Critical Aspects for Mechanical Simulation in Dental Implantology

Erika O. Almeida\textsuperscript{1,2} et al.*

\textsuperscript{1}Department of Biomaterials and Biomimetics, New York University College of Dentistry, New York, NY, \textsuperscript{2}Department of Dental Materials and Prosthodontics, Sao Paulo State University College of Dentistry, Araçatuba, SP, \textsuperscript{1}USA \textsuperscript{2}Brazil

1. Introduction

Dental implants have been widely used for the rehabilitation of completely and partially edentulous patients. (Branemark et al. 1969; Branemark et al. 1977; Adell et al. 1990; van Steenberghne et al. 1990) Despite the high success rates reported by a vast number of clinical studies, early or late implant failures are still unavoidable. (Esposito et al. 1998) Mechanical complications and failures have frequently been reported during prosthetic treatment. (Roberts 1970; Randow et al. 1986; Walton et al. 1986; Levine et al. 1999)

From a biomechanical point of view, forces occurring either from functional or parafunctional occlusal contact may result in a physiologic adaptation of the supporting tissues since implants are rigidly anchored to the bone. However, if the stress generated is beyond the adaptive capacity of the host, the response of the supporting tissues and prosthetic components may result in failures. Therefore, the load magnitude and duration employed to implant-restoration systems play a significant role in biomechanical stress dissipation on the implant-prosthesis system and surrounding tissues. (Menicucci et al. 2002) Other factors that are known to affect the stress/strain distribution on bone surrounding implants are the implant position and angulations, implant-abutment connection and the magnitude and direction of the occlusal load. (Stanford and Brand 1999; Kozlovsky et al. 2007; Lin et al. 2009)

\*Amilcar C. Freitas Júnior\textsuperscript{3}, Eduardo P. Rocha\textsuperscript{2}, Roberto S. Pessoa\textsuperscript{4}, Nikhil Gupta\textsuperscript{5}, Nick Tovar\textsuperscript{1} and Paulo G. Coelho\textsuperscript{1}

\textsuperscript{1}Department of Biomaterials and Biomimetics, New York University College of Dentistry, New York, USA
\textsuperscript{2}Department of Dental Materials and Prosthodontics, São Paulo State University College of Dentistry, Araçatuba, SP, Brazil
\textsuperscript{3}Postgraduate Program in Dentistry, Potiguar University, College of Dentistry – UnP, Natal, RN, Brazil
\textsuperscript{4}Department of Mechanical Engineering - Federal University at Uberlândia, FEMEC - Uberlândia, Uberlândia, MG, Brazil
\textsuperscript{5}Department of Mechanical and Aerospace Engineering, Polytechnic Institute of New York University, New York, USA
While randomized controlled clinical studies are strongly suggested (Bozini et al. 2011; Pieri et al. 2011; Turkyilmaz 2011) as the optimal approach to evaluate the performance of biomaterials and biomechanical aspects of dental implants and prosthetic components, often times these studies are not economically viable (especially in cases where multiple variables are to be included). Thus, well designed in vitro studies utilizing virtual models via finite element analysis (FEA) should be considered, as this method allows researchers to address a range of questions that are otherwise intractable due to the number of variations within clinical trials or the difficulty in solving analytically (Ross 2005).

Although there are several implant dentistry studies using the FEA method, (Rayfield 2004; Yokoyama et al. 2005; Galantucci et al. 2006; Huang et al. 2007; Assuncao et al. 2008; Lim et al. 2008; de Almeida et al. 2011) each step of the method deserves discussion in order to facilitate its understanding when applied to implant/restoration/bone system. Thus, the aim of this chapter is to conduct a critical review of simulated biomechanical scenarios describing essential basic aspects and approaches currently used in implantology.

2. Model creation

The first step in creating a finite element (FE) model is to determine the appropriate number of dimension to be utilized for evaluation (1, 2, or 3D). While three dimensions are more realistic for the complex anatomy/implant/restoration biomechanical interaction, it is proportionally more challenging when CAD modeling, solving, and software output interpretation is considered. (Richmond et al. 2005) For selected situations, 2D analyses are often adequate for the questions at hand. (Rayfield 2004; Assuncao et al. 2008; Freitas et al. 2010; Freitas Junior et al. 2010)

The choice between 2D and 3D FEA for investigating the biomechanical behavior of complex structures depends on several factors, including the complexity of the geometry, the type of analyses required, expectations in terms of accuracy as well as the general applicability of the results. (Romeed et al. 2006)

Two-dimensional FEA has previously been used in different areas of dental research. (Burak Ozcelik et al.; Freitas, Rocha et al. 2010; Freitas Junior, Rocha et al. 2010) However, its main limitation is known to be the lower accuracy and reliability relative to 3D and its utilization has drastically fallen from favor when dental treatment biomechanical simulations are considered. (Yang et al. 1999; Yang et al. 2001) In contrast, 3D FEA has acceptable accuracy/reliability while properly capturing the geometry of complex structures. However, the higher the complexity of 3D FE models the higher the difficulty in generating appropriate mesh refinement for simulation, which is more easily achievable for 2D models. (Romeed, Fok et al. 2006)

It is general consensus that 3D reconstruction is essential to study the interaction between anatomy and mechanical behavior of restorative components and dental implants. (Galantucci, Percoco et al. 2006) For that purpose, models may be manually constructed or may be generated from imaging methods such as computed tomography (Figures 1 A and B). The choice for building a model using either a manual or automatic technique depends on the purpose of the study and the structure of interest. Considering the specific morphology of biological structures, the non-uniform rational basis spline (NURBS) has
been commonly used for 3D modeling as it allows the achievement of reliable analytic or freeform parts based on an efficient management of curves and surfaces.

Fig. 1. Models of maxilla (A) and mandible (B) implant supported prosthesis based on (A) model reconstruction from imaging techniques and (B) manual construction on CAD software (Solid Works Corp).

The manual input technique generate structures in appropriate aided design software such as AutoCAD (Autodesk Inc, San Rafael, CA, USA), SolidWorks (SolidWorks Corp., Concord, MA, USA), Pro/Engineer (Wildfire, PTC, Needham, MA, USA), Rhino 3D (McNeel North America, Seatle, WA, USA) (Figure 1B).

Fig. 2. CT scan data as seen in Mimics 13.0 (Materialise, Leuven, Belgium). (A-C) The maxilla is presented in three different cross-sectional views. Masks have been applied according to voxel density thresholding to determine the regions of interest. (D) 3D representation of maxilla as a result of segmentation in Mimics.
The imaging approach involves transforming available medical imaging files from computed tomography (CT) scans, magnetic resonance images (MRI), ultrasound, and laser digitizers into wireframe models that are then converted into FE models. (Romeed, Fok et al. 2006) While laser scans offer a high-resolution representation of the outer surface, these lack information about internal geometry. (Kappelman 1998) Creating models from CT or MRI is often time-consuming but can provide accurate models with fine structural details based on image density thresholding. (Cohen et al. 1999; Ryan and van Rietbergen 2005) The 3D object is automatically created in the form of masks by thresholding the region of interest on the entire stack of scans (Figure 2A-D). The degree of automation and high resolution make this model creation method attractive, but determining the appropriate thresholding algorithms to the bone-air boundary reliably throughout a structure with varying bone thicknesses and density can be challenging. (Fajardo et al. 2002)

When advanced imaging data is used to generate solid models, surface smoothing is advised in order to decrease the number of nodes and elements in the discretized FE model as such an approach generally decreases computation time. (Wang et al. 2005; Magne 2007) Fig. 3A shows an excessive number of elements in a dental implant that was subsequently reduced (Fig. 3B) by computer software (Materialise, Leuven, Belgium). However, it is also advisable that when surface smoothing is performed it does not over simplify the geometries, causing a decrease in solution accuracy.

Fig. 3. (A) 3D CAD of the Nobel Speed™ RP Implant (Nobel Biocare, CA, USA) based on the micro-CT (µCT; CT40, Scanco Medical, Bassersdorf, Switzerland). (B) The Mimics Remesh (Materialise) function “quality preserving reduce triangle” and “reduce triangles” were used to reduce the number of elements at the implant CAD.
3. Material properties and software input limitation

Material properties such as heat conductivity, linear and nonlinear elastic properties, and temperature-dependent elastic properties may be utilized in FEA. (Richmond, Wright et al. 2005) In dentistry, the majority of previous FE scientific communications in implant prosthodontics has considered material properties to be isotropic, homogeneous, and linear elastic. (Huang et al. 2008; Canay and Akca 2009; Eser et al. 2009; Li et al. 2009; Chang et al. 2010; Chou et al. 2010; Okumura et al. 2010; Sagat et al. 2010; Wu et al. 2010; Burak Ozcelik et al. 2011) An isotropic material indicates that the mechanical response is similar regardless of the stress field direction, requiring Young’s modulus (E) and Poisson’s ratio (ν) values for the FE calculation. (Richmond, Wright et al. 2005)

The elastic, or Young’s modulus (E), is defined as stress/strain (σ/Є) and is measured in simple extension or compression. It is a measure of material deformation under a given axial load. In other words, a numerical description of its stiffness. Poisson’s ratio (ν) is the lateral strain divided by axial strain, thus representing how much the sides of a material as it is tensile tested. (Richmond, Wright et al. 2005)

Since bone is one of the structures to be simulated in FEA, (Peterson and Dechow 2003; Bozkaya et al. 2004; Danza et al. 2010; de Almeida et al. 2010) it is often treated as an anisotropic structure and three elastic modulus (E), three Poisson’s ratio (ν) and three shear modulus (G) are required (Table 1). (Chen et al. 2010; Sotto-Maior et al. 2010) Anisotropic materials are characterized by different stress responses under forces applied in varied directions within the structure. (O’Mahony et al. 2001; Natali et al. 2009; Eraslan and Inan 2010) The elastic behavior in cortical bone approximates to orthotropic, which is a type of anisotropy in which the internal structure of the material results in unique elastic behavior along each of the three orthogonal axes of the material. In this case, three elastic (E) and shear modulus (G) and six Poisson’s ratios (ν) are necessary for model input. (Richmond, Wright et al. 2005; Natali et al. 2010)

<table>
<thead>
<tr>
<th>MATERIAL PROPERTIES</th>
<th>TRABECULAR BONE</th>
<th>CORTICAL BONE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ɛₓ (MPa)</td>
<td>1,148</td>
<td>12,600</td>
</tr>
<tr>
<td>Ɛᵧ (MPa)</td>
<td>210</td>
<td>12,600</td>
</tr>
<tr>
<td>Ɛₗ (MPa)</td>
<td>1,148</td>
<td>19,400</td>
</tr>
<tr>
<td>Gₓᵧ (MPa)</td>
<td>68</td>
<td>4,850</td>
</tr>
<tr>
<td>Gᵧₗ (MPa)</td>
<td>68</td>
<td>5,700</td>
</tr>
<tr>
<td>Gₓₗ (MPa)</td>
<td>434</td>
<td>5,700</td>
</tr>
<tr>
<td>νᵧₓ</td>
<td>0,010</td>
<td>0,300</td>
</tr>
<tr>
<td>νᵧₗ</td>
<td>0,055</td>
<td>0,390</td>
</tr>
<tr>
<td>νₗᵧ</td>
<td>0,322</td>
<td>0,390</td>
</tr>
<tr>
<td>νₓᵧ</td>
<td>0,055</td>
<td>0,300</td>
</tr>
<tr>
<td>νₓₗ</td>
<td>0,010</td>
<td>0,253</td>
</tr>
<tr>
<td>νₗₓ</td>
<td>0,322</td>
<td>0,253</td>
</tr>
</tbody>
</table>

Table 1. Material properties used in an anisotropic model. The material axes correspond to the global coordinate system. E = Young’s modulus. G = shear modulus. νᵧₓ = Poisson’s ratio for strain in the y-direction when loaded in the x-direction.
Since in dental implantology bone quality has been related to the structural efficiency of the cortical and trabecular bone architecture and ratio (lower bone quality results in biomechanically challenged treatments), mechanical simulations of poor bone quality critical as clinical studies have shown that dental implants placed in regions of the jaw bones with lower density have a higher chance to fail than implants placed at regions with higher bone density. (Genna 2003)

To date, no consensus regarding the mechanical properties that are appropriate for simulating the different bone density scenarios clinically encountered in implant dentistry has been reached. For instance, the value of trabecular bone elastic modulus observed in the literature range from 0.3 to 9.5 GPa. (Zarone et al. 2003; Eskitascioglu et al. 2004; Sevimay et al. 2005; Yokoyama, Wakahiyashi et al. 2005) A different approach has been employed by Tada and coworkers (Tada et al. 2003) who assigned different elastic moduli to bone depending on its density from most dense (Type 1) to least dense (Type IV). The moduli utilized were 9.5 GPa, 5.5 GPa, 1.6 GPa and 0.69 GPa for bone types I, II, III, and IV, respectively. Recent work has also been carried out using these bone property values for different simulations (Table 2).

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>YOUNG’S MODULUS (GPa)</th>
<th>POISSON’S RATIO</th>
<th>REFERENCES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bone type I</td>
<td>9.5</td>
<td>0.3</td>
<td>Tada et al. (2003)</td>
</tr>
<tr>
<td>Bone type II</td>
<td>5.5</td>
<td>0.3</td>
<td>Tada et al. (2003)</td>
</tr>
<tr>
<td>Bone type III</td>
<td>1.6</td>
<td>0.3</td>
<td>Tada et al. (2003)</td>
</tr>
<tr>
<td>Bone type III</td>
<td>0.69</td>
<td>0.3</td>
<td>Tada et al. (2003)</td>
</tr>
<tr>
<td>Cortical bone</td>
<td>13.70</td>
<td>0.3</td>
<td>Shunmugasamy et al. (2011)</td>
</tr>
<tr>
<td>Titanium</td>
<td>110</td>
<td>0.35</td>
<td>Huang et al. (2008)</td>
</tr>
</tbody>
</table>

Table 2. Mechanical properties of the most used structures in implant prosthodontics research through finite element analysis.

Considering the fact that the most common bone quality types in mandible are type I or II in the anterior region and type III in the posterior region, the elastic modulus that would best represent the mandible would be 9.5 GPa or 5.5 GPa for the anterior region and 1.6 GPa for the posterior region. For the maxillary bone, the most prevalent types of bone are type II or III in the anterior region and type IV in the posterior region. Therefore, the Young’s modulus that would best represent these sites would be 5.5 GPa or 1.6 GPa for the anterior region and 0.69 GPa for the posterior region, respectively. For cortical bone, studies typically use the elastic modulus (E) of 13.7 GPa and the Poisson ratio (ν) similar to 0.3 for the trabecular and cortical bone (Table 2). (Sevimay, Turhan et al. 2005; Huang, Hsu et al. 2008)

4. Bone-implant interface

In implant therapy, the fact that bone quantity and related biomechanical behavior differs for each patient implies a challenge to model the percentage of osseointegration. A critical issue when evaluating a study is to analyze the conditions specified for the interface. What is to be specified is the ability of the interface to resist three different types of stresses: compressive stresses at a right angle to the interface; tensile stresses, also at a right angle to
the interface; and shear stresses in parallel with the interface (Figure 4). (Hansson and Halldin 2009)

Fig. 4. The different types of stresses occurring at the implant–bone interface.

Linear static models have been employed extensively in previous FEA studies. (Kohal et al. 2002; Menicucci, Mossolov et al. 2002; Romanos 2004; Van Staden et al. 2006; Lim, Chew et al. 2008; Eser, Akca et al. 2009; Hsu et al. 2009; Li, Kong et al. 2009; Faegh and Muftu 2010; Freitas Junior et al. 2010; Hasan et al. 2011) These analyses usually assumed that all modeled volumes were bonded as one unit, which indicates that the trabecular and cortical bone are perfectly bonded to the implant interface. (Winter et al. 2004; Maeda et al. 2007; Fazel et al. 2009; Chang, Chen et al. 2010; Okumura, Stegaroiu et al. 2010). However, the validity of a linear static analysis may be questionable when the investigation aims to explore more realistic situations that are generally encountered in the dental implant field. Some actual implant clinical situations will give rise to nonlinearities, mainly related to the change in interrelations between the simulated structures of a FE model. (Wakabayashi et al. 2008) Moreover, frictional contact mode potentially provides an improved accuracy with respect to the relative component’s micromotion within the implant system, and, therefore, a more reasonable representation of the real implant clinical condition. (Merz et al. 2000) This configuration allows minor displacements between all components of the model without interpenetration. Under these conditions, the contact zones transfer pressure and tangential forces (i.e. friction), but not tension. Some FE analyses have shown remarkable differences in the values and even in the distribution of stresses between “fixed bond” and “non-linear contact” interface conditions. (Brunski 1992; Van Oosterwyck et al. 1998; Huang, Hsu et al. 2008) Not only the stress and strain levels but also the stress and strain highly affected by the interface state (Figure 5). Van Oosterwyck et al. (Van Oosterwyck, Duyck et al. 1998) argued that through the bonded interface the force was dissipated evenly in both the compressive and the tension site. However, on the contact interfaces, tensions are not transferred and force is only passed on through the compressive site, which results in excessive stresses. Additionally, the condition of the bone to implant interface also influences the strain distribution and level inside of the implant system. (Pessoa et al. 2010)
Fig. 5. Equivalent strain (µe) distribution for 100N loaded implant in a superior central incisor region, in a median buccopalatal plane. The bone to implant interfaces were assumed as fixed bond (“glued”) (A) and frictional contact (B). The arrows indicate the loading direction for clarity.

Conventionally, the osseointegrated bone to implant interface is treated as fully bonded. (Kohal, Papavasiliou et al. 2002; Menicucci, Mossolov et al. 2002; Romanos 2004; Van Staden, Guan et al. 2006; Lim, Chew et al. 2008; Eser, Akca et al. 2009; Hsu, Fuh et al. 2009; Li, Kong et al. 2009; Faegh and Muftu 2010; Freitas Junior, Rocha et al. 2010; Hasan, Rahimi et al. 2011) This assumption is supported by experimental investigations in which removal of rough implants frequently resulted in fractures in bone distant from the implant surface, (Gotfredsen et al. 2000) suggesting the existence of an implant-bone “bond”. On the other hand, frictional contact elements are used to simulate a nonintegrated bone to implant interface (i.e. in immediately loaded protocols), which allows minor displacements between the implant and the bone. (Pessoa, Muraru et al. 2010) The occurrence of relative motion between implant and bone introduces a source of non-linearity in FEA, since the contact conditions will change during load application.

The friction coefficient (µ) to be used in such simulations depends on many factors including mechanical properties and the roughness of the contact interface, exposure to interfacial contaminants (Williams 2000) and in some cases the normal load. A µ = 0.3 was measured for interfaces between a smooth metal surface and bone, while a µ = 0.45, for interfaces between a rough metal surface and bone. (Rancourt et al. 1990) Frequently, the friction coefficient between bone and implant is assumed as being µ = 0.3. Huang et al. (Huang, Hsu et al. 2008) investigated the effects of different frictional coefficients (µ = 0.3, 0.45 and 1) on the stress and displacement of an immediately loaded implant simulation. The authors
demonstrated that the value of $\mu$ shows no significant influence for increasing or decreasing the tensile and compressive stresses of bone. Nevertheless, increasing $\mu$ from 0.3 to 1, the interfacial sliding between implant and bone was mainly reduced from 20% to 30–60%, depending on the implant design. (Natali, Carniel et al. 2009)

Moreover, when an implant is surgically placed into the jawbone, the implant is mechanically screwed into a drilled hole of a smaller diameter. Large stresses will occur due to the torque applied in the process of implant insertion. As the implant stability and stress state around an immediately loaded implant may be influenced by such conditions, this should be also considered in FE simulation of immediately loaded implants. However, this phenomenon has not been thoroughly investigated to date. The implementation of such implant insertion stresses in FEA is still unclear and should therefore be a matter of further investigations.

Another commonly observed assumption in dental implantology FE models, perfect bonding between implant, abutment and abutment screw also is not the most realistic scenario. Non-linear contact analysis was proven to be the most effective interface condition for realistically simulating the relative micromotions occurring between different components within the implant system. (Williams 2000; Pessoa, Muraru et al. 2010; Pessoa et al. 2010) Therefore, for correct simulation of an implant-abutment connection, frictional contact should be defined between the implant components. Accordingly, between implant, abutment and abutment screw regions in contact, a frictional coefficient of 0.5 was generally assumed in non-linear simulations of implant-abutment connection. (Merz, Hunenbart et al. 2000; Lin et al. 2007; Pessoa, Muraru et al. 2010) When using a contact interface between implant components, for lateral or oblique loading conditions, specific parts can separate, or new parts that were initially not in contact can come into contact. Consequently, higher stress levels may be expected to occur in an implant-abutment connection simulated with contact interfaces, compared to a glued connection. In this regard, the pattern and magnitude of deformation in both periimplant bone and implant components will be influenced by the implant connection design. (Pessoa, Vaz et al. 2010)

5. Mesh and convergence analysis

Since the components in a dental implant-bone system are complex from a geometric standpoint, FEA has been viewed as the most suitable tool for mechanically analyzing them. A mesh is needed in FEA to divide the whole domain into elements. The process of creating the mesh, elements, their respective nodes, and defining boundary conditions is called “discretization” of the problem domain. (Geng et al. 2001; Richmond, Wright et al. 2005)

The 2D structures are typically meshed with triangular or quadrilateral elements (Figure 6A). These elements may possess two or more nodes per side, with one node at each vertex. If nodes are only placed at the vertices, the element is called linear because a line function describes the element geometry and how the displacement field will vary along an element edge. (Richmond, Wright et al. 2005)

Typical 3D elements include eight-node bricks, six-node wedges, five-node pyramids, and four-node tetrahedral (Figure 6B). With increasing numbers of elements and nodes, the model becomes more complex and computationally more difficult and lengthy. (Assuncao et al. 2009)
To address this problem, researchers often devise strategies to minimize computational expense, such as taking advantage of symmetry when possible by modeling only half the structure, using a generally coarse mesh with finer elements only near regions of geometric complexity or high stress and strain, and/or using a 2D model when it suffices. (Richmond, Wright et al. 2005)

A convergence study of the model is always important to verify the mesh quality. A measurement of convergence is the degree of difference in the total strain energy between two successive mesh refinements. (Hasan, Rahimi et al. 2011) When the difference in energy is less than some tolerable limit specified by the user, the solution is considered converged.

**Convergence of Analysis**

![Bar graph showing maximum principal stress (σ_max) vs. number of elements in the model.](image)

Fig. 7. Maximum principal stress (σ_max) in hypothetic convergence testing of mandible FE models meshed by 0.6 mm elements with no refinement (mesh 1) and with refinement levels 2, 3, 4 and 5 at a posterior implant region (meshes 2, 3, 4 and 5, respectively).
Some authors considered that the convergence criterion between meshes refinement was a change of less than 5 (Li, Kong et al. 2009; Lin et al. 2010) or 6% (Huang, Hsu et al. 2008; Huang et al. 2009) in the maximum simulated stress of the bone-implant edge. Alternatively, a mesh can be considered converged when the rate of density change was less than $10^{-5}$ between subsequent iterations. (Hasan, Rahimi et al. 2011)

6. Loading and boundary conditions

A attempting to successfully replicate the clinical situation that an implant might encounter in the oral environment, it is also important to understand and correctly reproduce the natural forces that are exerted throughout the system. These forces are mainly the result of the masticatory muscles action, and are related to the amount, frequency and duration of the masticatory function.

Forces acting on dental implants possess both magnitude and direction, and are referred to as vector quantities. For accurate predictions on the implant-bone behavior, it is essential to determine realistic in vivo loading magnitudes and directions. However, at each specific bite point, bite forces can be generated in a wide range of directions. Also, although bite forces are generally act downward, toward the apex of the implant and thereby tending to compress the implant into the alveolar bone, tensile forces and bending moments may also be present depending on where the bite force is applied relative to the implant-supported prosthesis. This fact is even more important when the investigation aims to simulate multiple-implant treatment modalities, because of the geometric factors involving the restorations that are linking the implants, such as the existence of distal cantilevers. (Mericske-Stern et al. 1996; Fontijn-Tekamp et al. 1998) Nevertheless, although for implants used in single-tooth replacement simulations the in vivo forces ought to replicate the forces exerted on natural teeth, factors such as the width of the crown occlusal table, the height of the abutment above the bone level, and the angulation of the implant with respect to the occlusal plane will affect the value of the moment on the single-tooth implant.

A significant amount of investigations have assumed the direction of the load applied to the implant to be horizontal, vertical and oblique. However, the rationale for use of an oblique loading condition is based on finding the vertical (axial) forces directed to the implant system that are relatively low and well tolerated in comparison to oblique forces, which generate bending moments (Figure 8A). The combination of axial and transverse loading, termed as mixed loading, simulates practical conditions where the actual applied force may be inclined with respect to the implant axis and components can be solved in the longitudinal and transverse directions. (Shunmugasamy et al. 2011) These oblique forces have been considered more clinically realistic in FEA than vertical ones. (Chun et al. 2002; Pierrisnard et al. 2002)

Several investigators have tried to gain insight into implant loading magnitudes by performing tests using experimental, analytical, and computer-based simulations of various implant-supported prosthesis types. (Mericske-Stern, Piotti et al. 1996; Fontijn-Tekamp, Slagter et al. 1998; Duyck et al. 2000; Morneburg and Proschel 2002) Bite forces ranging from 50 to 400 N in the molar regions and 25 to 170 N in the incisor areas have been reported. These variations are influenced by patient's gender, muscle mass, exercise, diet, bite location, parafunction, number of teeth and implants, type of implant-supported prosthesis, physical status and age. (Duyck, Van Oosterwyck et al. 2000; Morneburg and Proschel 2002; Eser, Akca et al. 2009)
Fig. 8. (A) Transversal loading being applied on the lingual surface of the anterior crown. (B) Boundary conditions of the anterior maxilla at the upper and lateral sides. The green region represent the fixed support constrained of $x$, $y$ and $z$ directions (displacement = 0).

Ideally, the entire jawbone structure should be evaluated for its contribution to the force exerted onto the dental implant. However, since the simulation of the whole mandibular and maxillary bone is very elaborate, smaller models have been proposed. (Lin, Chang et al. 2007; Lin et al. 2008; Natali, Carniel et al. 2010; Pessoa et al. 2010). (Pierrisnard et al. 2003; Tada, Stegaroiu et al. 2003) It can be explained by the Saint Venant principles, the actual force system may be replaced by an equivalent load system and the distribution of stress and strains are only affected near the region of loading. (Ugural 2003) Thus, if the interest in the study is on the biomechanical peri-implant environment, the modeling of no more than the relevant segment of the bone is required (Figure 9). This procedure allows saving computing and modeling time. In addition, Teixeira et al. (Teixeira et al. 1998) demonstrated by a 3D FEA that modeling the mandible at a distance greater than 4.2 mm medially or distally from the implant did not result in any significant improvement in accuracy. Hence, besides the application of a proper implant loading, the determination of restrictions to the model displacement compatible with the anatomic segment to be simulated is advised. Obviously, if necessary, expanding the domain of the model could reduce the effect of inaccurate modeling of the boundary condition. (Zhou et al. 1999)

It has been common to apply fixed constraints to the upper region of the maxilla and to its lateral sides during the construction of the FE model to simulate the continuity of the bone (Figure 8B). (Van Staden, Guan et al. 2006) These fixed supports represent the constrained of $x$, $y$ and $z$ directions (displacement = 0). (Chaichanasiri et al. 2009; Hsu, Fuh et al. 2009; Faegh and Muftu 2010; Wu, Liao et al. 2010)

Ishigaki and coworkers (Ishigaki et al. 2003) determined the directions of displacement constrains, which were applied to the jawbone according to the angles of the closing pathways of chopping type and grinding type chewing patters, where the models were constrained at the base of the maxillary first molar to avoid sliding of the entire model.
Shunmugasamy and coworkers (Shunmugasamy, Gupta et al. 2011) simulated different macro-geometries of unit implants and fixed the base of the outer sides of the cortical bone.

Fig. 9. (A) 3D CT-based model of an upper central incisor extraction socket and an implant model positioned inside of the alveolus. (B) Proximal view. (C) Final model: only the relevant segment were included.

7. Validation and interpretation of FE modeling

The validation of the results is the final and most important step in FEA, depending on the degree to which the system reflects its biologic influence reality. (Richmond, Wright et al. 2005) For instance, a model would be precise but inaccurate if the mesh is exceedingly dense but the loading and boundary conditions are unrealistic. (Richmond, Wright et al. 2005)

Validation of FEA entails comparing the behavior of the model with in vivo or in vitro data gathered from parts of the modeled structures. A combination of in vitro and in vivo experimentation potentially offers the best validation. In vitro validation allows one to carefully control the loads and boundary conditions in order to assess the validity of the model’s geometry and elastic properties. (Richmond, Wright et al. 2005) Studies of the consistency of numeric models and their agreement with biologic data are scarce (Mellal et al. 2004; Ozan et al. 2010) and the level of agreement and consistency between different engineering methods, especially those regarding quantified stress/strain, remains a concern. (Iplikcioglu et al. 2003)

8. Analysis criteria involved in steps of the FEA

et al. 2010; Bevilacqua et al. 2011; Burak Ozcelik, Ersoy et al. 2011; de Almeida, Rocha et al. 2011; Okumura et al. 2011), (2) maximum and minimum principal elastic strain ($C_{\text{max}}$ and $C_{\text{min}}$) (Saab et al. 2007; Qian, Todo et al. 2009; Chou, Muftu et al. 2010; Danza, Quaranta et al. 2010; Eser, Tonuk et al. 2010; Limbert et al. 2010; Okumura, Stegaroiu et al. 2010; Pessoa, Muraru et al. 2010; de Almeida, Rocha et al. 2011) and (3) maximum and minimum principal stress ($\sigma_{\text{max}}$ and $\sigma_{\text{min}}$). (de Almeida, Rocha et al. 2010; Degerliyurt et al. 2010; Hudieb et al. 2010; Jofre et al. 2010; Wu, Liao et al. 2010; de Almeida, Rocha et al. 2011)

Although the majority of the studies have used the $\sigma_{\text{vM}}$ for evaluation of bone interface stress, several studies suggest that the magnitudes of the concentrations should be presented in the $\sigma_{\text{max}}$ for evaluation of stress distribution in a brittle structure as bone, (de Almeida, Rocha et al. 2010; Degerliyurt, Simsek et al. 2010; Wu, Liao et al. 2010) as this criterion offers the possibility of making a distinction between tensile stress and compressive stress. (Degerliyurt, Simsek et al. 2010) In addition, displacement components of specific points may provide information about the deformation of the model and facilitate interpretation of the results. Principal stress values of fragile compact bone can be compared with its ultimate compressive strength and ultimate tensile strength values. (Ciftci and Canay 2000; Furmanski et al. 2009)

In fact, interfacial failure and bone resorption under different stress types are attributed to different mechanisms. Accordingly, it may be erroneous to emphasize the peak compressive or $\sigma_{\text{vM}}$ without considering the risks of the tensile and the shear stresses at the interface. (Hudieb, Wakabayashi et al. 2010) However, when the titanium component is available as abutment, screw and implant, which are ductile structures, $\sigma_{\text{vM}}$ is a recommended analysis criterion. (Cattaneo et al. 2005; Degerliyurt, Simsek et al. 2010; Wu, Liao et al. 2010)

Considering the FEA characteristics described above, the main advantage is the virtual simulation of real structures that are difficult to be clinically evaluated, i.e. stress and strain distribution on periimplant bone. Moreover, it is a low cost alternative in comparison to other in vitro methods since only a virtual model is used. Limitations of the FEA models include mainly the patient specific anatomy, and parameters such as the wet environment and damage accumulation under repetitive loading.

### 9. Summary and final remarks

The success of a dental implant depends on a variety of biomechanical factors including the design and position of the implant, implant-abutment connection, cantilever length, surface roughness, bone quality and type, depth of insertion, arch configuration, the nature of bone-implant interface, and occlusal conditions. (Randow, Glantz et al. 1986; Adell, Eriksson et al. 1990; van Steenberghe, Lekholm et al. 1990; Bozkaya and Muftu 2004; Bozkaya, Muftu et al. 2004; Misch et al. 2005; Yokoyama, Wakabayashi et al. 2005; De Smet et al. 2007; Abreu, Spazzin et al. 2010; Turkyilmaz 2011) All these biomechanical factors have been simulated by FEA in previous studies. While an increased number and quality of investigations has been published over the last decade, the results are often contradictory due to differences in model construction and meshing.

The biomechanical behavior of all components used in implant prosthodontics has been regarded as an important factor in determining the life expectancy of the restoration. Although substantial improvement has been made concerning implant/restorative
Critical Aspects for Mechanical Simulation in Dental Implantology

component design, further improvement is achievable through better understanding the different dental implant treatment modalities’ biomechanical behavior. While the field has gained key knowledge in model fabrication (experimental designing) and improvements in all steps here described concerning the basic FEA experimental design, it is expected that all steps will be further refined based on future gains in computer power and correlations made between modeling results and clinical observation, ultimately providing improved care for patients in need of oral rehabilitation.

10. References


Duyck, J., H. Van Oosterwyck, J. Vander Sloten, et al. (2000). Magnitude and distribution of occlusal forces on oral implants supporting fixed protheses: an in vivo study,


Pessoa, R. S., L. Muraru, E. M. Junior, et al. (2010). Influence of implant connection type on the biomechanical environment of immediately placed implants - CT-based


Finite Element Analysis represents a numerical technique for finding approximate solutions to partial differential equations as well as integral equations, permitting the numerical analysis of complex structures based on their material properties. This book presents 20 different chapters in the application of Finite Elements, ranging from Biomedical Engineering to Manufacturing Industry and Industrial Developments. It has been written at a level suitable for use in a graduate course on applications of finite element modelling and analysis (mechanical, civil and biomedical engineering studies, for instance), without excluding its use by researchers or professional engineers interested in the field, seeking to gain a deeper understanding concerning Finite Element Analysis.

How to reference
In order to correctly reference this scholarly work, feel free to copy and paste the following:
