessiva não afetam os valores de tensão nas demais regiões (Horgan, 1989), mas dificultam a determinação precisa dos valores máximos de tensão. Com base nestas afirmações a respeito das análises de tensões, é possível sugerir que os valores de tensão elevados apresentados em regiões de extremidade de rosca podem não ser precisos. As variações nas modelagens das estruturas com rosca avaliadas no presente estudo demonstraram que pequenas alterações nos modelos, com relação à geometria das roscas e área de contato geram oscilações aleatórias dos valores de tensão, que podem não ser reais. Desta maneira, para a análise biomecânica de modelos mais complexos, o uso de superfícies de rosca simplificada ou sem rosca pode gerar valores de tensão mais confiáveis, uma vez que eliminam as regiões críticas para a solução pelo método de elementos finitos.

Uma vez determinada a geometria das interfaces com rosca e verificada a confiabilidade dos resultados apresentados por modelos simplificados, foram realizados questionamentos a respeito das condições de contorno dos modelos de elementos finitos. Assim, como nas situações de simplificações de roscas em modelos de elementos finitos, muitos artigos científicos são realizados com a aplicação apenas de cargas axiais, localizadas em apenas um elemento da reabilitação protética. Entretanto, sabe-se que os movimentos mastigatórios promovem forças axiais e momentos fletores que afetam as tensões geradas em implantes e estruturas ósseas (Geng *et al.*, 2001). A associação de forças axiais e forças laterais gera forças resultantes oblíquas, mais condizentes com a situação clínica dos movimentos mastigatórios. Foi observado que a aplicação de carga oclusal com força resultante oblíqua promoveu aumento significativo nos valores de tensão apresentados nos modelos avaliados, o que indica que valores de tensão obtidos por meio de aplicação de carga oclusal exclusivamente no sentido axial podem minimizar os efeitos nos tecidos de suporte quando a reabilitação está em função.

Além da influência do tipo de forças oclusais aplicadas nos modelos, foi observado que os tipos de conexão do implante com o componente protético geram valores de tensão diferentes nos componentes protéticos de reabilitações implantossuportadas. Em reabilitações em implantes posicionados paralelos, a conexão de hexágono interno apresentou menor tensão nos componentes protéticos, enquanto as conexões de hexágono externo e cone morse apresentaram menor tensão no osso cortical. Já quando consideradas as reabilitações com implante distal inclinado, os modelos de conexão do tipo cone morse apresentaram maior tensão nos componentes protéticos angulados do que os modelos de hexágono externo. Estes comportamentos podem ser decorrentes da variação na área de contato entre os componentes e os implantes.

Pouca variação nos valores de tensão nos tecidos de suporte foi observada, independente do tipo de conexão ou inclinação dos implantes. Entretanto, o padrão de distribuição de tensão na presença de componentes angulados indicou maior concentração de tensão no implante posicionado verticalmente, provavelmente decorrente da maior área de contato no implante inclinado de ambas as conexões, que compensaria o efeito do momento fletor sobre o mesmo.

Baseado na semelhança nos padrões de distribuição de tensão entre os sistemas de conexão e as variações nos valores de tensão dos componentes protéticos, sugere-se que a utilização de componentes de conexão interna pode gerar reabilitações com maior estabilidade entre os componentes protéticos, sem comprometer os tecidos de suporte.

4. Conclusão

Dentro das limitações deste estudo e da metodologia de elementos finitos. Pode-se concluir, para as situações e modelos avaliados, que:

- O comportamento biomecânico (valores de tensão e padrões de distribuição de tensão) de modelos sem representação de rosca é semelhante ao de modelos com representação de rosca;
- A aplicação de cargas oclusais oblíquas promove aumento significativo nos valores de tensão de reabilitações implantossuportadas;
- O tipo de conexão do implante não interfere no padrão de distribuição de tensão aos tecidos de suporte e componentes protéticos;
- O tipo de conexão do implante gera diferentes valores de tensão nos componentes protéticos;
- O posicionamento inclinado de implantes não compromete a integridade dos tecidos de suporte.

5. Referências*

Abduo J, Bennani V, Waddell N, Lyons K, Swain M. Assessing the fit of implant fixed prostheses: a critical review. Int J Oral Maxillofac Implants 2010; 3(25): 506-15.

Akca K, Cehreli MC, Iplikcioglu H. A comparison of three-dimensional finite element stress analysis with in vitro strain gauge measurements on dental implants. Int J Prosthodont 2002; 2(15): 115-21.

Assuncao WG, Barao VA, Tabata LF, Gomes EA, Delben JA, dos Santos PH. Biomechanics studies in dentistry: bioengineering applied in oral implantology. J Craniofac Surg 2009; 4(20): 1173-7.

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; 11(25): 847-58.

Branemark PI, Hansson BO, Adell R, Breine U, Lindstrom J, Hallen O *et al.* Osseointegrated implants in the treatment of the edentulous jaw. Experience from a 10-year period. Scand J Plast Reconstr Surg Suppl 1977; 16): 1-132.

Canay S, Hersek N, Akpinar I, Asik Z. Comparison of stress distribution around vertical and angled implants with finite-element analysis. Quintessence Int 1996; 9(27): 591-8.

* De acordo com a norma UNICAMP/FOP, baseado no modelo Vancouver. Abreviatura dos periódicos em conformidade com a Base de dados Medline.

Cehreli M, Duyck J, De Cooman M, Puers R, Naert I. Implant design and interface force transfer. A photoelastic and strain-gauge analysis. Clin Oral Implants Res 2004; 2(15): 249-57.

Chun HJ, Shin HS, Han CH, Lee SH. Influence of implant abutment type on stress distribution in bone under various loading conditions using finite element analysis. Int J Oral Maxillofac Implants 2006; 2(21): 195-202.

Clelland NL, Gilat A, McGlumphy EA, Brantley WA. A photoelastic and strain gauge analysis of angled abutments for an implant system. Int J Oral Maxillofac Implants 1993; 5(8): 541-8.

Degerliyurt K, Simsek B, Erkmen E, Eser A. Effects of different fixture geometries on the stress distribution in mandibular peri-implant structures: a 3-dimensional finite element analysis. Oral Surg Oral Med Oral Pathol Oral Radiol Endod 2010; 2(110): e1-11.

Eskitascioglu G, Usumez A, Sevimay M, Soykan E, Unsal E. The influence of occlusal loading location on stresses transferred to implant-supported prostheses and supporting bone: A three-dimensional finite element study. J Prosthet Dent 2004; 2(91): 144-50.

Fugazzotto PA. Shorter implants in clinical practice: rationale and treatment results. Int J Oral Maxillofac Implants 2008; 3(23): 487-96.

Geng JP, Ma QS, Xu W, Tan KB, Liu GR. Finite element analysis of four thread-form configurations in a stepped screw implant. J Oral Rehabil 2004; 3(31): 233-9.

Geng JP, Tan KB, Liu GR. Application of finite element analysis in implant dentistry: a review of the literature. J Prosthet Dent 2001; 6(85): 585-98.

Halg GA, Schmid J, Hammerle CH. Bone level changes at implants supporting crowns or fixed partial dentures with or without cantilevers. Clin Oral Implants Res 2008; 10(19): 983-90.

Hansson S, Werke M. The implant thread as a retention element in cortical bone: the effect of thread size and thread profile: a finite element study. J Biomech 2003; 9(36): 1247-58.

Holmes DC, Loftus JT. Influence of bone quality on stress distribution for endosseous implants. The Journal of oral implantology 1997; 3(23): 104-11.

Horgan CO. Recent developments concerning Saint-Venant's principle: An update. Appl Mech Rev 1989; 11(42): 295-303.

Jemt T. Single implants in the anterior maxilla after 15 years of follow-up: comparison with central implants in the edentulous maxilla. Int J Prosthodont 2008; 5(21): 400-8.

Jung UW, Choi JY, Kim CS, Cho KS, Chai JK, Kim CK *et al.* Evaluation of mandibular posterior single implants with two different surfaces: a 5-year comparative study. Journal of periodontology 2008; 10(79): 1857-63.

Kano SC, Binon P, Bonfante G, Curtis DA. Effect of casting procedures on screw loosening in UCLA-type abutments. J Prosthodont 2006; 2(15): 77-81.

Kao HC, Gung YW, Chung TF, Hsu ML. The influence of abutment angulation on micromotion level for immediately loaded dental implants: a 3-D finite element analysis. Int J Oral Maxillofac Implants 2008; 4(23): 623-30.

Karl M, Winter W, Taylor TD, Heckmann SM. In vitro study on passive fit in implant-supported 5-unit fixed partial dentures. Int J Oral Maxillofac Implants 2004; 1(19): 30-7.

Kong L, Hu K, Li D, Song Y, Yang J, Wu Z *et al.* Evaluation of the cylinder implant thread height and width: a 3-dimensional finite element analysis. Int J Oral Maxillofac Implants 2008; 1(23): 65-74.

Korioth TW, Versluis A. Modeling the mechanical behavior of the jaws and their related structures by finite element (FE) analysis. Crit Rev Oral Biol Med 1997; 1(8): 90-104.

Lin CL, Wang JC, Ramp LC, Liu PR. Biomechanical response of implant systems placed in the maxillary posterior region under various conditions of angulation, bone density, and loading. Int J Oral Maxillofac Implants 2008; 1(23): 57-64.

Maeda Y, Satoh T, Sogo M. In vitro differences of stress concentrations for internal and external hex implant-abutment connections: a short communication. J Oral Rehabil 2006; 1(33): 75-8.

Quaresma SE, Cury PR, Sendyk WR, Sendyk C. A finite element analysis of two different dental implants: stress distribution in the prosthesis, abutment, implant, and supporting bone. The Journal of oral implantology 2008; 1(34): 1-6.

Sahin S, Cehreli MC, Yalcin E. The influence of functional forces on the biomechanics of implant-supported prostheses--a review. J Dent 2002; 7-8(30): 271-82.

Sethi A, Kaus T, Sochor P. The use of angulated abutments in implant dentistry: five-year clinical results of an ongoing prospective study. Int J Oral Maxillofac Implants 2000; 6(15): 801-10.

Sutpideler M, Eckert SE, Zobitz M, An KN. Finite element analysis of effect of prosthesis height, angle of force application, and implant offset on supporting bone. Int J Oral Maxillofac Implants 2004; 6(19): 819-25.

Tanino F, Hayakawa I, Hirano S, Minakuchi S. Finite element analysis of stress-breaking attachments on maxillary implant-retained overdentures. Int J Prosthodont 2007; 2(20): 193-8.

Van Oosterwyck H, Duyck J, Vander Sloten J, Van der Perre G, De Cooman M, Lievens S *et al.* The influence of bone mechanical properties and implant fixation upon bone loading around oral implants. Clin Oral Implants Res 1998; 6(9): 407-18.