UNESP – Universidade Estadual Paulista
“Júlio de Mesquita Filho”
Faculdade de Odontologia de Araraquara

Cássia Bellotto Corrêa

Comportamento mecânico de sistemas prótese/implante em região anterior de maxila: Análise pelo método dos elementos finitos e interferometria holográfica

Araraquara

2013

UNESP – Universidade Estadual Paulista
“Júlio de Mesquita Filho”
Faculdade de Odontologia de Araraquara

Cássia Bellotto Corrêa

Comportamento mecânico de sistemas prótese/implante em região anterior de maxila: Análise pelo método dos elementos finitos e interferometria holográfica

Tese apresentada ao Programa de Pós-Graduação em Odontologia - Área de Implantodontia, da Faculdade de Odontologia de Araraquara, Universidade Estadual Paulista “Júlio de Mesquita Filho”, para obtenção do título de Doutor em Odontologia.

Orientador: Prof. Dr. Luís Geraldo Vaz
Coorientadores: Prof. Dr. Rogério Margonar e Prof. Dr. Pedro Yoshito Noritomi

Araraquara

2013

Corrêa, Cássia Belloto.
Comportamento mecânico de sistemas prótese/implante em região anterior de maxila: análise pelo método dos elementos finitos e interferometria holográfica / Cássia Belloto Corrêa. – Araraquara: [s.n.], 2013.
93 f. ; 30 cm.

Tese (Doutorado) – Universidade Estadual Paulista, Faculdade de Odontologia
Orientador : Prof. Dr. Luis Geraldo Vaz.

1. Biomecânica 2. Prótese dentária fixada por implante 3. Implantes dentários 4. Análise numérica assistida por computador 5. Interferometria. I. Título

Ficha catalográfica elaborada pela Bibliotecária Ceres Maria Carvalho Galvão de Freitas, CRB-8/4612 , Serviço Técnico de Biblioteca e Documentação da Faculdade de Odontologia de Araraquara / UNESP

Cássia Bellotto Corrêa

Comportamento mecânico de sistemas prótese/implante em região anterior de maxila: Análise pelo método dos elementos finitos e interferometria holográfica

COMISSÃO JULGADORA

TESE PARA OBTENÇÃO DO GRAU DE DOUTOR

Presidente e orientador: Luís Geraldo Vaz

2° Examinador: Carlos Alberto dos Santos Cruz

3° Examinador: Valfrido Antonio Pereira Filho

4° Examinador: Mário Augusto Pires Vaz

5° Examinador: Rodrigo Alvarenga Rezende

Araraquara, 27 de março de 2013.

DADOS CURRICULARES

Cássia Bellotto Corrêa

Nascimento: 19/12/1984 – Piracicaba/SP

Filiação: Ariovaldo Corrêa
Ana Inês Bellotto Corrêa

2003/2006 **Graduação em Odontologia**
Faculdade de Odontologia de Araraquara – UNESP

2004/2006 **Estágio de Iniciação Científica na Disciplina de Materiais Dentários**
Orientador: Prof. Dr. Luís Geraldo Vaz
Faculdade de Odontologia de Araraquara – UNESP

2005/2006 **Bolsista FAPESP de Iniciação Científica (2005/03166-0)**
Orientador: Prof. Dr. Luís Geraldo Vaz

2006/2007 **Curso de Aperfeiçoamento em Reabilitação Oral com Procedimentos Integrados**
Associação Paulista de Cirurgiões Dentistas/Escola de Aperfeiçoamento Profissional – APCD/EAP (Regional Araraquara)

2007 **Estágio de Atualização na Disciplina de Clínica Integrada**
Orientadora: Profa. Dra. Andréia Affonso Barreto Montandon

Faculdade de Odontologia de Araraquara – UNESP

2007 **Bolsista FAPESP de Treinamento Técnico (2006/06844-2)**
Orientador: Prof. Dr. Luis Geraldo Vaz

2007/2008 **Curso de Capacitação em Prótese sobre Implante**
Associação Paulista de Cirurgiões Dentistas/Escola de Aperfeiçoamento Profissional – APCD/EAP (Regional Araraquara)

2008/2009 **Curso de Atualização Implantes Dentários**
CAECO- Centro de Atendimento Odontológico, Araraquara-SP

2009/2011 **Especialização em Implantodontia**
Centro Universitário de Araraquara, UNIARA

2009/2013 **Pós-Graduação em Odontologia – Nível Doutorado**
Área de Concentração Periodontia
Faculdade de Odontologia de Araraquara – UNESP
Bolsista CAPES / CNPq

2011/2012 **Estágio de Doutorado Sanduíche**
Faculdade de Engenharia Mecânica
Laboratório de Ótica e Mecânica Experimental
Universidade do Porto, Porto- Portugal
Bolsista CAPES

DEDICATÓRIA

Dedico este trabalho a **Deus**, por ter estado sempre comigo, me conduzindo e iluminando todas as minhas decisões que possibilitaram a concretização de mais uma etapa da minha formação profissional.

Aos meus pais **Ana Inês e Ariovaldo,** que sempre me deram apoio, suporte e amor. Pelo exemplo de vida, caráter, sabedoria e coragem e pelos ensinamentos dos reais valores da vida. Minha eterna gratidão.

Às minhas irmãs, **Ariana, Alina Flávia e Ana Elisa** pelo carinho, amizade, incentivo e apoio para seguir em frente em busca dos meus sonhos.

Aos meus sobrinhos, **Luka, Enzo, Pedro e Júlia,** pelo carinho e pela alegria proporcionada a toda a família.

AGRADECIMENTOS ESPECIAIS

Ao meu orientador, **Prof. Dr. Luis Geraldo Vaz,** agradeço pela oportunidade de crescimento pessoal e profissional.

A agradável convivência, amizade e confiança em mim depositada, tornaram a relação orientador/orientada muito prazerosa.

Aos meus co-orientadores:

Prof. Dr. Rogério Margonar, com quem compartilhei as primeiras idéias deste trabalho. Seu incentivo e apoio foram fundamentais na execução deste trabalho.

Prof. Dr. Pedro Yroshito Noritomi, pela generosidade e paciência com que me ensinou a dar os primeiros passos nessa linha de pesquisa.

Ao **Prof. Dr. Mário Augusto Pires Vaz,** pela oportunidade de realização do estágio de doutorado sanduíche, pelo acolhimento na Universidade do Porto e excelente suporte.

AGRADECIMENTOS

À Faculdade de Odontologia de Araraquara – UNESP, em nome da Ilma. Sra. Diretora, Prof.ª Dr.ª Andréia Affonso Barretto Montandon, e do Ilma. Sr. Vice- Diretora, Prof.ª Dr.ª Elaine Maria Sgavioli Massucato, pela oportunidade de ganho profissional.

Ao Centro de Tecnologia da Informação Renato Archer, Campinsa-SP, pelo assessoramento e imprescindível assistência na execução do trabalho, Dr. Pedro Yoshito Noritomi, Cesar Augusto Rocha Laureti, Tathy A. Xavier, Daniel Takanori Kemmoku e toda equipe de Tecnologias Tridimensionais.

À Faculdade de Engenharia da Universidade do Porto – FEUP pelo acolhimento, em especial ao LOME, Laboratório de Ótica e Mecânica Experimental na pessoa de Nuno Viriato Ramos e Jaime Monteiro pela competência e excelente direcionamento na execução dos trabalhos.

À CAPES e ao CNPq, pela concessão da Bolsa de estudos.

À FAPESP, pelo financiamento concedido à solicitação Auxílio à Pesquisa.

À empresa Neodent (Curitiba-PR, Brasil), pelo fornecimento dos CADs e apoio ao desenvolvimento deste projeto.

Aos professores da Disciplina de Periodontia, Implantodontia, Materiais Dentários e Odontologia Integrada da Faculdade de Odontologia de Araraquara - UNESP, pelos conhecimentos adquiridos.

A todos os funcionários do Departamento de Diagnóstico e Cirurgia e Departamento de Materiais Dentários e prótese da Faculdade de Odontologia de Araraquara – UNESP.

Aos funcionários da Biblioteca da Faculdade de Odontologia de Araraquara – UNESP, pela atenção e disponibilidade em nos auxiliar.

Aos funcionários da Seção de Pós Graduação, da Faculdade de Odontologia de Araraquara – UNESP, em especial a Mara, pela dedicação e atenção em nos auxiliar.

Aos meus amigos companheiros de turma de Pós Graduação.

A toda equipe do professor Geraldo: As pós-doutorandas Ana Lúcia e Laiza Maria pela amizade, disponibilidade, paciência, sinceridade e imensa ajuda. Em especial à Ana Lúcia pelos 7 anos de excelente convivência e amizade. Pelo acolhimento em Portugal e em Araraquara.

Aos alunos de Iniciação Científica que tive a oportunidade e prazer de co-orientar, Sarah Bessi Lopes e Henrique Eije Tabuse, dos quais tenho grande admiração.

Aos amigos de Especialização pela oportunidade de compartilhar experiências, em especial ao meu parceiro de clínica Prof. Dr. Renato Berro.

Às Cirurgiãs-Dentistas: Ariana Bellotto Corrêa Kassawara e Kathya Palatim Semêncio pelo exemplo de profissionalismo.

Às minhas amigas queridas que sempre com muito carinho me apoiaram e torceram por mim, Melissa, Melina, Ana Paula, Cássia, Andréia, Paula, Karina, Gisele, Júlia, Camilla, Michele, Mara.

À minha amiga querida Karina pelo carinho e companheirismo em toda nossa estadia em Portugal.

A todos aqueles que, direta ou indiretamente, contribuíram para a realização deste trabalho.

Corrêa CB. Comportamento mecânico de sistemas prótese/implante em região anterior de maxila: análise pelo método dos elementos finitos e interferometria holográfica [tese de doutorado]. Araraquara: Faculdade de Odontologia da Unesp; 2013.

Resumo

A proposta deste estudo foi analisar, por meio do método de elementos finitos (MEF), o comportamento mecânico das estruturas envolvidas em uma reabilitação implanto suportada de 4 elementos na região anterior de maxila, utilizando apenas 2 implantes, variando os posicionamentos dos implantes e o tipo de conexão protética (hexágono externo, hexágono interno e cone-Morse) e ainda validar metodologia de análise computacional por meio da interferometria holográfica, uma técnica experimental. Modelos tridimensionais baseados em um banco de tomografias computadorizadas foram criados para todas as situações estudadas (Implantes nos Incisivos Laterais - IL; Implantes nos Incisivos Centrais - IC e Implante no Incisivo Central e Incisivo Lateral - ILIC). Uma carga equivalente à 150N foi aplicada a 45° na região de cíngulo de cada elemento dentário. Para a validação da metodologia de elementos finitos por meio da interferometria holográfica, criou-se um modelo prototipado com as mesmas situações estudadas e comparou-se os deslocamentos direcionais resultantes da aplicação de 3 intensidades de força. Como resultado foi observado: o menor deslocamento da estrutura protética e maior deslocamento na estrutura óssea no grupo IC. No tecido ósseo, a tensão de von Misses localiza-se predominantemente na região cortical. O valor máximo de tensão de von Misses foi observado no implante vizinho ao cantilever no grupo ILIC. Na conexão cone Morse, o parafuso protético mostrou metade da tensão de von Misses em relação a conexão hexágono interno e hexágono externo. Comparando-se os deslocamentos direcionais nas duas metodologias, verificou-se que havia uma correlação positiva entre os resultados, porém os valores obtidos experimentalmente eram superiores aos do modelo computacional. Pode-se concluir que o posicionamento dos implantes e o tipo de conexão protética em reabilitação anterior de maxila influenciam a distribuição e intensidade de tensão/deformação nas estruturas da prótese implanto-suportada e ainda que o MEF pode ser usado como uma eficiente técnica para a análise de tensão, deformação e deslocamento quando utilizada com modelos validados experimentalmente.

Palavras-Chave: Biomecânica, Prótese Dentária Fixada por Implante, Implantes Dentários, Análise Numérica Assistida por Computador e Interferometria.

Corrêa CB. Mechanical behavior of prosthesis / implant systems in the anterior maxilla: analysis by finite element method and holographic interferometry [tese de doutorado]. Araraquara: Faculdade de Odontologia da Unesp; 2013.

Abstract

The purpose of this study was to analyze, by finite element method (FEM), the mechanical behavior of structures involved in an implant supported rehabilitation of 4 elements in the anterior maxilla, using only 2 implants, varying the placements of the implants and prosthetic connection types (external hexagon, internal hexagon and Morse taper) Moreover to validate the methodology of MEF by holographic interferometry, an experimental technique. Three-dimensional models based on a database CT scans were created for all situations studied (Implants in Lateral Incisors - IL; Implants in Central Incisors IC and Implant in Central e Lateral Incisor - ILIC). An equivalent load of150N was applied at 45° in the cingulum of each tooth. To validate the finite element method by Holographic Interferometry, a prototyped model was created with the same situations studied and compared the directional displacements, resulting from the application of the 3 intensities of force. As results it was observed: the lowest prosthetic displacement structure and greater displacement in the bone structure in the IC group. In bone tissue, the von Mises stress is located predominantly in the cortical region. The maximum value of von Mises stress was observed in the implant neighboring to cantilever in the group ILIC. In Morse taper connection, the prosthetic screw showed half the von Misses stress in relation to internal and external hexagon connection. Comparing the directional displacement in both methodologies, an positive correlation between the results was found, although the experimentally values obtained were higher than those of the computational model. It can be concluded that the placement of implants and prosthetic connection type have influence in the intensity and distribution of stress/strain on the implant-supported rehabilitation of anterior maxilla and moreover the FEM can be used as an efficient technique for analysis of stress, strain and displacement when used with experimentally validated models.

Keywords: Biomechanics, Dental Prosthesis Implant-Supported, Dental Implants, Numerical Analysis, Interferometry.

SUMÁRIO

Introdução

A reabilitação de pacientes edêntulos foi revolucionada a partir de 1969 por Branemark[3] quando as próteses totais começaram a ser implanto suportadas, o que possibilitou uma melhora na devolução da função mastigatória e estética para os pacientes reabilitados. A partir disso, os implantes se tornaram uma alternativa viável para a substituição de dentes perdidos, tanto em casos de edentulismo total, parcial ou perdas unitárias.

Os altos índices de sucesso do tratamento reabilitador com implantes dentais e próteses implanto-suportadas tornaram possíveis, não apenas a ampliação das indicações funcionais, como também o aumento da demanda estética por parte dos pacientes e clínicos. Dessa forma, a obtenção de um resultado estético em uma reabilitação, vem sendo um dos principais fatores a serem abordados desde o planejamento, até a execução e finalização dos tratamentos restauradores.

Dentre os critérios para a estética satisfatória para o tratamento com implantes dentais, está o estabelecimento de tecidos moles com adequado contorno, perfil gengival e presença de papilas interproximais[5]. A ausência de papilas interproximais tanto em dentição hígida, como em casos restaurados com próteses fixas e próteses sobre implantes, pode levar a deformidades estéticas, dificuldade fonética e impacção alimentar[16]. Por outro lado, a presença de uma papila saudável que preencha todo o espaço interdental, mantém não só uma estética satisfatória, mas também da saúde, fazendo com que as funções estéticas, fonéticas e mastigatórias sejam mantidas.

Para obtenção de resultados estéticos e funcionais satisfatórios é indispensável o adequado posicionamento tridimensional dos implantes. O posicionamento vestibulolingual do implante é determinado principalmente pelas condições anatômicas e oclusais do paciente e pelo planejamento protético. A posição vestibulolingual ideal para instalação de um implante deve ser obtida a partir da visualização de uma linha imaginária que liga a margem incisal dos dentes adjacentes[4] e também, pela manutenção de espessura óssea com cerca de 2mm na parede vestibular e 2mm na parede lingual, estando o implante instalado no centro da crista óssea[10].

Em um estudo clínico e radiográfico realizado por Novaes et al.[13], observou-se que há a presença de papila quando a distância horizontal entre implantes era de 2 ou 3mm sem diferença significante em termos de formação de papila. E, para assegurar a presença da papila, a distância vertical, da crista óssea ao ponto de contato era de 3mm entre implantes adjacentes e 3, 4 ou 5mm entre dente e implante[8].

Salama et al.[14] em seu estudo clínico publicou um guia de previsibilidade da ocorrência da papila gengival. Segundo o autores, em áreas proximais localizadas entre dentes naturais a papila terá previsibilidade de altura de 5mm; em áreas localizadas entre implante e dentes naturais, a papila gengival terá uma previsibilidade de altura de 4,5mm, o que é bem semelhante à condição encontrada para os dentes naturais. Todavia, em áreas proximais localizadas entre dois implantes, os autores afirmam que a previsibilidade da altura da papila gengival é de apenas 3,5mm. É interessante salientar que os autores observaram também as áreas proximais em pônticos de próteses fixas e notaram que, entre dentes naturais e um pôntico, a altura papilar pode chegar, em média, a 6,5mm; para uma área proximal entre implante e pôntico, a altura da papila tem uma previsibilidade de altura em torno de 5,5mm; e a altura da papila gengival, entre dois pônticos chega a ter, em média, 6mm de altura. Os autores tentam, com essas observações, apresentar um guia indispensável para o planejamento e previsibilidade estética das reabilitações na região anterior da maxila.

Quando as distâncias ideais não forem respeitadas, poderá ocorrer perda da crista óssea entre dente e implante, causando assim a perda da papila. Porém, se esse distanciamento for respeitado, a perda óssea deverá ser localizada apenas ao redor do implante e, com a crista óssea ao redor dos dentes preservada, a sustentação dos tecidos moles peri-implantares será mantida.

Em uma região anterior superior com ausência dos quatro incisivos, para a colocação de um implante para cada dente o espaço necessário é de aproximadamente 29mm, uma vez que cada implante possui cerca de 4mm de diâmetro e o espaço ideal entre implantes é de 3mm e entre dente e implante é 2mm. Grande parte dos pacientes não possui esse espaço disponível para respeitar o posicionamento tridimensional adequado, sendo assim, é indicada a colocação de apenas dois ou três implantes associados a um ou dois elementos suspensos, para a otimização do espaço e ganho estético, como sugerem Tarnow et al.[16,17].

Com a necessidade de utilização de pônticos em uma prótese múltipla, os implantes de fixação estarão submetidos a uma maior intensidade de força. Sabe-se ainda que uma das causas de insucesso na implantodontia está na falta de conhecimento sobre biomecânica, visto que o sistema prótese/implante não deve sofrer tensões excessivas, tanto nos eixos axiais quanto oblíquos, o que resultaria no processo de reabsorção óssea e consequentemente insucesso do tratamento[7].

Pesquisas alertam a respeito da sobrecarga existente sobre o complexo prótese/implante[3,11,12], a qual justifica a presença de cargas axiais e não axiais com perda óssea marginal em implantes[2], o que prejudica a estética final da reabilitação protética.

Após a instalação de um implante, cargas são transmitidas e distribuídas diretamente ao osso adjacente, e este assim como os demais tecidos adjacentes, sofre uma modificação para o estabelecimento de novas distâncias biológicas, de qual faz parte a reabsorção óssea. Se forças excessivas forem aplicadas, essa reabsorção óssea é acentuada[11,12,15].

A longevidade do tratamento pode aumentar se a distribuição das cargas no osso for melhor compreendida[6]. Deste modo, todo esforço deve ser feito para conseguir uma localização ideal do implante para manter a estética ideal e também evitar o carregamento excêntrico e consequente fracasso dos implantes.

Existem alguns métodos para analise da biomecânica nos implantes, entre eles: o método dos elementos finitos, a fotoelasticidade, a mensuração de cargas in vivo e in vitro, os testes de cisalhamento, tração e compressão, a extensometria e a interferometria holográfica, por exemplo.

Devido ao crescente número de pesquisas, o método dos elementos finitos tornou-se uma ferramenta valiosa para avaliar a distribuição de tensões no sistema prótese/implante e sua inter-relação com o osso de suporte. Por meio do método de elementos finitos, um modelo computacional de geometria complexa pode ser gerado e nele podem ser simuladas as condições obtidas *in vivo*. Entretanto, a interferometria holográfica vem complementar e validar essa metodologia, pois é uma técnica experimental de análise de tensões. As metodologias combinadas levam a um melhor conhecimento da mecânica envolvida no sistema, o que resultará provavelmente na diminuição da taxa de insucesso dos implantes, tornando-se de grande valia na compreensão, no planejamento e instalação dos implantes osseointegráveis.

CAPÍTULO 1

Mechanical behavior of dental implants in different positions in the rehabilitation of the anterior maxilla

Manuscrito submetido para publicação - The Journal of Prosthetic Dentistry

Mechanical behavior of dental implants in different positions in the rehabilitation of the anterior maxilla

Cássia Bellotto Corrêa, DDS, PhD Student[a], Rogério Margonar DDS, MSc, PhD[b], Pedro Yoshito Noritomi DDS, MSc, PhD[c], Luis Geraldo Vaz, BSc, PhD, Associate Professor[d]

[a]Doctoral student, Departmento de Diagnóstico e Cirurgia, Faculdade de Odontologia de Araraquara, Univ Estadual Paulista, UNESP, SP, Brazil.

[b]Associate Professor, Department of Health Sciences, Implantology Post Graduation Course, School of Dentistry University Center of Araraquara-UNIARA, Araraquara, SP, Brazil.

[c]Mechanical Engineer, Researcher, Center for Information Technology Renato Archer, Campinas, SP, Brazil.

[d]Associate Professor, Department de Materiais Dentários e Prótese, Faculdade de Odontologia de Araraquara, Univ Estadual Paulista, UNESP, SP, Brazil.

Corresponding author:
School of Dentistry at Araraquara, Univ Estadual Paulista
Rua: Humaitá, 1680 FOAr-UNESP, 14801-093, Araraquara, SP - Brazil
Fone: +55 16 33016420
e-mail: cassiabcorrea@hotmail.com

Title

Mechanical behavior of dental implants in different positions in the rehabilitation of the anterior maxilla

Abstract

Statement of problem. In dental rehabilitations involving implants, the number of implants is sometimes smaller than the number of lost teeth. This fact can affect the biomechanical behavior and success of the implants.

Purpose. The aim of this study was to investigate the mechanical behavior of different implant positions in the rehabilitation of the anterior maxilla.

Materials and methods. Three dimensional models of the maxilla were created based on computed tomography images for three different anterior prosthetic rehabilitations: group IL, the implants were placed in the lateral incisor positions with pontics in the central incisor positions, group IC the implants were in the central incisor positions with cantilevers in the lateral incisor positions, and group ILIC one implant was in a lateral incisor position and one was in a central incisor position, with a pontic and a cantilever in the remaining positions. A 150 N load was distributed and applied at the center of the palatal surface of each tooth at a 45 degrees angle to the long axis of the tooth. The resulting stress/strain distribution was analyzed for each group.

Results. The lowest displacement of the prosthetic structure was observed in group IC, although the same group exhibited the largest displacement of the bone tissue. In the bone tissue, the von Mises stress was mainly observed in the cortical bone in all groups. The maximum value of the von Mises stress shown in the cortical tissue was 35 MPa in the implant neighboring the cantilever in group ILIC. The maximum Von Mises stress in the trabecular bone was 3.5 MPa.

Conclusion. The prosthetic configuration of group IC limited the displacement of the prosthetic structure but led to greater displacement of the bone structure. The use of a cantilever increased the stress concentration in the implant and in the bone structure adjacent to the cantilever under the conditions studied here.

Clinical Implications. Overload is the most common mechanism of failure in dental implants. The appropriate positioning of dental implants and knowledge of the distribution of strain/stress can help to avoid mechanical overload, prevent failures and ensure the success of dental implants.

INTRODUCTION

The high rate of success of rehabilitation with dental implants and implant-supported prostheses has not only increased the number of indications for functional recovery but also increased esthetic demands made by patients and clinicians.[1-4]

To obtain satisfactory functional and esthetic results, it is essential to achieve appropriate three-dimensional positioning of the implants. The buccopalatal positioning of the implant is mainly determined by prosthetic planning and by the anatomical occlusal conditions of the patient. The implant should be placed in the center of the crest bone by visualizing an imaginary line connecting the incisal of the adjacent teeth[5] and also must preserve the bone thickness of approximately 2 mm on the buccal and palatal bone walls.[4]

Furthermore, the adequate buccopalatal and mesiodistal positioning of the implant is also extremely important. Thus, the implant must be kept at a distance of 2 mm from the adjacent teeth and 3 mm from other implants to ensure the preservation of the crestal bone.[6]

However, for many patients, the mesiodistal distance in the arch is not sufficient to allow the installation of an implant for each lost tooth; in this case, implants are associated with suspended prosthetic elements to optimize spacing and reduce esthetic damage.[7]

In some cases, the use of suspended elements may overload the implant and lead to bone loss.[8-13] This work uses the Finite Element Method (FEM) to study the biomechanical behavior of a system missing four anterior teeth with only two pillars implanted in the maxilla.

FEM is a technique in which a physical prototype can be studied by the representation through virtual mathematical models.[14-15] The geometrical model is subdivided into small elements that are interconnected by nodes, generating what is called a finite element mesh.[16-17] This model can be solved by software implementing a large number of mathematical equations to simulate the specific structure.[18-21] In the current work, a maxilla was geometrically modeled based on real computed tomography (CT scan) data and its physical properties were found based on the literature, making it possible to determine the stress and strain resulting from the application of a force in the finite element model. The null hypothesis was that no significant difference among the groups would be found in the mechanical behavior of dental implants.

Materials and Methods

To represent the working case, a three dimensional model (3D) of the anterior maxilla was created with an intercanine distance of 27.3 mm based on a database of CT scans available at the Center for Information Technology Renato Archer (CTI, Campinas, Brazil).

This anatomical model was generated by combining several anatomical maxillar structures averaged over the database of CT scan.[22] Then, the model was used to construct a simplified geometric model using the software Rhinoceros 4.0 SR8 (McNeel, Seattle - Wash).

The geometric models of the abutments, screws and internal hexagon implants were constructed in the same modeling software used for the maxilla based on images provided by the manufacturer (Neodent Ltda, Curitiba, Paraná, Brazil).

A specific model was created for each group:

Group IL - Implants in the lateral incisor positions and pontics in the central incisor positions.

Group IC - Implants in the central incisor positions and cantilevers in the lateral incisor positions.

Group ILIC - One implant in a lateral and one in a central incisor position, with a pontic and a cantilever in the other positions.

Each model was imported by the FEMAP software version 10.1.1 for pre-processing, and all the information required to characterize the biomechanical situation was provided.

The mechanical properties of the structures were characterized in terms of the elastic modulus (E) and Poisson ratio (nu), with the values shown in Table 1 given by the manufacturer (Neodent Ltda) or available in the literature. All materials were considered isotropic, homogeneous, linear and elastic.[26,27]

Table 1: Material properties used in finite element analysis studies of dental implants.

	Young modulus (E), Mpa	**Poisson ratio (nu)**	**Reference**
Cortical Bone	13.7	0.3	Meijer et al. 1993[23] Menicucci et al. 2002[24]
Trabecular Bone	1.3	0.3	Meijer et al. 1993[23] Menicucci et al. 2002[24]
Implant (Titanium c.p. grade 4)	103	0.361	Manufacturer
Abutment and Abutment Screw (Titanium Alloy)	105	0.361	Manufacturer
Prosthetic Structure (Ni Cr Alloy)	210	0.28	Anusavice KJ, 2003[25]

The mesh was constructed with tetrahedral elements with 10 nodes.

A total load of 150 N was distributed and applied at the center of the palatal surface of each tooth at a 45degrees angle to the long axis to simulate functional occlusion (Figure 1). [28-33]

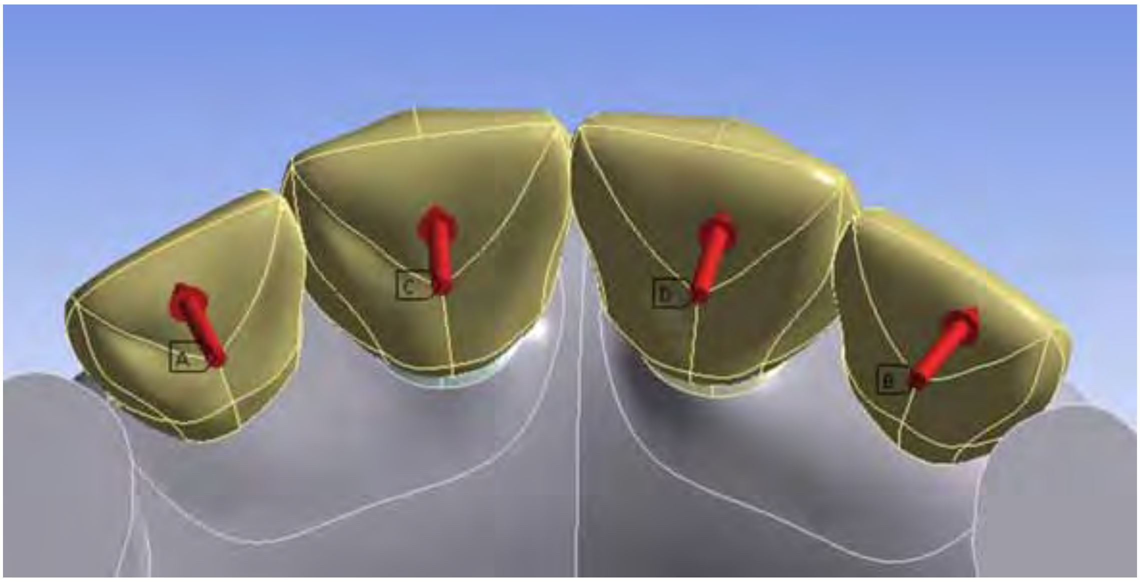


Fig. 1. Representation of load application.

The bone-implant interface was considered bonded, and all screw threads were filled with bone to simulate an osseointegrated implant. The interface between the implant and the abutment and between the implant and the screw were considered common contacts.

The mesh contained an average of 1 245 854 nodes and 806 171 elements.

Results

In group IL, the peak displacement in the prosthetic structure is 0.095 mm in the region of the central incisors, with less displacement observed when approaching the cervical region of the lateral incisors and the ends of the maxilla (Figure 2A). The stress dissipation in the maxilla is symmetric. The peak of the displacement is approximately 0.01 mm from the buccal region of the implant toward the buccal and mesial region in the maxillary bone (Figure 3A).

In group IC, the peak displacement in the prosthetic structure is lower than in the other groups at approximately 0.079 mm, is homogeneously distributed and ends at the maxilla (Figure 2B). However, this group features the highest displacement of the maxillary bone, with a maximum displacement about of 0.013 mm (Figure 3B).

In group ILIC, the maximum displacement in the prosthesis region is similar to that of group IL, but the peak stress is shifted to the lateral incisor, corresponding to the cantilever side (Figure 2C). The distribution of displacement in the maxillary bone is asymmetric as is the prosthesis, with the highest displacement observed in the region adjacent to the implant in the central incisor, with a peak displacement slightly higher than that in group IL (Figure 3C).

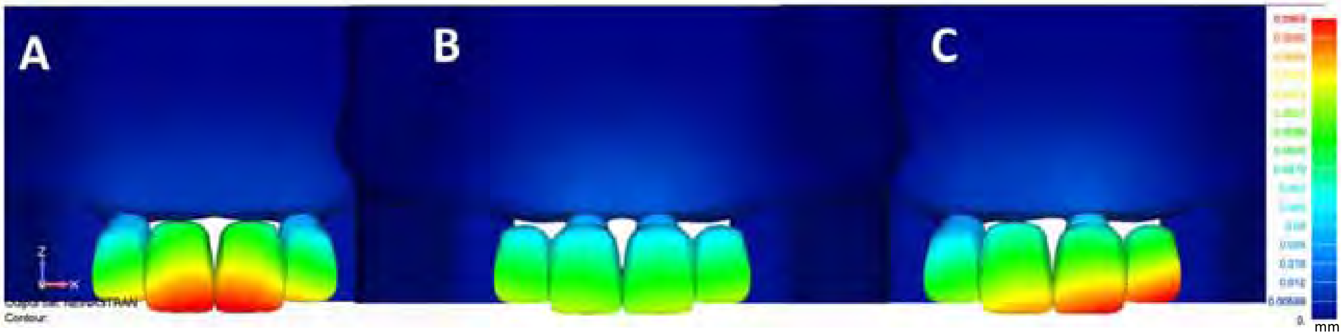


Fig. 2. Front view of displacement of prosthetic structure of A, group IL; B, group IC; C, group ILIC.

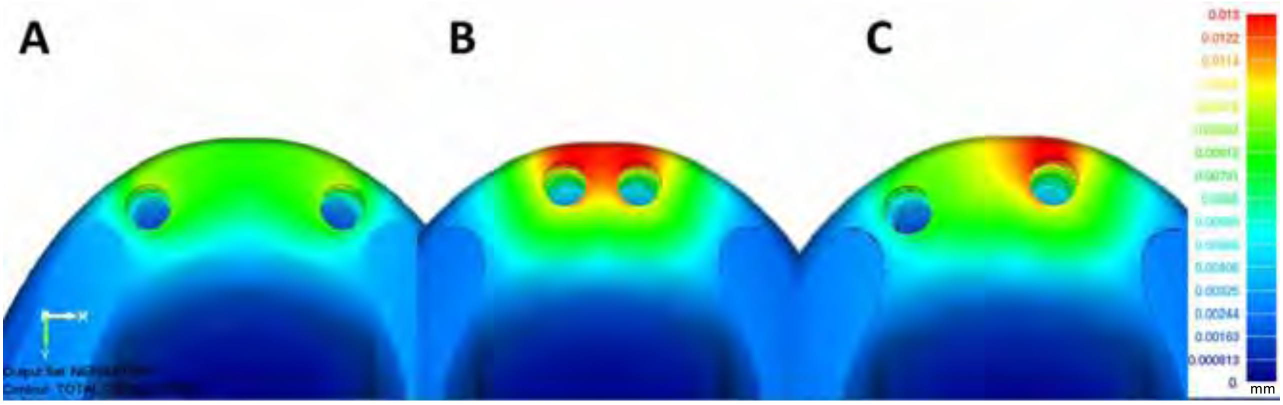


Fig. 3. Occlusal view of displacement of maxillary bone of 3A, group IL; 3B, group IC; 3C, group ILIC.

Analyzing the maximum principal stress in the cortical bone, the tensile stresses are located in the mesiodistal region, the site of implant insertion, with values ranging from 14 MPa to 18 MPa (Figure 4A).

Analyzing the minimum principal stress in the cortical bone, the compressive stress in group IL could be located on the buccal wall where the implants were inserted. These compressive stresses range in magnitude from -20 MPa to -22 MPa (Figure 4D).

In group IC, the tensile stress is located in the mesial palatal and distal region of the implant insertion site. The tensile stresses are between 9 MPa and 12 MPa (Figure 4B).

The compressive stress is also located in the buccal region of the implants, with values ranging from -12 MPa to -15 MPa (Figure 4E).

In group ILIC, the tensile stress is located in the mesial-palatal and distal incisor and in the distal portion of the implant in the lateral incisor implant, with values of 15 MPa to 18 MPa. Stresses greater than 20 MPa likely result from premature contact with the mesh, viewed as dappled areas (Figure 4C). The compressive stress is also located in the buccal region of the implant insertion site, with values between -15 MPa and -20 MPa and in the region of lateral implant and values between -20 and -30 in the region of central implant (Figure 4F).

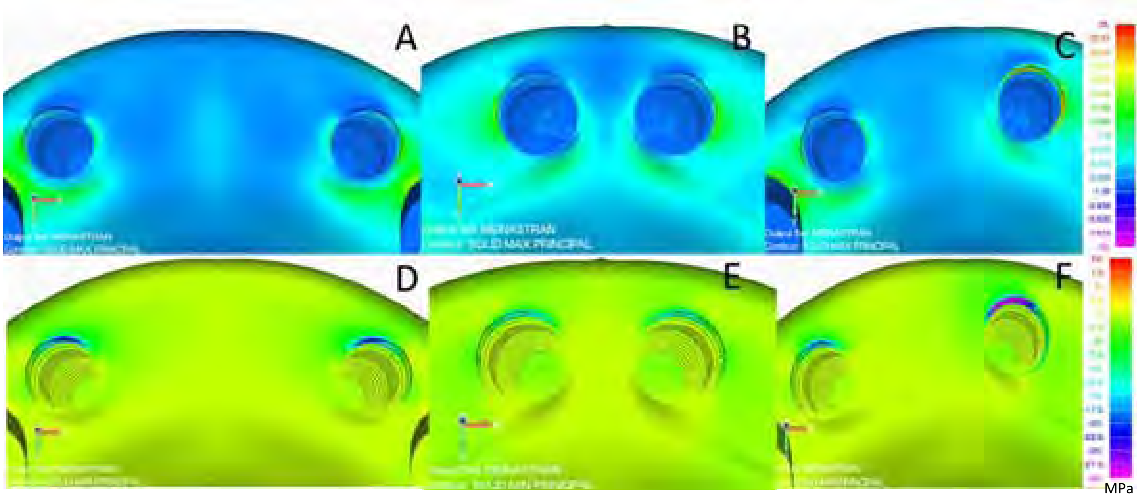


Fig. 4. Results of Maximum Principal Stress; occlusal view of cortical bone of 4A, group IL; 4B, group IC; 4C, group ILIC. Results of Minimum Principal Stress; occlusal view of cortical bone of 4D, group IL; 4E, group IC; 4F, group ILIC.

Analyzing the maximum principal stress in the trabecular bone, the tensile stresses in all groups are located in the buccal wall (region near to cortical region) and region of apex of the implants, with values ranging from 2 MPa to 3 MPa (Figure 5A, 5B and 5C).

Analyzing the minimum principal stress in the trabecular bone, the compressive stress in all groups are located in the buccal wall (region near to cortical region) and region of apex of the implants. These compressive stresses range in magnitude from -2MPa to -3MPa (Figure 5D, 5E and 5F).

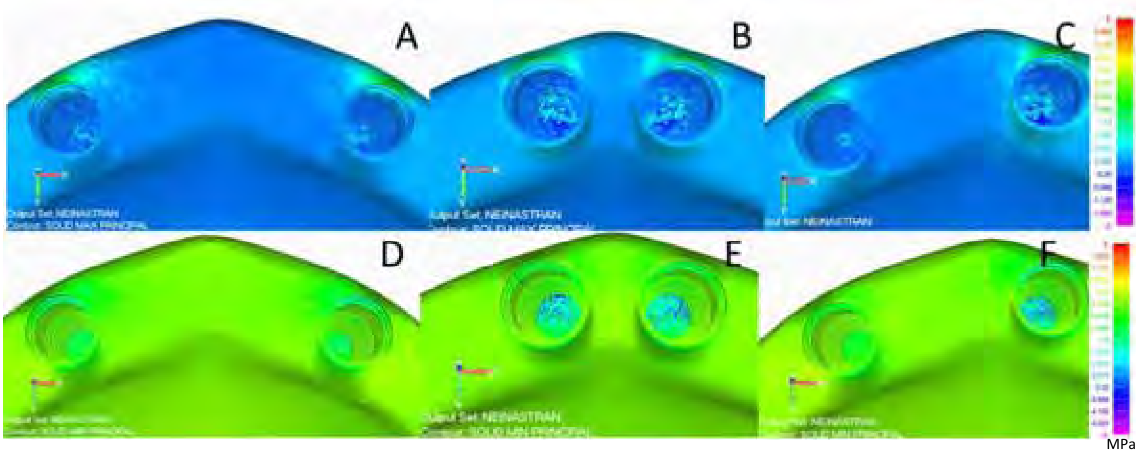


Fig. 5. Results of Maximum Principal Stress; occlusal view of trabecular bone of 5A, group IL; 5B, group IC; 5C, group ILIC. Results of Minimum Principal Stress; occlusal view of trabecular bone of 5D, group IL; 5E, group IC; 5F, group ILIC.

In this work, the von Mises stress was used as a parameter to view the mechanisms of stress dissipation and to compare among the models, not as a parameter for the identification of critical regions of stress.

Analyzing the von Mises stress in the sagittal section of the left implants, the stress was found to be mostly dissipated by the cortical maxillary bone. The highest von Mises stress was observed in group ILIC (note the scale), followed by group IL and group IC (Figures 6A, 6B and 6C).

The view of the trabecular region of the left implants verifies that the maximum stress value 3.5 MPa, with a similar value observed for the implants on the right side, much lower than the stress observed in the cortical region. The areas of concentration of stress in trabecular bone are located in the cervical region, adjacent to the area of greatest concentration of stress on the cortical bone. The stress is directed toward the apex from the buccal wall (Figure 6D, 6E and 6F).

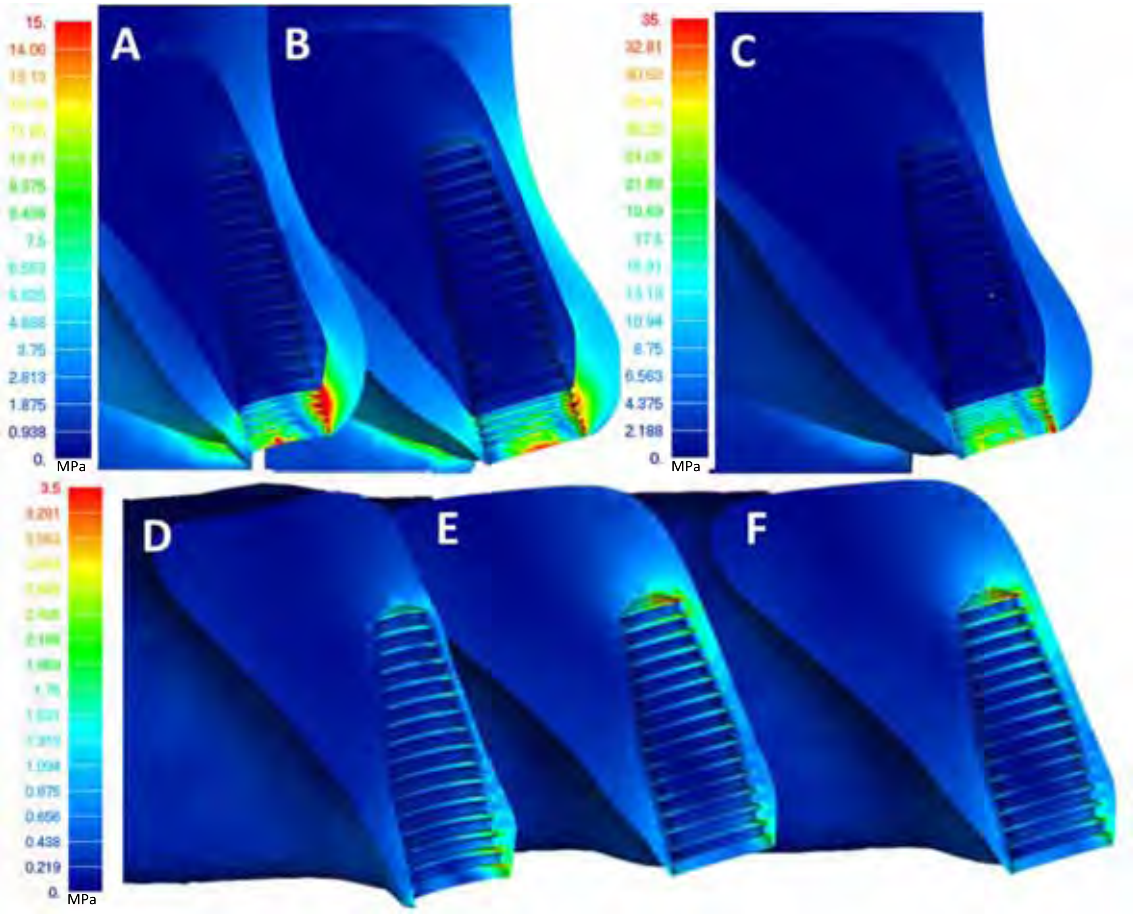


Fig. 6. Sagittal view of Von Mises Stress of cortical and trabecular bone of left implants, 6A, group IL; 6B, group IC; 6C, group ILIC. Sagittal view of Von Misses Stress of trabecular bone 6D, group IL; 6E, group IC; 6F, group ILIC.

The right implants have similar stress distributions among the three groups. The lowest values are observed in the implants of group ILIC. The stress in all groups is highest in the cervical region. Similarly to the left implants, the stress is directed toward the apex from the buccal wall (Figure 7A, 7B, 7C, 7D, 7E and 7F).

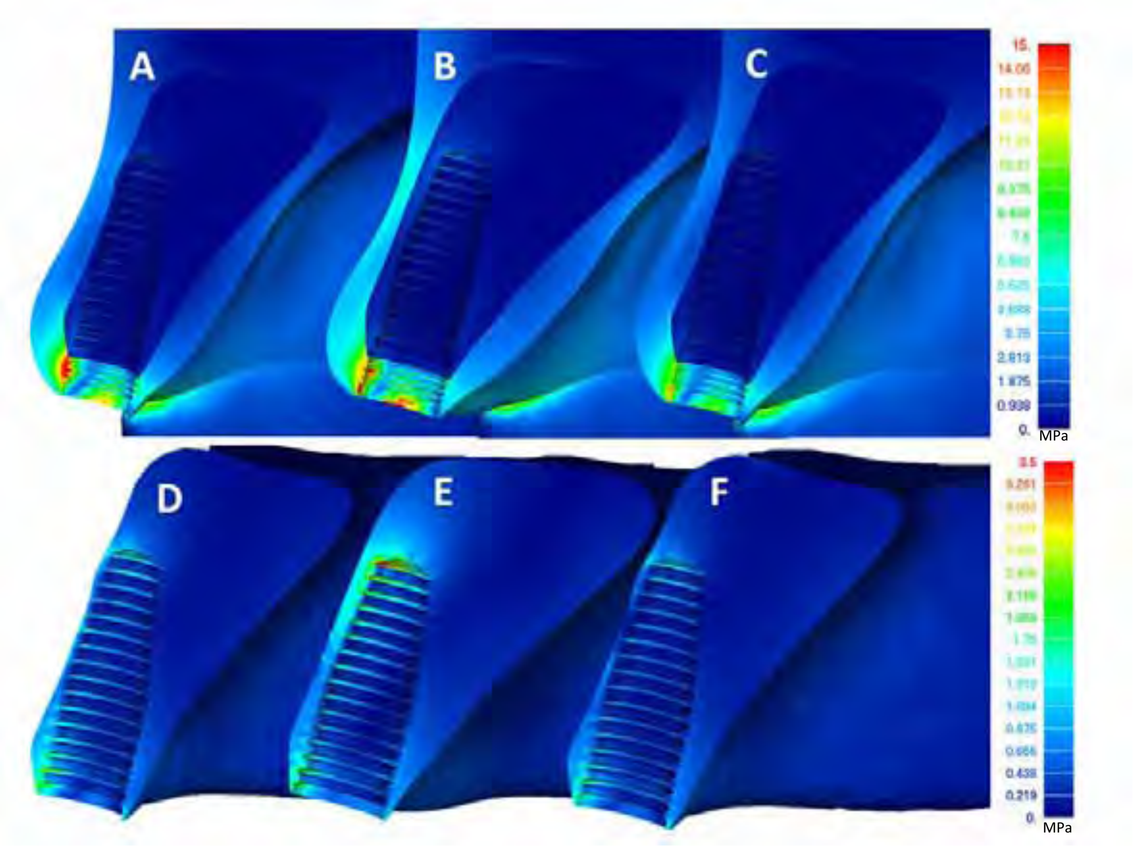


Fig. 7. Sagittal view of Von Mises Stress of cortical trabecular bone of right implants, 7A, group IL; 7B, group IC; 7C, group ILIC. Sagittal view of Von Misses Stress of trabecular bone 7D, group IL; 7E, group IC; 7F, group ILIC.

Analyzing the von Mises stress in dental implants, in all groups, considerable stress exists in the buccal region of the internal hexagon and in the buccal region of the platform implant (Figure 8A, 8B and 8C).

Groups IC and IL show similar stress distributions, predominantly on the inner side of the hexagon of the implants, with group IC showing fewer areas of maximum stress (Figure 8A and 8B). In group ICIL, the stress is mainly distributed at the implant plateau and on the inner side of the hexagon of the implant (Figure 8C).

The stress is predominantly distributed in the first threads of the implant, corresponding to the region of insertion of the implant in the cortical bone (Figure 8A, 8B and 8C).

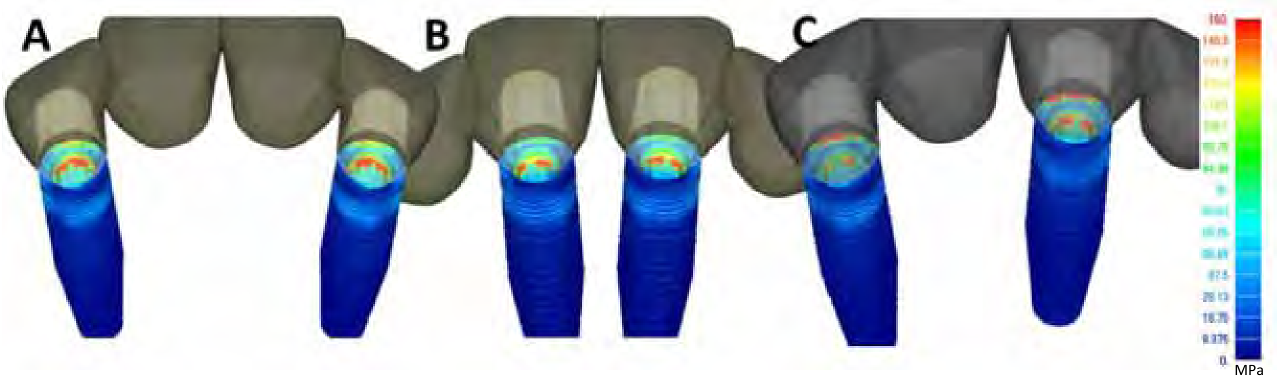


Fig. 8. Von Mises Stress of dental implants 8A, group IL; 8B group IC; 8C group ILIC.

Analyzing the von Mises stress in the abutments, a significant area of stress is observed in the buccal region for all abutments, more specifically on the plateau of the abutment and at the edges of the hexagon. In the posterior region, the stress is mainly located at the top of the hexagon. The bending motion of the prosthetic abutment is characterized in Figure 9A, 9B and 9C.

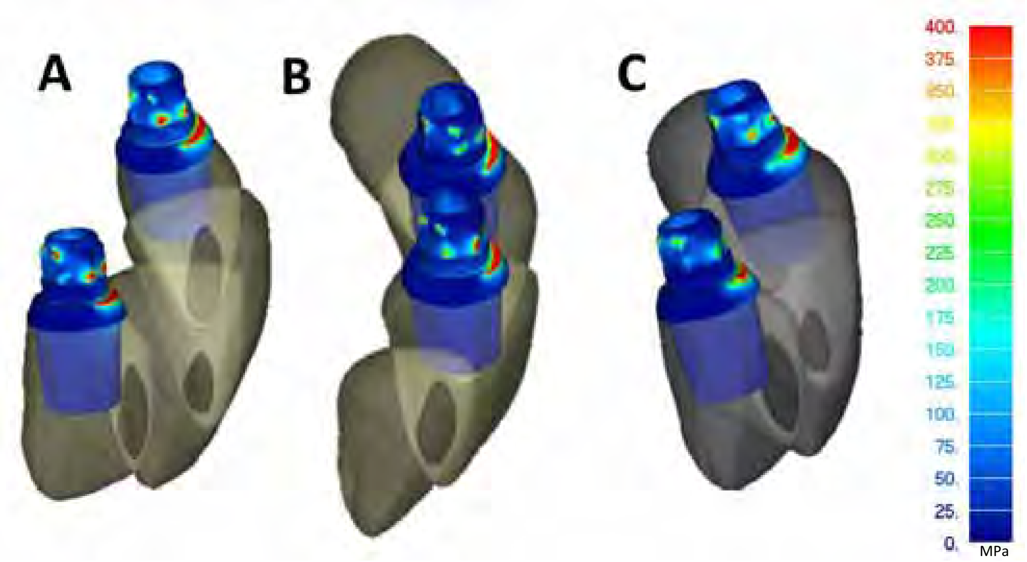


Fig. 9. Von Mises Stress of abutments, 9A, group IL; 9B group IC; 9C group ILIC.

Analyzing the von Mises stress in the abutment screws in a frontal view, the stress appears to be concentrated in the region in which the head screw makes contact with the abutment, propagates along the screw shaft and increases again in the third screw thread. A higher value of stress is observed in the screw of group IL, followed by group ILIC and then group IC (Figure 10A, 10B and 10C).

The distribution of stress on the palatal side is similar to the distribution on the buccal side (front view), but the values are higher on the palatal side. Higher values of stress in the abutment screws are again observed in Group IL followed by Group ILIC and then Group IC (Figure 10D, 10E and 10F).

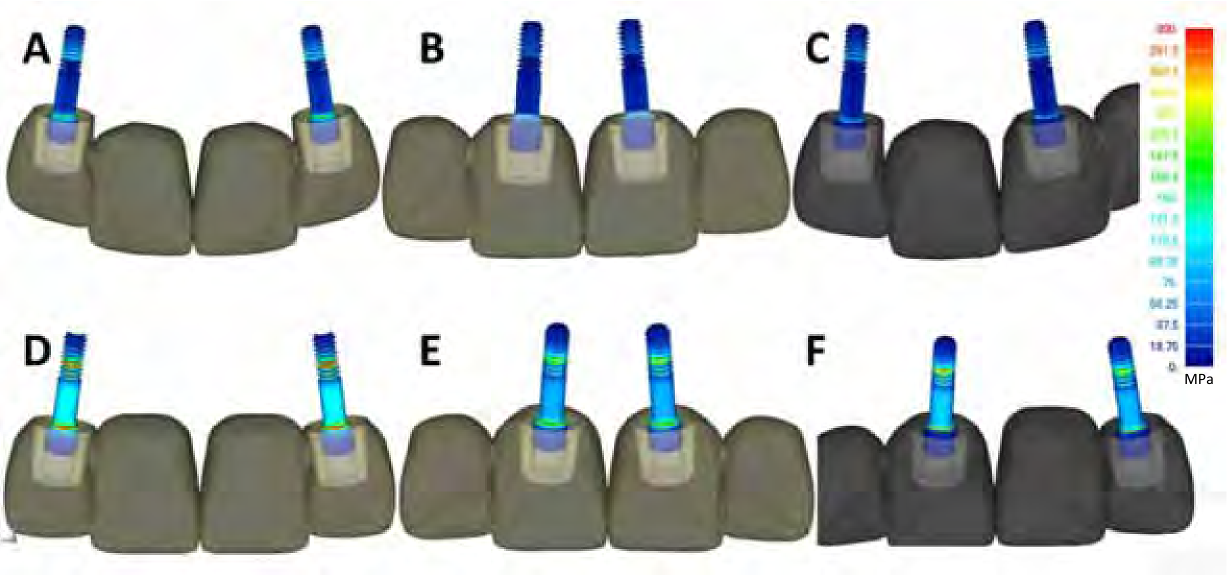


Fig. 10. Front view of Von Mises Stress in abutment screws of 10A, group IL; 10B, group IC; 10C group ILIC. Posterior view of Von Mises Stress in abutment screws, 10D, group IL; 10E, group IC; 10F group ILIC.

Discussion

The null hypothesis of this study was rejected based on the difference among the mechanical behavior of dental implants in different positions studied. The success of treatment with dental implants depends mostly on the biomechanical behavior of the implant and its components.[15] Application of an overload to the prosthetic structure causes stresses, strains and displacement in the implant system, which can affect the bone remodeling process around implants.[8,9,12,13,21]

The lowest displacement values of the bone structure are observed in group IL in which the implants are fixed in the region of the lateral incisors. The displacement results presented in this study are consistent with those presented in the study by Hsu et al.[21] that used FEM to evaluate the stress/strain distribution in bone around maxillary implants under 3 different off-axis loading conditions.

Although the groups have different stress distributions, from the biomechanical point of view, there are no contraindications to the use of any of the conditions studied here because the maximum tensile and compressive stresses in bone tissue found in this study are far below the overload and likely to fail limits (above 100 MPa and 170 MPa for tensile and compressive stress, respectively) according to the classification adopted in the study by Bozkaya et al.[17]

In the trabecular portion, Principal stresses of -2MPa to -3MPa are observed, mainly at the apex of the implant and in the cervical region of implant, which is near the highest stress concentration in the cortical region. This result agrees well with the finding by Martin et al.[11] who reported a stress in trabecular bone ranging from 2 MPa to 5 MPa.

While group IC exhibits the lowest displacement of the prosthetic structure, indicating that this configuration facilitates the stability and preservation of the prosthetic structure, this configuration also leads to a higher displacement of the bone tissue adjacent to the implant compared with other groups.

The propagation of stress in the implant body follows the same pattern of propagation in all studied groups, which means that all the stress was mainly focused on the first threads of the implants, the region corresponding to the cortical bone. These results are in agreement with studies performed by, Natali et al.[14], Bonnet et al.[22] and Chun et al.[26] and with in vivo studies demonstrating bone loss, particularly during the first year after implantation.[9,10]

In the region of the internal hexagon of the implant and the abutment, the stress distribution patterns are similar among all three groups. This distribution pattern occurs in response to a bending movement of the prosthesis in response to the direction of the applied force. The implant configuration in group IC results in lower levels of stress in the internal region of the hexagon and on the implant platform. The pattern of distribution of stress inside the implant and abutment is according to the study by Pessoa et al.[31]

Group IC exhibits the lowest values of stress in the prosthetic screw in both views (buccal and palatal). The location of the stress in the screw is similar to that found in a study by Wang et al.[27], with the highest values of von Mises stress observed in the region of the top of the screw shaft, the bottom of the screw shaft and the top of the screw thread.

The present study assumed an occlusal force of 150 N. The force was applied to the cingulum of each tooth at a 45 degrees angle to the long axis of the tooth to simulate mastication.[30,32,33] For dentate humans, the bite force varies among individuals and among different regions of the dental arch. Regalo et al.[29] investigated the maximum masticatory force in indigenous and white Brazilian individuals with complete dentition; normal occlusion; no neurological, psychiatric or movement disorders; no reports of toothaches; satisfactory periodontal health; no large facial skeletal alterations (typical Class II and Class III individuals); and no previous treatments using occlusal splints. The masticatory force in the region corresponding to the incisor was 194 N for the indigenous individuals and 117 N for the white individuals.

This variation of the force of occlusion can be related to many factors including age, sex, parafunction, and muscle size.

There are inherent limitations in any FEM study that limit the extrapolation of the results to clinical situations. The structures in the model were all assumed to be homogenous, isotropic and linear elastic. Additionally, the model considered 100% osseointegration, which does not necessarily represent the actual clinical situation.[21] The interfaces between the prosthetic structure and the abutment or abutment screw and implant were assumed to be bonded.

However, FEM is an excellent method to obtain detailed quantitative data, and it enables accurate visualization of the stress distribution and displacement in models of complex geometries such as the maxilla.[16,20] The use of a fine mesh (larger amount elements) is important to enable an accurate FEM model.[21] Thus, the finite element method is able to provide information that is not available from clinical and experimental studies.[18]

The outcomes of this study, with its limitations kept in mind, can help clinicians to comparatively evaluate the performance of different implant distributions and prosthetic configurations.

For future studies the three-dimensional model validation by an experimental technique will have great relevance and importance to ensure correlation between the results obtained with the finite element analysis with the same structure and behavior in vivo.

Conclusions

The prosthetic configuration in which the implants are located in the lateral incisors limited the displacement of the prosthetic structure but led to greater displacement of the bone structure than other configurations.

Regardless of the positioning of the implants, the cortical part of the bone receives and dissipates the most of the stress. However, the cortical and trabecular bone were not overloaded for any of the groups.

The use of a cantilever led to a greater stress concentration in the implant and bone structure adjacent to the cantilever; however, these values are below the limit considered to represent an overload.

Acknowledgements

The authors wishes to acknowledge the scholarships from CAPES (Committee for Postgraduate Courses in Higher Education), the FAPESP (Research Support Foundation of São Paulo State) grant number 2010/01059-0, the CTI (Centro de Tecnologia da Informação Renato Archer) and the Neodent® implant system for providing the implant CAD models.

REFERENCES

1. Salama H, Salama M, Garber D, Adar P. Developing optimal peri-implant papillae within the esthetic zone: guided soft tissue augmentarion. J Esthet Dent 1995;7:125-9.

2. Binon PP. Implants and components: entering the new millennium. Int J Oral Maxillofac Implants 2000;15:76-94.

3. Gastaldo JF, Cury PR, Sendyk WR. Effect of the vertical and horizontal distances between adjacent implants and between a tooth and an implant on the incidence of interproximal papilla. J Periodontol 2004;75:1242-6.

4. Grunder U, Gracis S, Capelli M. Influence of 3-D bone to implant relationship on esthetic. Int J Periodontics Restorative Dent 2005;25:113-9.

5. Chiche FA, Leriche MA. Multidisciplinary implant dentistry for improved aesthetics and function. Pract Periodontics Aesthet Dent 1998;10:177-88.

6. Novaes Jr AB, Barros RRM, Muglia VA, Borges GJ. Influence of interimplant distances and placement deph on papilla formation and crestal resorption: A clinical and radiographic study in dogs. J Oral Implantol 2009;35:18-27.

7. Tarnow DP, Cho SC, Wallace SS. The effect of inter-implant distance on the height of inter-implant bone crest. J Periodontol 2000;71:546-9.

8. Naert I, Duyck J, Vandamme K. Occlusal overload and bone/implant loss. Clin Oral Implants Res 2012;23:95-107.

9. Skalak R. Biomechanical considerations in osseointegrated prostheses. J Prothet Dent 1983;49:843-8.

10. Isidor F. Loss of osseointegration caused by occlusal load of oral implants. A clinical and radiographic study in monkeys. Clin Oral Implants Res 1996;7:143-52.

11. Barbier L, Vander Sloten J, Krzesinski G, Schepers E, Van Der Perre G. Finite element analysis of non-axial loading versus axial loading oral implants in the mandible of the dog. J Oral Rehabil 1998;25:847-58.

12. Martin RB, Burr DB, Sharvey NA. Skeletal tissue mechanics. 1st ed. New York: Springer; 1998. p. 71-2.

13. Duyck J, Ronald HJ, Van Oosterwyck H, Naert I, Vander Sloten J, Ellingsen JE. The influence of static and dynamic lading on marginal bone reactions around osseointegrated implants: an animal experimental study. Clin Oral Impl Res 2001;2:207-18.

14. Mish CE, Suzuki JB, Misch-Dietsh FM, Bidez MW. A positive correlation between occlusal trauma and peri-implant bone loss: literature support. Implant Dent 2005;14:108-16.

15. Natali AN, Carniel EL, Pavan PG. Investigation of bone inelastic response in interaction phenomena with dental implants. Dent Mater 2008;24:561-9.

16. Djebbar N, Series B, Bouiadjra BB, Benbarek S, Drai A. Analysis of the effect of load direction on the stress distribution in dental implant. J Mater Design 2010;31:2097-2101.

17. Holmgren EP, Seckinger RJ, Kilgren LM, Mante F. Evaluating parameters of osseointegrated dental implants using finite element analysis – a two-dimensional comparative study examining the effect of implant diameter, implant shape, and load direction. J Oral Implantol 1998;24:80-8.

18. Bozkaya D, Muftu S, Muftu A. Evaluation of load transfer characteristics of five different implants in compact bone at different load levels by finite elements analysis. J Prosthet Dent 2004;92:523-30.

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

20. Kitagawa T, Tanimoto Y, Odaki M, Nemoto K, Aida M. Influence of implant/abutment join designs on abutment screw loosening in a dental implant system. J Biomed Mater Res B Appl Biomater 2005;75:457-63.

21. Van Staden RC, Guan H, Loo YC. Application of the finite element method in dental implant research. Comput Methods Biomech Biomed Engin 2006; 9:257-70.

22. Hsu ML, Chen FC, Kao HC, Cheng CK. Influence off-axis loading of an anterior maxillary implant: a 3-dimensional finite element analysis. Int J Oral Maxillofac Implants 2007;22:301-9.

23. Bonnet AS, Postaire M, Lipinski P. Biomechanical study of mandible bone supporting a four-implant retained bridge: finite element analysis of the influence of bone anisotropy and foodstuff position. Med Eng Phys 2009;31:806-15.

24. Menicucci G, Lorenzetti M, Pera P, Preti G. Mandibular implant-retained overdenture: finite element analysis of two anchorage systems. Int J Oral Maxillofac Implants 1998;13:369-76.

25. Meijer GJ, Starmans FJ, de Putter C, van Blitterswijk CA. The influence of a flexible coating on the bone stress around dental implants. J Oral Rehabil 1995;22:105-11.

26. Anusavice KJ. Phillips’science of dental materials. 11th ed. St. Louis: Elsevier; 2003. p. 93-4.

27. Chun HJ, Cheong SY, Han JH, Heo SJ, Chung JP, Rhyu IC, Choi YI, et al. Evaluation of design parameters of osseointegrated dental implants using finite element analysis. J Oral Rehabil 2002;29:565-74.

28. Wang RF, Kang B, Lang LA, Razzoog ME. The dynamic natures of implant loading. J Prosthet Dent 2009;101:359-71.

29. Morneburg TR, Prosche AP. Measurement of Masticatory Forces and Implant Loads: A Methodologic Clinical Study. Int J Prosthodont 2002;15:20-7.

30. Regalo SCH, Santos CM, Vitti M, Regalo CA, Vasconcelos PB, Mestriner Jr W, et al. Evaluation of molar and incisor bite force in indigenous compared with white population in Brazil. Arch Oral Biol 2008;53:282-6.

31. Eraslan O, Aykent F, Yucel MT, Akman S. The finite element analysis of the effect of ferrule height on stress distribution at post-and-core-restored all-ceramic anterior crowns. Clin Oral Invest 2009;13:223-27.

32. Pessoa RS, Muraru L, Marcantoni Jr EM, Vaz LG, Sloten JV, Duycky J, Jaecques. 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;12:219-34.

33. Alikhasi M, Siadat H, Geramy A, Hassan-Ahangari A. Stress distribution around maxillary anterior implants as a factor of labial bone thickness and occlusal load angles: a 3D-finite-element analysis. J Oral Implantol 2011 Jul 25. [Epub ahead of print].

34. Freitas AC Jr, Bonfante EA, Rocha EP, Silva NRFA, Marotta L, Coelho PG. Effect of implant connection and restoration design (screwed vs. cemented) in reliability and failure modes of anterior crowns. Eur J Oral Sci 2011;119:323-30.

CAPÍTULO 2

Comparative analysis by finite elements method for internal and external hexagon and Morse-taper implants in the prosthetic rehabilitation of the anterior maxilla

Manuscrito redigido nas normas da revista científica Clinical Implant Dentistry and Related

Comparative analysis by finite elements method for internal and external hexagon and Morse-taper implants in the prosthetic rehabilitation of the anterior maxilla

Cássia Bellotto Corrêa, DDS, PhD Student[a], Rogério Margonar DDS, MSc, PhD[b], Pedro Yoshito Noritomi DDS, MSc, PhD[c], Luis Geraldo Vaz, BSc, PhD, Associate Professor[d]

[a]Doctoral student, Department de Cirurgia e Diagnóstico, Faculdade de Odontologia de Araraquara, Univ Estadual Paulista, UNESP, SP, Brazil.

[b]Associate Professor, Department of Health Sciences, Implantology Post Graduation Course, School of Dentistry University Center of Araraquara-UNIARA, Araraquara, SP, Brazil.

[c]Mechanical Engineer, Researcher, Center for Information Technology Renato Archer, Campinas, SP, Brazil.

[d]Associate Professor, Departamento de Materiais Dentários e Prótese, Faculdade de Odontologia de Araraquara, Univ Estadual Paulista, UNESP, SP, Brazil.

Corresponding author:
School of Dentistry at Araraquara, Univ Estadual Paulista
Rua: Humaitá, 1680 FOAr-UNESP, 14801-093, Araraquara, SP - Brazil
Fone: +55 16 33016420
e-mail: cassiabcorrea@hotmail.com

Comparative analysis by finite elements method for internal and external hexagon and Morse-taper implants in the prosthetic rehabilitation of the anterior maxilla

ABSTRACT

Purpose: The purpose of the present study was to evaluate the mechanical behavior of the anterior rehabilitation of the maxilla comparing three types of connection: external hexagon, internal hexagon and Morse taper.

Material and Methods: It was created a 3D model, based on CT images of the maxilla. The prosthetic rehabilitation was simulated with three connections implant systems. All the rehabilitations used implants that were in the position of the lateral incisors with the pontic located in the central incisors. The applied load of 150N was distributed at the center of the palatal surface of each tooth, with an angle of 45° related to the tooth long axis. The distribution of strain/stress was analyzed in all groups.

Results: The lower displacement of prosthetic structure was found in the rehabilitation simulated with the external hexagon connection. The more significant results were found in the prosthetic screw. The prosthetic screw of internal hexagon and external hexagon had the von Mises stress twice the value that the one shown by prosthetic screw in the Morse taper connection.

Conclusion: The implant connection design has influence in the distribution and intensity of stress/strain on the prosthesis/implant system in the studied cases. The Morse taper connection was the connection type that showed the smallest values of stress in the abutment screw and can be favorable to be used in a cemented prosthetic rehabilitation, avoiding the abutment screw loosening.

KEYWORDS: finite element analysis, prosthetic connection, dental implant, maxilla

INTRODUCTION

The aesthetic and functional rehabilitation with dental implants has become very common. Clinical observations have indicated that the major causes of implant failure are: deficient osseointegration, peri-implantitis and mechanical complications[1]. Among the biomechanical problems, abutment screw loosening, screw fracture, abutment rotation and fracture are the most reported[2].

From the biomechanical point of view, the connection between the implant and abutment has influence on the stress and displacement of the entire system and can be used to ensure longevity to the treatment and reduce the peak bone interface stresses and strains.

The finite elements method (FEM) has been applied to the dental implant field to predict stress distribution patterns in the implant bone interface not only in comparisons of shapes of implants (cylindrical or conical)[3], diameters[4] and lengths[5], but also to model various clinical scenarios[6] and prosthesis designs[7-9].

FEM is a numerical method that enables the calculation of stress, displacement and deformation based on the evaluation of the mechanical behavior equation of materials. The method has the advantage of solving complex structural problems, such as the maxillary structure, by dividing the complex geometries of the structure in much smaller domains (elements), to be able to calculate the result of applyed force on this structure[10].

The aim of this study was to comparatively evaluate the biomechanical behavior of three types of prosthetic connections (external and internal hexagon and Morse taper) in anterior rehabilitation of the maxilla.

MATERIAL AND METHODS

In this study, three connection systems were compared: external and internal hexagon, and Morse-taper. The diameters and heights of implants were the same for all connection types. The same type of component (conical mini pilar) was used in all of them.

The finite element model

The anatomical model was constructed by the software Rhinoceros v4.0 SR8 (McNeel North America, Seattle, WA, USA), generated by an average of several anatomical structures of maxillar bones, based on data from the CT database owned by the CTI (Centro de Tecnologia da Informação Renato Archer – Campinas, São Paulo-Brasil).

Three-dimensional (3D) CAD models of the implants and abutments were provided by the manufacturer (Neodent Ltda) and edited by the same software used to generate the maxillar bone geometry, in order to promote some essential simplifications (figure 1).To simplify the bone model, a plateau was drawn in the region of the posterior teeth and the threads of implants were represented by symmetric rings.

Figure 1: CAD of implants, abutments and abutment screws.

The abutments Morse taper, hexagon internal and external, abutment screw and implants materials properties were provided by the manufacturer. The properties of NiCr alloy, used in this prosthesis (VerabondII), cortical bone and trabecular bone were got in the literature. These data are shown in table 1.

Table 1: Material properties used in FEM studies of dental implants.

	Young's modulus (E) – Mpa	Poisson's ratio (nu)	Reference
Cortical Bone	13.70	0.30	Meijer et al. 1993[11]; Menicucci et al. 2002[12]
Trabecular Bone	1.30	0.30	Meijer et al. 1993[11] Menicucci et al. 2002[12]
Implant (Titanium c.p. grade 4)	103	0.36	Manufacturer
Abutment and Abutment screw (Titanium alloy)	105	0.36	Manufacturer
Prosthetic Structure (Ni Cr alloy)	210	0.28	Anusavice KJ, 2003[13]

A controlled FEM mesh was used to represent the model of the implants and the bone (Figure 2). The number of elements and nodes used in this study are given in table 2. The contacts between the surfaces are listed in table 3.

The maxilla mechanical models were configured as linear-elastic, isotropic, and homogenous properties[14-15].

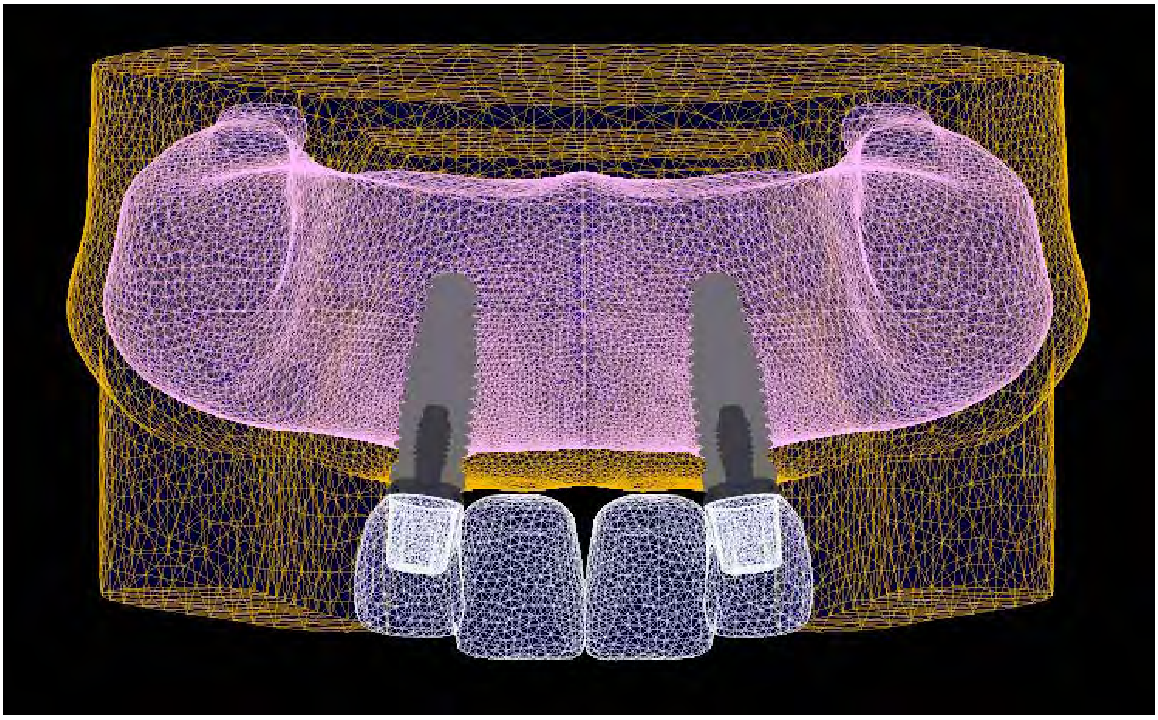


Figure2. Fine mesh of model of external hexagon connection.

Table 2: Number of elements and nodes used in finite element models.

	Models of Morse-taper connection	Models of external hexagon connection	Models of internal hexagon connection
Elements	140,136	1,038,096	1,235,794
Nodes	908,588	669,839	796,626

Table 3: Type of contact in the studied regions.

Interface	Contact type
Cortical/Trabecular	Bonded
Trabecular/Implant	Bonded
Cortical/Implant	Bonded
Abutment /Implant	Common
Abutment/Abutment screw	Common
Abutment screw/Implant	Bonded
Prosthetic Structure/Abutment	Bonded

Loads

The forces were applied on the region equivalent to the cingulum of each tooth present in the prosthesis, perpendicular to the surface of each tooth and in an angle of 45° related to the long axis of the tooth[6,16]. With the total load of 150N applied on the prosthetic structure[17] (Figure 3).

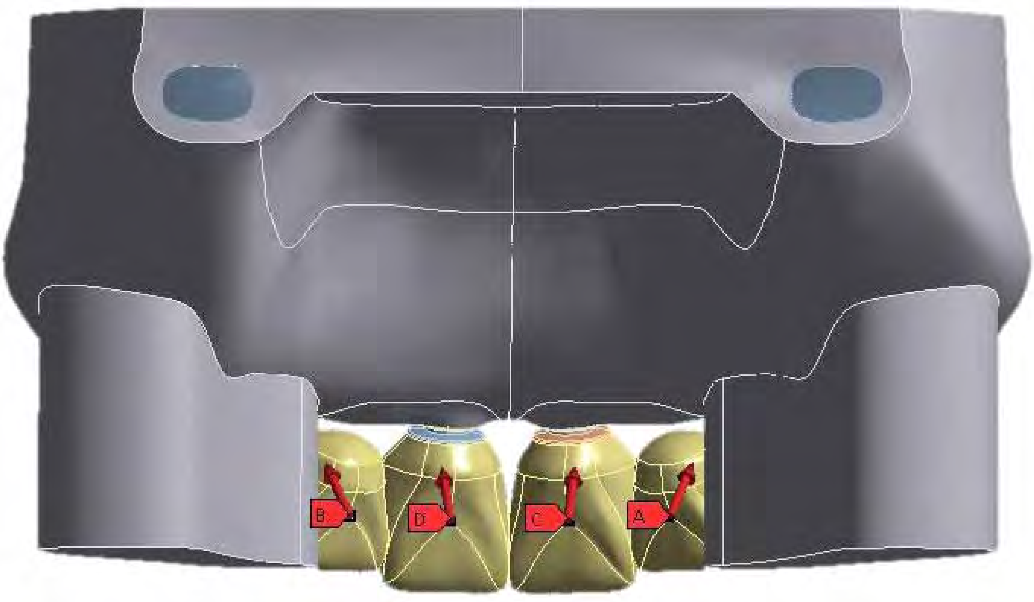


Figure 3: Representation of load application.

RESULTS

The figures 4A, 4B and 4C show a front view of the results of displacement, for the prosthetic structure of the models with Morse taper, external hexagon and internal hexagon implants respectively. The color scale of the figures shown is standardized.

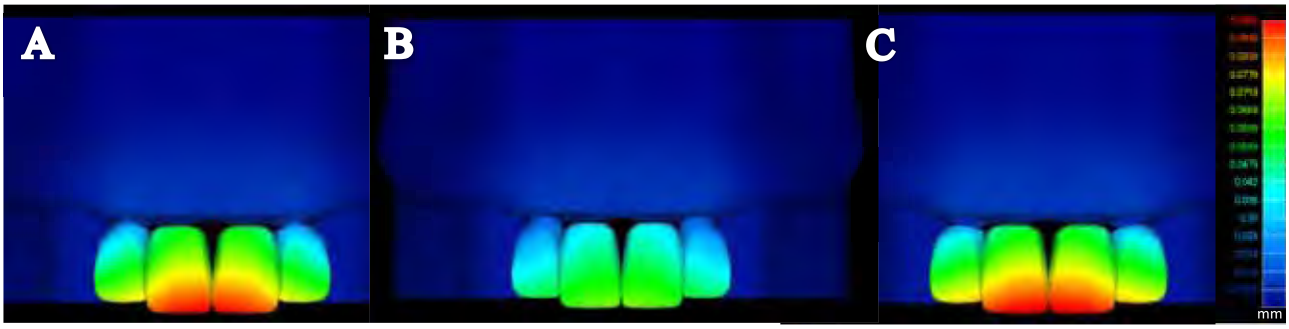


Figure 4. Front view of displacement of prosthetic structure of A, Morse-taper; B external hexagon and C, internal hexagon.

In the Figures 4A and 4C it is possible to observe that in this type of prosthetic configuration, the internal hexagon and Morse taper connections have similar profiles of prosthetic displacement. The external hexagon connection has shown the same pattern of stress distribution, but with differences in terms of absolute values of 0.02mm lower than the other groups (figure 4B).

The pattern of distribution in the connection Morse taper has its peak of von Mises stress located in the anterior region of the cone with a concentration in the top of the cone. On the back, these stresses are more dissipated and have less area of concentration (fig 5A).

In external hexagon connection, the von Mises stress is located in the anterior region, specifically in the base of the prosthetic abutment, where there is contact between the pillar and the implant. Small amount of stress is distributed by the edges and walls of the hexagon (Fig 5B).

In the internal hexagon connection, the propagation of von Mises stress starts in base of the abutment and in the coronal portion of the edges of the cone to the back and bottom of the hexagon (Fig 5C).

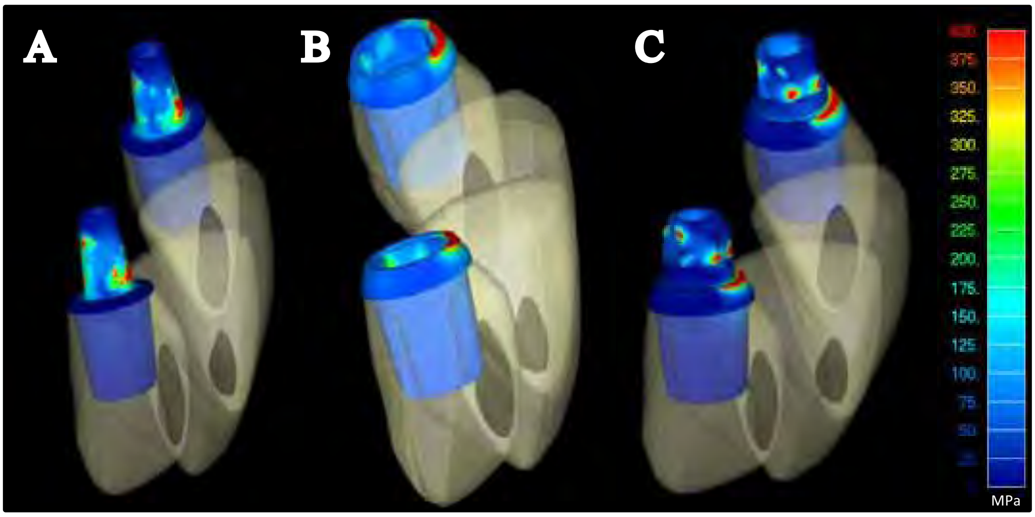
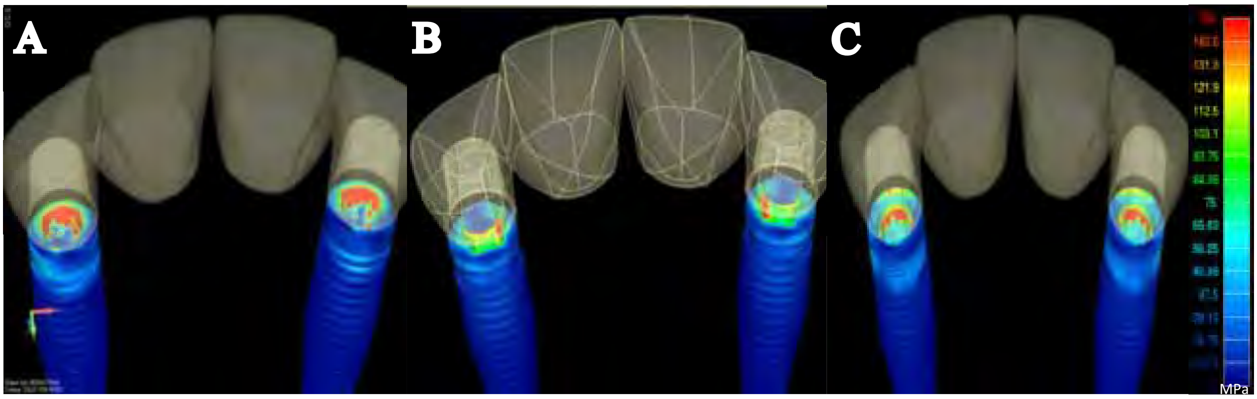


Figure 5. von Mises stress in prosthetic abutments of the connections of A, Morse-taper; B, external hexagon and C, internal hexagon.

The Morse-taper and internal hexagon implants have their von Mises stress distribution predominantly located in the anterior region. Morse-taper implants presented greater von Mises stress in the platform area and internal region of the cone. The implants of internal hexagon connection have a smaller area of von Mises stress distribution compared to Morse-taper implants, and lower areas and lower von Mises peak stress on the platform of the implant and lower von Mises stress in the internal region of the hexagon (Fig 6A e Fig 6C).

The external hexagon implant presented the highest von Mises stresses in the posterior region of the implant base, walls and edges of the external part. The peak stress in this prosthetic configuration is located at the edges of the hexagon of the implant (Fig 6B).

There is also a trend in the von Mises stress distribution to be located in the first threads of the implant insertion, which is the region corresponding to the cortical bone.



Figures 6. von Mises stress in the implant of A, Morse taper; B, external hexagon and C, internal hexagon connections.

Figures 7A, 7B and 7C are the front views of the von Mises stress for the abutment screws of different connections. For a description of results, the screws were divided into five parts: screw head, screw shaft top, screw shaft bottom, screw thread screw top and thread bottom.

The von Mises stress of the abutment screw of Morse taper connection appears in the bottom region of the screw thread and mainly at the screw shaft bottom, with the peak of stress less than 150MPa (Fig 7A).

In the abutment screw of the external hexagon connection the peak of von Mises stress is observed in the base of the screw head and shaft, with an intensity of about

270MPa. Stress concentration is also observed with lower intensity in the region of top screw thread top (Fig 7B).

The abutment screw of internal hexagon connection presents its von Mises stress peak of 260MPa, in the regions of the base of the head, shaft top and thread top of the screw (Fig 7C).

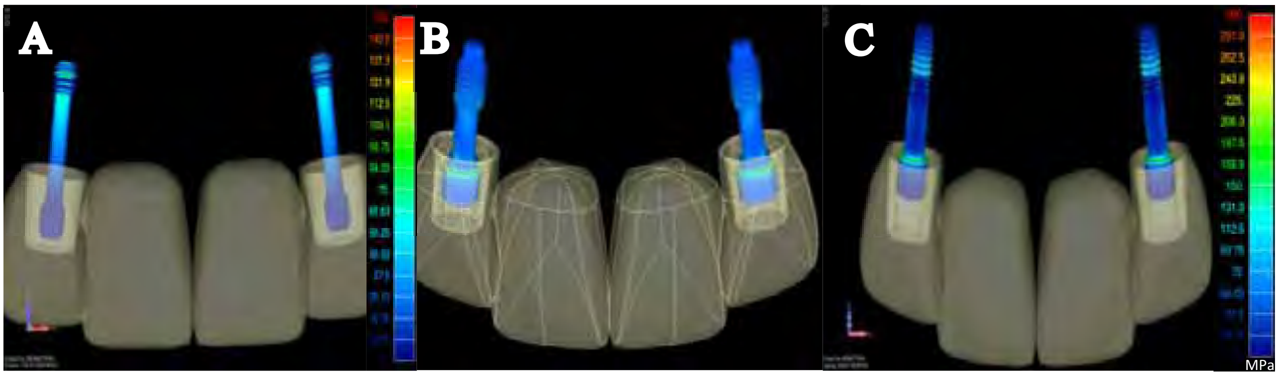


Figure 7. Front view of Von Mises Stress in abutment screws anterior of A, Morse taper; B, external hexagon and C, internal hexagon connections.

In the posterior view (Figures 8A, 8B, 8C), all the abutment screws show high values for von Mises stress, distributed on large areas.

In the screw of the Morse taper connection the von Mises stress is presents in the region of contact in the screw head, shaft bottom and thread bottom. The peak stress value of this screw is about 150MPa (Fig8A).

The screws of internal and external hexagon connections have higher von Mises stress values in relation to the screw of Morse taper. The maximum von Mises stress in those screws is about 300MPa. For the screw of external hexagon connection, the stress is mainly located at the base of the screw head, shaft bottom and thread top (Fig 8B).

In the screw of the internal hexagon, the area of maximum stress is wider than the one of the external hexagon, but with the same peak value of von Mises stress of 300MPa. The regions of highest stress concentration are: base of the screw head, shaft top and thread top (Fig 8C).

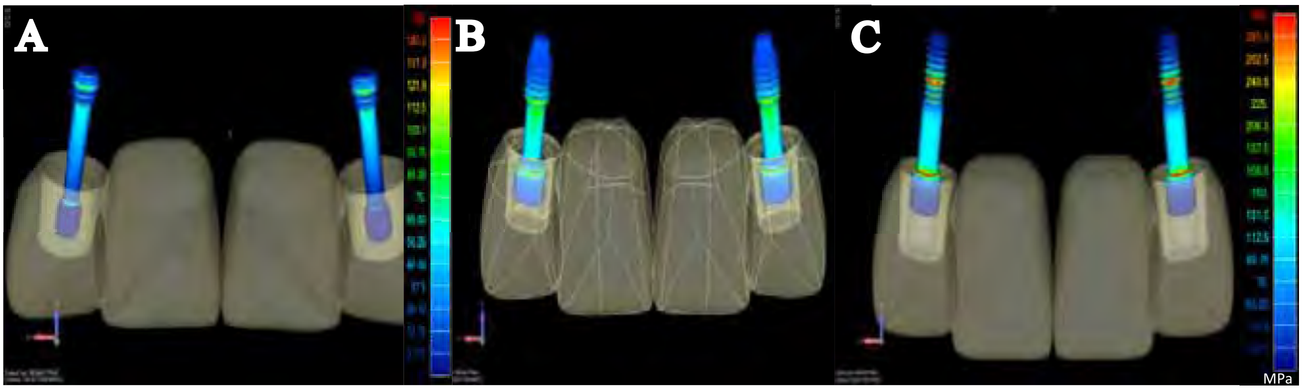


Figure 8. Posterior view of Von Mises Stress in abutment screws A, Morse taper; B, external hexagon and C, internal hexagon connections.

DISCUSSION

An accurate and precise connection between the implant and the abutment is an essential condition for appropriate functionality and stability of the implant rehabilitation.

All systems of abutment connection have their advantages and disadvantages. According to Maeda et al. 2006[18], the external hexagon connection has advantages such as: suitability for the two stage method, provision of an anti-rotation mechanism, reversibility and compatibility among different systems and more versatility for solving prosthetic laboratory problems[19]. Such disadvantages can present: micro-movement of the abutment, high center of rotation that leads to lower resistance for lateral and rotational loading and the presence of micro-gap[18].

In the internal hexagon connection the advantages are: ease in abutment connection, suitability for one stage implant installation, higher resistance to lateral loading, consequently high stability. The disadvantages are: thinner lateral fixture wall at the connecting part and difficulty for adjustment of divergences between implants installed[18].

Some presented advantages for Morse-taper connection are better sealing capacity due to the intimate contact of the internal walls of the cone to implant and high mechanical stability[20].

Mechanical stress distribution and displacement are important factors in the implant success[21].

The analysis of the results of displacement of the prosthetic structure of the connections shows that the external hexagon connection system presented the smallest displacement, what is very favorable for the maintenance and longevity of the prosthetic structure. But when the von Mises stress is observed in the abutment screws, it is two times higher than those presented in the screw-Morse cone connection.

According to a study done by Wang et al. 2009[15], the location of the stress in the screw of external hexagon implant is similar to that found in this study, with the highest values of von Mises stress observed in the region screw shaft top, screw shaft bottom and screw thread top.

Several retrospective clinical studies have shown a high incidence of screw loosening and/or fracture, usually associated with external hexagon implant connection[22-24].

One hypothesis for the higher incidence of screw loosening in external hexagon connection may be due to the fact that the short external hexagon does not stabilize mechanically the system on the application of lateral forces, and then the screw has to absorb much of the force applied.

Several studies have indicated a mechanical advantage of conical connections over external hexagon connection. The mechanics of the Morse taper connections resulted in lower incidences of mechanical complications, specifically abutment screw loosening and fracture, in comparison with those reported for external hexagon implants[20,25].

In this study, the high stress observed in the abutment screws connecting external hexagon, can justify the results of the studies cited above. The results of von Mises stress in the abutment screws of internal hexagon connection also showed similar results to those presented by the external hexagon connection. Fact that should be analyzed with caution during prosthetic rehabilitation with a cemented prosthesis, in which the screw loosening or screw fracture can cause the loss of all prosthetic treatment carried out.

In contrast, the Morse taper connection seems to reduce the stress on the portion of the abutment screw.

However, the screws loosening phenomenon can be reduced or avoided by choosing abutments with accuracy for the implant abutment fitting[26], suitable materials (Ti alloys are less susceptible to loosening screw Tsuge et al., 2009[27]) and execution of preload of abutment screw.

The FEM is an excellent method to obtain detailed quantitative data, and it enables accurate visualization of the stress distribution and displacement in models of complex geometries such as the maxilla. However, there are inherent limitations in any FEM study that limit the extrapolation of the results to clinical situations.

For ensure that the results of the MEF studies correspond exactly to the behavior of the in vivo structure is necessary a validation by an experimental technique.

CONCLUSION

The type of prosthetic connection has influence in the distribution and intensity of stress/strain in the prosthesis/implant system when two implants are used in a cemented rehabilitation of four dental elements procedure.

The external hexagon connection promotes less displacement of the prosthetic structure in the case studied; however the abutment screw of this connection receives the most of the von Mises stress of the system and can be subjected to mechanical failure.

The Morse taper connection showed small values of von Mises stress in the screw, what can be favorable to a cemented prosthetic rehabilitation, avoiding the abutment screw loosening.

REFERENCES

1- Huang HM, Tsai CM, Chan CC, Lin CT, Lee SY. Evaluation of loading conditions on fatigue-failed implants by fracture surface analysis. Int J Oral Maxillofac Implants 2005; 20:854-859.

2- Stuker RA, Teixeira ER, Beck JCP, Costa NP. Preload and torque removalevaluation of three different abutment screws for single standing implant restorations. J Appl Oral Sci. 2008; 16:55-8.

3- Siegele D, Soltesz. Numerical investigation of the implant shape on stress distribution in the jaw bone. Int J Oral Maxillofac Implants 1989; 4:333-340.

4- Himmlova L, DostalovaT, Kacovky A, Konvickova S. Influence of implant length and diameter on stress distribution: a finite element analysis. J Prosthet Dent 2004; 1:20-25.

5- Pierrisnard L, Renouard F, Renault P, Barquins M. Influence of implants length and bicortical anchorage on implants stressdistribution. Clin Implant Dent Relat Res. 2001; 5:254-262.

6- Pessoa RS, Muraru L, Marcantonio Júnior E, Vaz LG, Sloten JV, Duyck J, Siegfried V.N.J. 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.

7- Da Silva EF, Pellizzer EP, Mazaro JVQ, Garcia Júnior IR. Influence of the Connector and Implant Design on the Implant–Tooth-Connected Prostheses. Clin Implant Dent and Relat Res 2010; 3:51-55.

8- Sevimay M, Turhan F, Kilicxarslan MA, Eskitascioglu G. Three-dimensional finite element analysis of the effect of different bonequality on stress distribution in an implant-supported crown. J Prosthet Dent 2005; 93:227-34.

9- Daas M, Dubois G., Bonnet AS, Lipinski P, Rignon-Bret C. A complete finite element model of a mandibular implant-retained overdenture with two implants: Comparison between rigid and resilient attachment configurations. Med Eng Phys 2008; 30:218–225.

10- Van Staden RC, Guanh, Loo YC. Application of the finite element method in dental implant research. Comput Methods Biomech Biomed Eng 2006; 9:257–270.

11- Meijer HJ, Starmmans FJ, Steen WH, Bosman F. A three dimensional, finite element analysis of bone around dental implants in an edentulous human mandible. Arch Oral Biol. 1993; 38:491-496.

12- Menicucci G, Moslovoc A, Mozzati M, Lorenzetti M, Preti G. Tooth implant connection: some biomechanical aspects based on finite element analyses. Clin Oral Implants Res 2002; 13: 334-341.

13- Anusavice KJ. Phillips’ science of dental materials. 11th ed. St. Louis: Elsevier; 2003. p. 93-4, 598-603.

14- Chun HJ, Cheong SY, Han JH et al. Evaluation of design parameters of osseointegrated dental implants using finite element analysis. J Oral Rehabil. 2002;29:565–574.

15- Wang RF, Kang B, Razzoog ME. The dynamic natures of implant loading. J Prosthet Dent 2009; 101:359-371.

16- Eraslan O, Aykent F, Yucel TM, Akman Serhan. The finite element analysis of the effect of ferrule height on stress distribution at post-and-core-restored all-ceramic anterior crowns. Clin Oral Invest 2009; 13:223–227.

17- Morneburg TR, Pröschel PA. Measurement of masticatory forces and implant loads: A methodologic clinical study. Int J Prosthodont 2002; 15:20–27.

18- 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.

19- Carr BR, Dersh DA, Harrison WR, Kinsel RP. When contemplating treatment involving endosseous implants, what clinical and laboratory factors most significantly affect your choice of an implant system? Int J Oral Maxillofac Surg 2001; 16:123-7.

20- Norton MR. An in vitro evaluation of an internal conical interface compared to a butt joint interface in implant design. Clin Oral Implants Research. 1997; 8:290-298.

21- Nishioka RS, Vasconcellos LGO, Melo GN. Comparative strain gauge analysis of external and internal hexagon, morse taper, and influence of straight and offset implant configuration. Implant Dent 2001; 20:24-30.

22- Zarb GA, Schmitt A. The longitudinal clinical effectiveness of osseointegrated dental implants: the Toronto study. Part III: Problems and complications encountered. J Prosthet Dent 1990; 64:185-194.

23- Jemt T, Lindén B, Lekholm U. Failures and Complications in 127 Consecutively Placed Fixed Partial Prostheses Supported by Brånemark Implants: From Prosthetic Treatment to First Annual Checkup. Int J Oral Maxillofac Implants 1992; 7:40-44.

24- Becker W, Becker BE. Replacement of maxillary and mandibular molars with sinle endosseous implant restorations: a retrospective study. J Prosthet Dent 1995; 1:51-5.

25- Merz BR, Hunenbart S, Beser UC. Mechanics of the implant-abutment connection: an 8-degree taper compared to a butt joint connection. Int J Oral Maxillofac Implants 2000; 15:519–526.

26- Schwarz MS. Mechanical complications of dental implants. Clin Oral Impl Res 2000: 11(Suppl.):156-158.

27- Tsuge T, Hagiwara Y. Influence of lateral-oblique cyclic loading on abutment screw loosening of internal and external hexagon implants. Dent Mater J 2009; 28:373–381.

CAPÍTULO 3

Validation of a 3D finite element model of an anterior maxilla prosthesis by electronic speckle pattern interferometry

Manuscrito redigido nas normas do manual FOAr/UNESP, adaptadas das normas Vancouver

Capítulo3

Validation of a 3D finite element model of an anterior maxilla prosthesis by electronic speckle pattern interferometry

Cássia Bellotto Corrêa(*), Nuno Viriato Ramos[2], Jaime Monteiro [3], Luis Geraldo Vaz[4] and Mário A Pires Vaz[5].

[1]Departmento de Diagnóstico e Cirurgia, Faculdade de Ododntologia de Araraquara, Univ Estadual Paulista – UNESP, Brazil

[2,3,5]Laboratory of Optics and Experimental Mechanics, INEGI, FEUP, University of Porto, Portugal.

[4]Department de Materias Dentários e Prótese, Faculdade de Odontologia de Araraquara, Univ Estadual Paulista – UNESP, Brazil

(*)*Email:* cassiabcorrea@hotmail.com

Validation of a 3D finite element model of an anterior maxilla prosthesis by electronic speckle pattern interferometry

ABSTRACT

The Finite Element Analysis (FEA) has been very used in dentistry to evaluate the mechanical behavior of implants, prosthesis and also the bone itself. However FEA is a computational technique which relies in a mathematical model simulating the reality, this way some mistakes or incorrections made during the programming process may influence the final results.

The aim of this study is to validate 3D Finite Element model of a dental rehabilitation using experimental data obtained with a technique known by electronic speckle pattern interferometry (ESPI). A 3D model of the maxilla was constructed from a tomographic database. This model had the absence of the maxillary incisors whose region was rehabilitated by an implant-fixed anterior prosthesis. Using an ESPI setup and a 3D model, an experimental measurement was performed to adjust and validate the computational simulation.

This study can confirm for the potentialities of FEA as an effective technique of stress/strain and displacement analysis for internal biomechanics if used with validated models.

Keywords: FEA, ESPI, Biomechanics, Prosthesis, Implants.

INTRODUCTION

With the predictability and longevity obtained by rehabilitation with dental implants, special attention has been given to the distribution of mechanical stress, strain and displacement in the implant vicinity.

Several methods have been used to analyze the biomechanical system representing the dental implants. The finite element analysis (FEA) is being increasingly used due to its versatility. The FEA has become a valuable tool for evaluating the distribution of stresses/ strains in the system prosthesis / implant and its relation with the supporting bone[6].

However, the FEA is a numerical technique which uses mathematic models and may lead to errors if special care is not taken during its preparation. An inappropriate discretization, a bad loading simulation or wrong boundary conditions can distort the final results. In order to ensure and validate the computer model is necessary to compare the obtained data with experimental results. ESPI was the technique chosen for this study due to its non-intrusive and non-destructive nature, with a sub-micrometer resolution, which can be used with no-contact over diffused surfaces and no special preparation to assess the displacement distribution[14].

ESPI is a speckle interferometry technique that uses laser radiation and an optical setup where the interferometric patterns are recorded on a video camera. The classic setups were adapted to the new digital image recording devices and this helped in the dissemination of this technique[11]. The basic principle of the ESPI technique is that the speckle pattern intensity distribution is a function of the relative phases of two interfering waves. ESPI results are obtained as interferometric fringe patterns representing the regions of equal displacement in the direction of the sensitivity vector[9].

Using temporal phase modulation the deformation phase map can be assessed and quantitative data is recorded.

MATERIAL AND METHODS

A 3D model of the anterior maxilla was created based on a CT scan database available at the Center for Information Technology Renato Archer (CTI, Campinas, Brazil). To construct this model a simplified geometry was obtained with software Rhinoceros 4.0 SR8 (McNeel North America, Seattle, WA, USA).

The geometric models of the abutments, screws and implants were constructed in the same modeling software used for the maxilla, based on CAD (Computer Aided Design) provided by the manufacturer Neodent Ltda, Curitiba, Paraná, Brazil.

A specific model for each group was created and the problem was classified in three possibilities (Figure1):

Group IL- Implants in position of lateral incisors and the pontic in the position of central incisors.

Group ILIC- Implants in position of the lateral incisor and central incisor, with a pontic and cantilever.

Group IC- Implants in position of central incisors and cantilever of the lateral incisors.

Figure 1 – Front view of the three models used to simulate the cases studied, group IL, ILIC and IC.

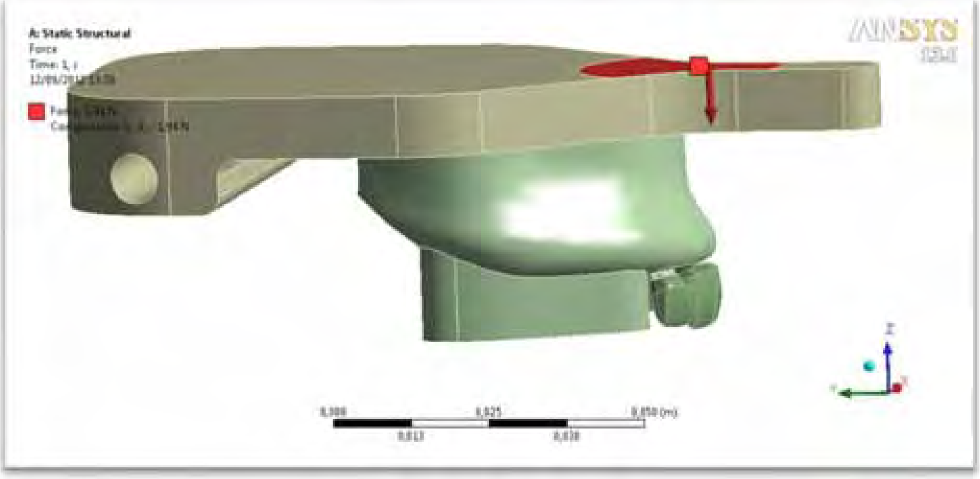
Each CAD model was imported by the software Ansys® version _13.00 for the pre-processing of the numerical model, in which all information to characterize the biomechanical situation were given.

The mechanical properties attributed to the components of the dental structures were characterized by Modulus of Elasticity (E) and Poisson's ratio (μ), whose values were given by manufacturer (Neodent Ltda), and others were obtained from the literature or calculated by the authors as are shown in Table 1. All materials were considered as isotropic, homogeneous, and linear elastic.
A total load of 1,94N, 2,98N or 4,78N was distributed and applied at the top of the rig structure; which is a mechanical device were the model can be fixed and with displacements equivalents of the biomechanics. The model used in the numerical simulation is represented on Figure 2.

Table 1 - Material properties used in FEM studies.

	Young's modulus (E) – MPa	Poisson's ratio (n)	Reference
Epoxi Resin	6,88	0.3	Authors
Implant (Titanium c.p. grade 4)	103	0.361	Manufacturer
Abutment and prosthetic screw (Titanium alloy)	105	0.361	Manufacturer
Prosthetic Structure (NiCr alloy)	210	0.28	Anusavice KJ, 2003

Figure 2 - Representation of load conditions applied to the FEA model.



The 3D model was prototyped and duplicated in Epoxi Resin (Maxiepoxi®) with the implants positioned in accordance with each respective group. The prosthetic structure was also prototyped and duplicated in NiCr alloy (Verabond II), using the same protocols used in clinics.

To perform the measurements with ESPI, the models were fixed in an articulator and loads of 1,94N, 2,98N or 4,78N were applied. For coherent illumination a Coherent Verdi laser, delivering up to 2W in 532nm wavelength was used. The interferometric patterns were recorded on a video camera and their analysis was carried out on the computer. A picture of the loading rig can be seen on Figure 3. The calculation of directional displacement obtained from these images were compared with the FEA results.

Figure 3 - Loading rig obtained for the ESPI measurements.

RESULTS

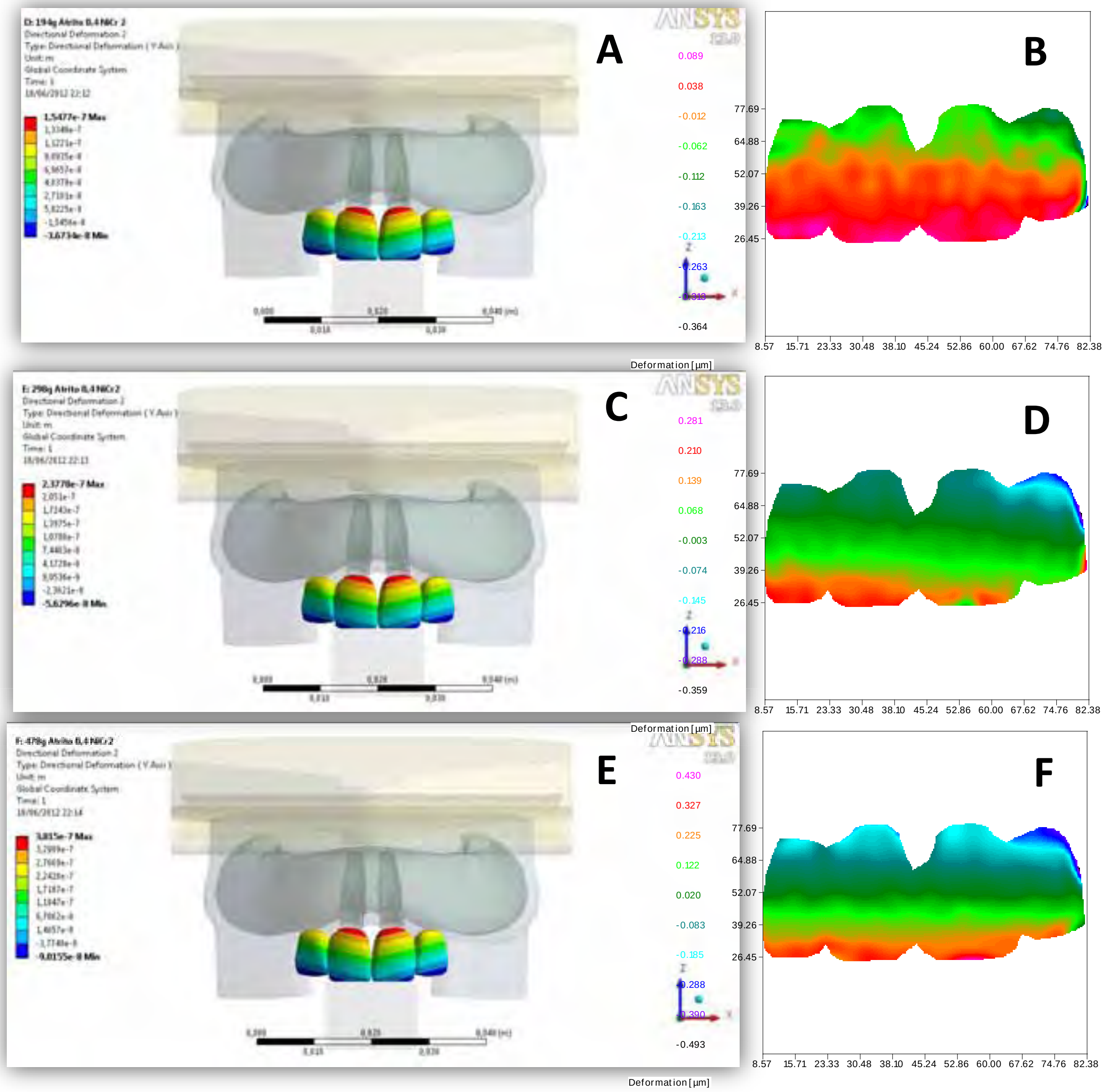
The data obtained from the computational simulation with the 3D model were compared with ESPI results obtained from the measurements over the prototypes. The behaviour of prosthetic structure was similar in FEA and ESPI, with the incisal of teeth moving forward and cervical moving backward (Figure 4A, 4C and 4E). The total deformation was considered as the sum of forward moving and backward moving.

The behaviour of the models were similar in the FEA and ESPI. This can be observed by the arrangement of the fringes in every loading (Figure 4A/4B; 4C/4D and 4E/4F).

The highest displacement was obtained with the group IL, followed by the group ILIC and group IC. However, the values of directional deformation in ESPI were higher than those presented in the FEA. The results of FEA and ESPI of all groups are represented in Figure 4.

An analysis was done to verify the correlation between the two methodologies. The results in graphic of Figure 5 show an positive correlation between the FEA and ESPI.

Figures 4A, 4C and 4E - FEA of group IC with loading 1,94N, 2,98N and 4,78N respectively. Figures 4B, 4D and 4F: ESPI of group IC with loading 1,94N, 2,98N and 4,78N respectively.



The data presented in the Figures 4A, 4C and 4E were obtained with a FEA model were the prototypes, dental model and prosthesis with implants were model by a 52036 tetrahedral elements mesh with 91263 nodes. The three cases were obtained by diferent positioning of the implants.

Figure 5 - Comparison of the results of deformation obtained by MEF and ESPI for the group IL , group ILIC and group IC in the three loading conditions (1,94N, 2,98N and 4,78N).

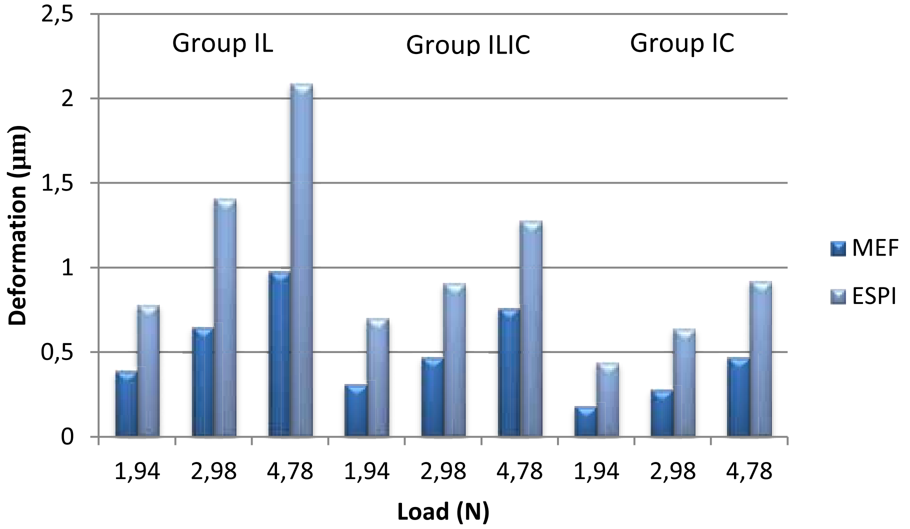
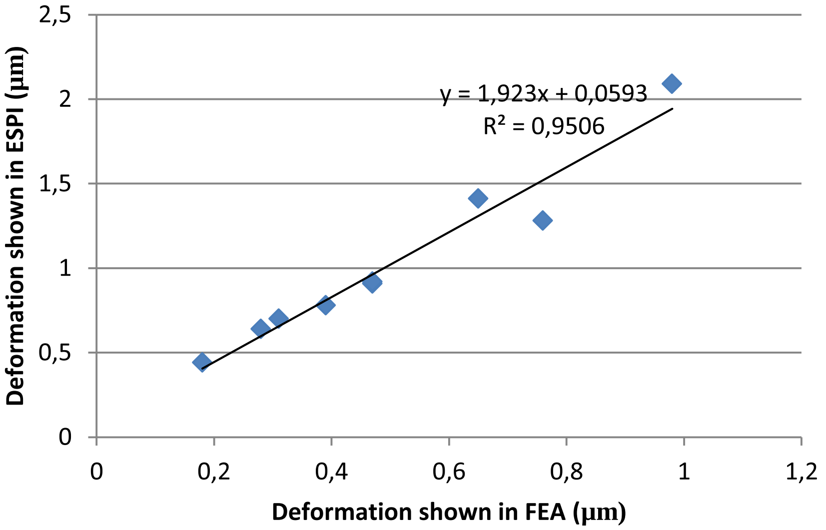


Figure 6 - Correlation between FEA and ESPI results.



DISCUSSION

The FEA has been used in the dentistry due to their ability to deal with complex geometries, this way it was applied to different kind of studies such as: to verify the influence of dental implant design[3,15] and to evaluate the bone quality on stress distribution around the bone[15] .

This method offers advantage of solving difficult problems by dividing complex structures into smaller and simpler interrelated sections by using mathematical techniques[6,16].

However, some difficulties are encountered during the simulation of the mechanical behavior involving complex structures such human bone tissue.

Some assumptions and simplifications need to be done to make the modeling and solving process possible. These simplifications can influence the accuracy of the FEA results significantly are: detailed geometry of the biomechanical structures to be modeled, material properties, and boundary conditions.

Often, simplifications are made on the structure studied. For example, the transformation of the threads of the implant in parallel rings[5,8,10,12,13], the bone structure as homogeneous, linear and elastic[1], the osseointegrated implants is assumed such bonded to the cortical and trabecular bone[1]. This does not occur so exactly in clinical situations. There an imperfect contact and this may have effect on load transfer from implant to supporting bone.

Another simplification that some authors adopt is the assumption of the connection of abutment and implant, screw and abutment and prosthesis and abutment how bonded contact, which does not always correspond to the reality, where there is the presence of micro displacements between these structures. This assumptions are made to give conditions so that the computational program can solve the problem proposed.

Through the results, is possible to note that the simplifications assumed in this study influence the reproducibility of the real conditions. This can be seen by the non-accuracy of the deformation values obtained through the directional ESPI compared to the FEA.

However, the FEA accurately reflects the mechanical behavior observed in ESPI. This can be confirmed by the positive correlation shown by the two techniques, which makes the technique FEA valid and useful for studies in dentistry involving complex geometries. And still provides a detailed stress/strain and displacement distributions in all regions of the studied structure.

The results of this study are in agreement with Gröning et al. 2012[7] that found some discrepancies in comparison of the results in FEA and Digital speckle pattern interferometry (DSPI) that second the authors are most probably caused by inaccuracies in the models' geometries and the degree of simplification of the modeled material properties.

It is noteworthy that the three-dimensional model validation by an experimental technique has great relevance and importance[2,4]. With an experimental validation, the FEA may be used to compare different scenarios, checking the distribution of stress and displacement in any system.

Other great advantage of FEA is the possibility of viewing the mechanical behavior of all structures of a system in any 3D position[8,16].

CONCLUSION

This study shows that FEA is an effective technique of stress analysis if used with validated models. ESPI is an experimental technique suitable for the assessment of deformation in the set up used. According to ESPI data the 3D FEA model is defined correctly and can be used for further investigations.

ACKNOWLEDGMENTS

The authors wish to acknowledge thanks the grants and scholarships from CAPES (Committee for Postgraduate Courses in Higher Education), FAPESP (Research Support Foundation of São Paulo State), CTI (Centro de Tecnologia da Informação Renato Archer), the Neodent® implant system for providing the implant CAD models, dental implants and abutments.

REFERENCES[*]

1. Bozkaya D, Muftu S, Muftu A. Evaluation of load transfer characteristics of five different implants in compact bone at different load levels by finite elements analysis. J Prosthet Dent. 2004; 92(6): 523-30.

2. Chatzgianni A, Keiling L, Duschner H, Gotz H, Eliades T, Bourauel. Comparative analysis of numerical and experimental data of orthodontic mini-implants. Eur J Orthod. 2011; 35(5): 468-75.

3. Chun HJ, Cheong SY, Han JH, Heo SJ, Chung JP, Rhyu IC, Choi YI, et al. Evaluation of design parameters of osseointegrated dental implants using finite element analysis. J Oral Rehabil. 2002; 29(6): 565–74.

4. Doyle BJ, Killion J, Callanan A. Use of the photoelastic method and finite element analysis in the assessment of wall strain in abdominal aortic aneurysm models. J Biomech. 2012; 45(10): 1759-68.

5. Faegh S, Muftu S. Load transfer along the bone-dental implant interface. J Biomech. 2010; 43(9): 1761-70.

6. Geng, JP, Tan KB, Liu, GR. Application of finite element analysis in implant dentistry: a review of the literature. J Prosthet Dent. 200; 85(6): 585-98.

7. Groning F, Bright JA, Fagan MJ, Higgins PO. Improving the validation of finite element models with quantitative full-field strain comparisons. J Biomech. 2012; 45(8):1498-1506.

---

[*] De acordo com o estilo Vancouver. Disponível em: http://www.nlm.nih.gov/bsd/uniform_requirements.html

8. Holmgren EP, Seckinger RJ, Kilgren LM, Mante F. Evaluating parameters of osseointegrated dental implants using finite element analysis – a two-dimensional comparative study examining the effect of implant diameter, implant shape, and load direction. J Oral Implantol. 1998; 24(2): 80-8.

9. Jones R, Wykes C. Holographic and Speckle Interferometry. 2nd ed. New York: Cambridge University Press; 1989.

10. Meijer GJ, Starmans FJ, de Putter C, van Blitterswijk CA. The influence of a flexible coating on the bone stress around dental implants. J Oral Rehabil. 1995; 22(2): 105-11.

11. Monteiro JP, Vaz MAP, Melo FQ, Gomes JFS. Use of interferometric techniques for measuring the displacement field in the plane of a part-through cracking existing in a plate. Int J Pressure Vessels Pip. 2001; 78(4): 253-9.

12. Pierrisnard L, Renouard F, Renault P, Barquins M. Influence of implant length and bicortical anchorage on implant stress distribution. Clin Implant Dent Relat Res. 2003; 4:254-62.

13. Sertgoz A. Finite element analysis study of the effect of superstructure material on stress distribution in an implant-supported fixed prosthesis. Int J Prosthod. 1997;10(1):19-27.

14. Smigielski P. Holographie industrielle. Toulouse: Teknea; 1994.

15. Tada S, Stegaroiu R, Kitamura E, Miyakawa O, Kusakari H. Influence of implant design and bone quality on stress/strain distribution in bone around implants: a 3-dimensional finite element analysis. Int J Oral Maxillofac Implants. 2003; 18(3): 357–68.

16. Van Staden RC, Guan H, Loo YC. Application of the finite element method in dental implant research. Comput Methods Biomech Biomed Engin. 2006; 9(4): 257-70.

Conclusão Geral

Levando em consideração as metodologias utilizadas (Apendice 1), é possível concluir que:

A configuração protética em que os implantes estão localizados nos incisivos laterais limitou o deslocamento da estrutura protética, porém levou a um maior deslocamento na estrutura óssea que as outras configurações.

A região cortical recebe e dissipa a maior quantidade de tensão no tecido ósseo.

O uso de um cantilever em reabilitação anterior de maxila leva a uma maior concentração de tensão no implante e maior deslocamento na estrutura óssea.

A conexão protética hexágono externo proporciona um menor deslocamento da estrutura protética, entretanto, o parafuso protético nesta conexão fica sujeito aos maiores valores de tensão de von Misses observados.

A conexão cone-Morse parece preservar o parafuso protético, no sentido de dissipar cerca de metade da tensão de von Misses em relação à observada nas demais conexões.

MEF pode ser usado como uma eficiente técnica para a análise de tensão, deformação e deslocamento, porém os resultados quantitativos devem ser observados com cautela na extrapolação clínica, ressaltando assim a importância de modelos validados experimentalmente.

Mais investigações científicas devem ser realizadas com a finalidade de esclarecer as relações entre o comportamento mecânico das estruturas envolvidas em uma reabilitação implanto suportada e a resposta biológica in vivo (Apêndice 2).

Referências[*]

1. Baffi L, Cappelloni I, Di Girolama M. The influence of implant diameter and length on stress distribution of osseointegrated implants related to crestal bone geometry: a threedimensional finite element analysis. J Prosthet Dent. 2008; 100(6): 422-31.

2. Barbier L, Schepers E. Adaptative bone remodeling around oral implants under axial and nonaxial loading conditions in the dog mandible. Int J Oral Maxillofac Implants. 1997; 12(2): 215-23.

3. Branemark PI, Adell R, Breine U, Hansson BO, Lindstroem J, Ohlsson A. Intra-osseous anchorage of dental protheses. I. Experimental studies. Scand. J Plast Reconstr Surg. 1969; 3(2): 81-100.

4. Chiche FA, Leriche MA. Multidisciplinary implant dentistry for improved aesthetics and function. Pract Periodontics Aesthet Dent. 1998; 10(2): 177-88.

5. Choquet V, Hermans M, Adriaenssens P, Daelemans P, Tarnow D, Maelevez C. Clinical and radiographic evaluation of the papilla level adjacent to single-tooth dental implants. A retrospective study in the maxillary anterior region. J Periodontol. 2001; 72(10): 1364-71.

---

[*] De acordo com o estilo Vancouver. Disponível em: http://www.nlm.nih.gov/bsd/uniform_requirements.html

6. Coward TJ, Watson RM. Locating angulated abutments: a technical note. Int J Oral Maxillofac Implants. 1997**;** 12(1): 82-3.

7. Duyck J, Ronald HJ, Van Oosterwyck H, Naert I, Vander Sloten J, Ellingsen JE. The influence of static and dynamic loading on marginal bone reactions around osseointegrated implants: an animal experimental study. Clin Oral Impl Res. 2001; 12(3): 207-18.

8. Gastaldo JF, Cury PR, Sendyk WR. Effect of the vertical and horizontal distances between adjacent implants and between a tooth and an implant on the incidence of interproximal papilla. J Periodontol. 2004; 75(9): 1242-6.

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

10. Grunder U, Gracis S, Capelli M. Influence of 3-D bone to implant relationship on esthetic. Int J Periodontics Restorative Dent. 2005; 25(2): 113-9.

11. Isidor F. Loss of osseointegration caused by occlusal load of oral implants. A clinical and radiographic study in monkeys. Clin Oral Implants Rev. 1996; 7(2): 143–52.

12. Misch CE, Suzuki JB, Misch-Dietsh FM, Bidez MW. A positive correlation between occlusal trauma and peri-implant bone loss: literature support. Implant Dent. 2005; 14(2):108-16.

13. Novaes Jr AB, Barros RRM, Muglia VA, Borges GJ. Influence of interimplant distances and placement deph on papilla formation and crestal resorption: a clinical and radiographic study in dogs. J Oral Implantol. 2009; 35(1): 18-27.

14. Salama H, Salama MA, Garber D, Adar P. The interproximal height of bone: a guidepost to predictable aesthetic strategies and soft tissue contours in anterior tooth replacement. Pract Periodontics Aesthet Dent Pract. 1998; 10(9): 1131-41.

15. Skalak R. Biomechanical considerations in osseointegrated prostheses. J Prothet Dent. 1983; 49(6): 843-8.

16. Tarnow DP, Cho SC, Wallace SS. The effect of inter-implant distance on the height of inter-implant bone crest. J Periodontol. 2000; 71(4): 546-9.

17. Tarnow DP, Elian N, Fletcher P, Froum S, Magner A, Cho SC, et al. Vertical distance from the crest of bone to the heigh of interproximal papilla between adjacent implants. J Periodontol. 2003; 74(12):1785-8.

Apêndice

A.1 Material e método

A.1.2 Metodologia do Capítulo 1 e 2

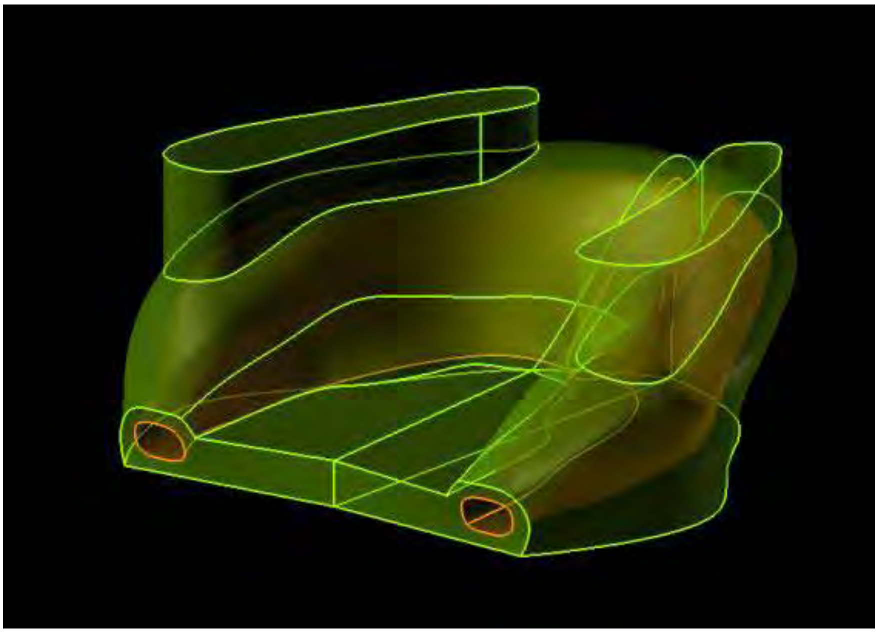
Ensaios em Método dos Elementos Finitos

O MEF é uma técnica pela qual um protótipo pode ser estudado mediante a criação de um modelo matemático preciso. Este método faz uso de um computador para resolver um grande número de equações matemáticas as quais simulam as propriedades mecânicas da estrutura a ser analisada.

O método possui duas características essenciais: os elementos finitos e a função de interpolação. Os elementos finitos são subdivisões do modelo, pequenas o suficiente para abranger todos os detalhes de um modelo de geometria complexa e ainda tornar possível a abordagem analítica em cada um destes elementos e obter a combinação de seus efeitos. Estes elementos são interconectados por pontos de união denominados pontos nodais ou nós. As funções de interpolação permitem, uma vez determinados os deslocamentos em cada nó, interpolar deslocamentos e calcular deformações e tensões em qualquer ponto da estrutura.

Os modelos para o ensaio de Métodos dos Elementos Finitos foram confeccionados a partir de um banco de dados de tomografias computadorizadas disponibilizado pelo Centro de Tecnologia da Informação Renato Archer (CTI - Campinas). Com base nas informações das imagens de tomografia, foram construídos os modelos geométricos simplificados (Figura 1) com auxílio do software Rhinoceros 4.0 SR8 (McNeel North America, Seatle, WA, USA).

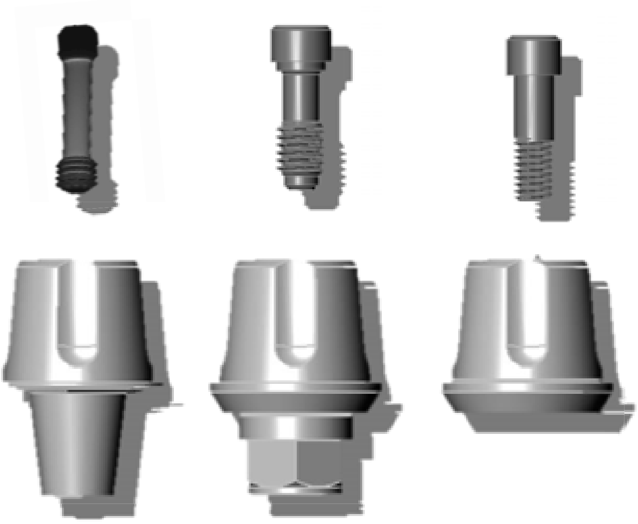
Figura 1 - Desenho em CAD da maxila elaborado por meio do software Rhino.



Os modelos geométricos dos componentes dos implantes foram construídos no mesmo software de modelagem utilizado para a maxila, com base nas imagens fornecidas pelo fabricante (Neodent®): implantes (Alvin 4.3X13mm) com conexão hexágono interno, hexágono externo e cone-Morse (Figura 2) e componentes protéticos (Pilar de Preparo Reto) também com os três tipos de diferentes conexões e os parafusos passantes (Figura 3).

Figura 2 - Ilustração dos CADs dos implantes nas diferentes conexões fornecidos pela empresa (Neodent®).

Figura 3 - Ilustração dos CADs dos componentes e parafusos passantes nas diferentes conexões fornecidos pela empresa (Neodent®).



As estruturas protéticas, com o respectivo encaixe nos componentes protéticos também foram modeladas no mesmo software de modelagem utilizado para a maxila.

Um modelo específico para cada grupo estudado foi gerado.

Divisão dos grupos para o capítulo 1:

Grupo IL - Implante cônico na posição dos incisivos laterais e pônticos na região dos incisivos centrais (Figura 4).

Figura 4 - Modelo gerado a partir do posicionamento dos implantes na região dos incisivos laterais.

Grupo IC- Implante cônicos na posição dos incisivos centrais e pônticos na região dos incisivos laterais (Figura 5).

Figura 5 - Modelo gerado a partir do posicionamento dos implantes na região dos incisivos centrais.

Grupo ILIC- Implante cônicos alternados com elementos suspensos (Figura 6).

Figura 6 - Modelo gerado a partir do posicionamento dos implantes alternados com elementos suspensos.

No capítulo 2 o Grupo IL foi dividido em:

Grupo1- Implantes com a conexão Cone-Morse

Grupo 2- Implantes com a conexão Hexágono Externo

Grupo 3- Implantes com a conexão Hexágono Interno

O modelo específico gerado para cada caso foi importado pelo software de análise pelo método dos elementos finitos FEMAP versão 10.1.1 para a fase de pré-processamento, em que foram dadas todas as informações para caracterizar a situação de solicitação biomecânica.

As propriedades mecânicas atribuídas às estruturas foram caracterizadas pelo Módulo de Elasticidade (E) e Coeficiente de Poisson, cujos valores encontram-se disponíveis na literatura ou foram fornecidos pelo fabricante e estão listados na tabela 1. Todos os materiais simulados foram considerados isotrópicos, homogêneos, elásticos e lineares.

Tabela 1 - Propriedade dos materiais usados em Método dos Elementos Finitos.

	Young modulus (E), Mpa	**Poisson ratio (nu)**	**Reference**
Cortical Bone	13.7	0.3	Meijer et al. 1993 Menicucci et al. 2002
Trabecular Bone	1.3	0.3	Meijer et al. 1993 Menicucci et al. 2002
Implant (Titanium c.p grade 4)	103	0.361	Manufacturer
Abutment and Abutment Screw (Titanium Alloy)	105	0.361	Manufacturer
Prosthetic Structure (Ni Cr Alloy)	210	0.28	Anusavice KJ, 2003

A malha foi construída com elementos tetraédricos com 10 nós.

Um carregamento de 150N foi aplicado no centro da superfície dos elementos protéticos, com uma angulação de 45º para simular a oclusão funcional.

A interface osso-implante foi considerada como uma adesão perfeita, e todo filete de rosca foi preenchido por osso para simular o implante osseointegrado. As interfaces entre as diversas estruturas do modelo estão listadas na tabela 2.

Tabela 2 - Tipos de contatos utilizados durante o processamento dos modelos.

Interface	**Tipo de contato**
Cortical/Trabecular	Colado
Trabecular/Implante	Colado
Cortical/Implante	Colado
Componente/Implante	Comum
Componente/Parafuso protético	Comum
Parafuso protético/Implante	Colado
Estrutura Protética/Componente	Colado

O processamento foi realizado pelo software NEiNastran versão 10.0.2.4 e o pós processamento foi realizado com o software FEMAP, para a visualização gráfica dos resultados na forma de campos de tensão de von Misses, Máxima Principal e deslocamento.

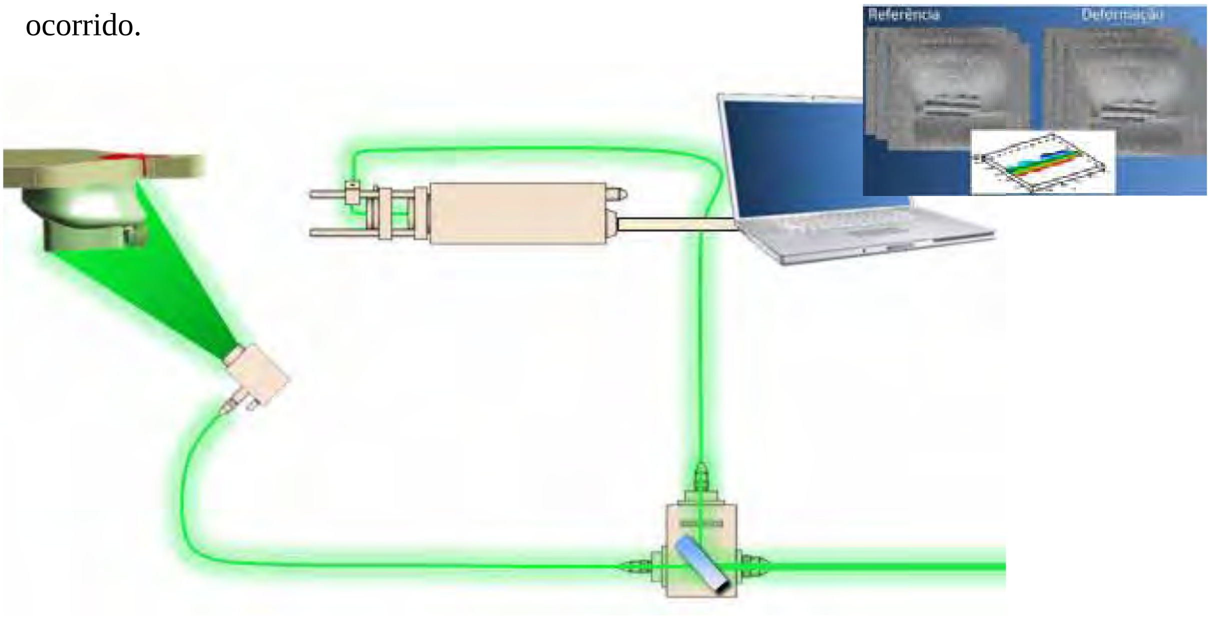
A.1.3 Metodologia do Capítulo 3

Interferometria Holográfica e Ensaios em Método dos Elementos Finitos.

A interferometria holográfica é uma técnica ótica, não invasiva, que possibilita a detecção de deformações da ordem dos 100 nm.

A metodologia de Interferometria holográfica é uma técnica experimental que utiliza radiação laser, um *setup* ótico e uma câmera de vídeo acoplada a um computador para gravação dos padrões interferométricos. Neste *setup* há o registro da situação inicial, utilizado como referência e um segundo registro que apresenta as eventuais modificações da superfície. Os resultados obtidos são padrões de interferência em forma de bandas, que correspondem às regiões de igual deslocamento. Esse padrão de bandas granitadas claras e escuras é gravado em uma câmera conectada ao computador as imagens são utilizadas na quantificação do deslocamento.

Figura7 - *Setup* ótico – feixe de laser verde dividido em dois feixes: feixe referência e feixe objeto. A luz difundida pela superfície do objeto é captada pelas lentes do sistema óptico e combinada com o feixe de referência. A câmera registra e armazena as imagens. Correlacionando as imagens é possível quantificar o deslocamento direcional ocorrido.



A.1.3.1Obtenção do modelo mestre

Foi confeccionado por meio de prototipagem um modelo 3D da região anterior de uma maxila com distância intercaninos de 27,3mm baseado em um banco de tomografias computadorizadas disponibilizado pelo Centro de tecnologia e Inovação Renato Archer (CTI).

Esse modelo de maxila anatômica foi gerado a partir de uma média de todas as estruturas anatômicas da região, dados esses adquiridos a partir do banco de tomografias pertencentes ao CTI (Figura 8 e 9).

Figura 8 - Vista frontal do modelo prototipado

Figura 9 - vista posterior do modelo prototipado

Propôs-se elaborar o modelo mestre simplificado, já com as respectivas perfurações para guiar os implantes a serem inseridos e, além disso, confeccionou-se um guia para auxiliar na fresagem dos modelos duplicados (Figura 10).

Figura 10 - Modelo prototipado com as perfurações.

O modelo prototipado foi então polido e envernizado para obtenção de uma superfície extremamente lisa, facilitando assim a duplicação do mesmo.

A.1.3.2 Confeção do guia

Um guia prototipado foi criado para auxiliar na fresagem dos modelos duplicados (Figura11).

Figura 11 - Guia prototipado posicionado no modelo prototipado

A.1.3.3 Duplicação do modelo

Para a duplicação do modelo mestre foi confeccionada uma moldeira individual (Figura 12). O modelo mestre foi copiado pela técnica de moldagem única com silicona de condensação leve e pesada (Zetaplus, Oranwash e Endurente -Zemarck), como mostra a figura 13.

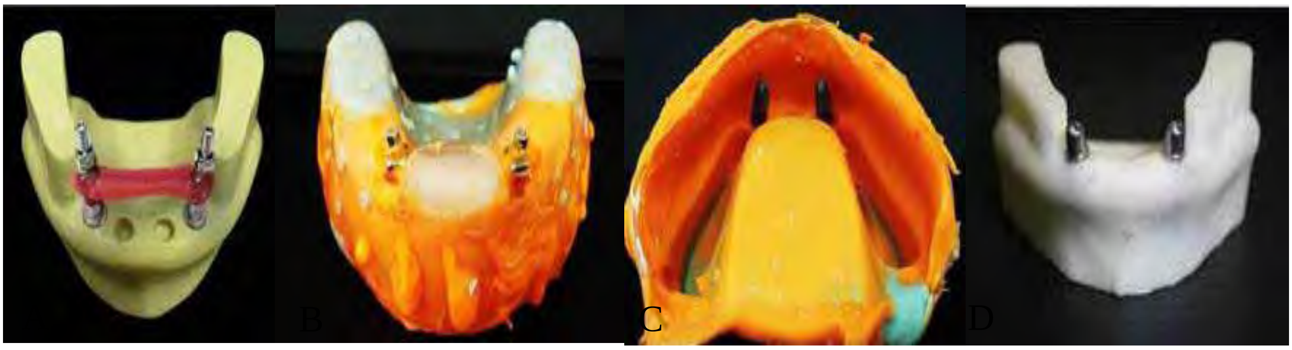
Figura 12 - Moldeira individual

Figura 13 - Molde do modelo mestre

O molde foi vazado com gesso pedra de baixa expansão de presa, aguardou-se a presa e o modelo foi então perfurado seguindo-se a sequencia de fresas recomendadas pelo fabricante com o auxílio da guia cirúrgica. O motor utilizado para fresagem dos modelos foi o Driller BLM600.

Após a instalação dos implantes no modelo de gesso, os transferentes para moldeira aberta foram posicionados nos implantes e unidos com resina duralay (Figura 14A), realizou-se a moldagem de transferência dos implantes (Figura 14B), no molde acoplaram-se os implantes (Figura 14C) e em seguida vazou-se a resina epóxi (Maxiepoxi®) com a finalidade de copiar a posição exata dos implantes e suas espiras estivessem em íntimo contato com a resina epóxi após o vazamento do molde (Figura 14D).

Figura 14 - A-transferentes em posição unidos com duralay; B- Moldadem de transferência; C- Implantes adaptados no molde; D- Modelo em Resina Epoxi.



A.1.3.4 Duplicação da estrutura protética

A estrutura protética prototipada (Figura 15) foi moldada (Figura 16), os cilindros calcináveis foram colocados nos conjuntos implante/componente protético (Figura 17), resina duralay foi inserida no molde e este levado em posição no modelo em resina epóxi. A estrutura em duralay recebeu um acabamento e foi fundida em liga de NiCr (VerabondII) (figura18).

Figura 15 - Estrutura protética prototipada

Figura 16 - Molde da estrutura protética

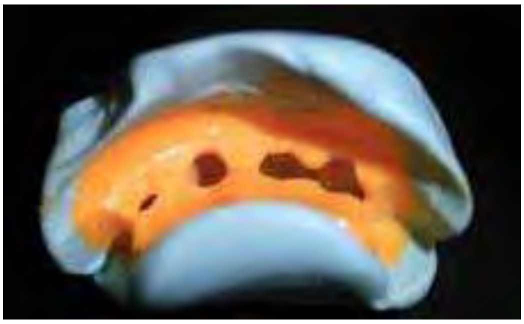


Figura 17 - Modelo com cilindros calcináveis

Figura 18 - Estrutura protética em resina duralay

Para a análise da deformação da estrutura protética utilizando a interferometria holográfica, os modelos pertencentes aos grupos IC, IL e ILIC foram fixados no ramo superior de um articulador, com as incisais dos incisivos apoiadas em uma base metálica.

Uma força de 1,94N, 2,98N e 4,78N foi aplicada para análise dos padrões interferométricos, e as imagens obtidas foram gravadas em uma câmera de vídeo (Figura 19). A partir dessas imagens foi realizado o cálculo do deslocamento direcional.

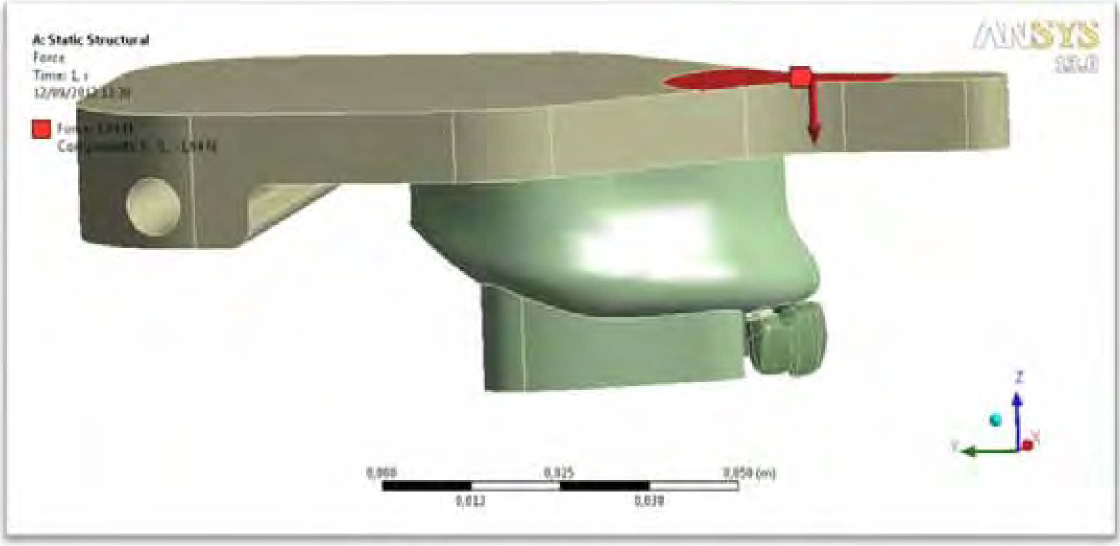
Figura 19 - Aplicação do carregamento no ensaio de interferometria holográfica.

Um modelo 3D, seguindo as mesmas etapas descritas anteriormente (Pré-processamento, processamento e pós-processamento), para a representação das condições aplicadas no ensaio de interferometria holográfica.

As situações estudadas foram as mesmas descritas anteriormente (Grupo IL, Grupo IC, Grupo ILIC).

Um carregamento de 1,94N, 2,98N e 4,78N foi aplicado na parte superior do articulador, como mostra a Figura 20.

Figura 20 - Representação do carregamento.



O resultado do deslocamento direcional nos diferentes grupos com os três carregamentos foram comparados aos obtidos no modelo computacional.

Apêndice

A.2 Sugestões para trabalhos futuros

- Estudo em métodos dos elementos finitos em reabilitação anterior de maxila utilizando implantes de diferentes diâmetros e componentes angulados.

- Estudo *in vivo* utilizando a técnica de subtração radiográfica em pacientes com reabilitação da região anterior de maxila com dois implantes para quatro elementos dentários perdidos, variando o posicionamento desses implantes para comparação com o estudo já realizado em método dos elementos finitos.

Araraquara, 27 de março de 2013.

CÁSSIA BELLOTTO CORRÊA