We are IntechOpen,
the world's leading publisher of
Open Access books
Built by scientists, for scientists

6,900
Open access books available

186,000
International authors and editors

200M
Downloads

Our authors are among the

154
Countries delivered to

TOP 1%
most cited scientists

12.2%
Contributors from top 500 universities

CLARIVATE ANALYTICS
**BOOK
CITATION
INDEX**
INDEXED

WEB OF SCIENCE™

Selection of our books indexed in the Book Citation Index
in Web of Science™ Core Collection (BKCI)

Interested in publishing with us?
Contact book.department@intechopen.com

Numbers displayed above are based on latest data collected.
For more information visit www.intechopen.com

2

Finite Element Analysis in Dentistry – Improving the Quality of Oral Health Care

Carlos José Soares[1], Antheunis Versluis[2],
Andréa Dolores Correia Miranda Valdivia[1],
Aline Arêdes Bicalho[1], Crisnicaw Veríssimo[1],
Bruno de Castro Ferreira Barreto[1] and Marina Guimarães Roscoe[1]
[1]Federal University of Uberlândia,
[2]University of Tennessee,
[1]Brazil
[2]USA

1. Introduction

The primary function of the human dentition is preparation and processing of food through a biomechanical process of biting and chewing. This process is based on the transfer of masticatory forces, mediated through the teeth (Versluis & Tantbirojn, 2011). The intraoral environment is a complex biomechanical system. Because of this complexity and limited access, most biomechanical research of the oral environment such as restorative, prosthetic, root canal, orthodontic and implant procedures has been performed in vitro (Assunção et al., 2009). In the in vitro biomechanical analysis of tooth structures and restorative materials, destructive mechanical tests for determination of fracture resistance and mechanical properties are important means of analyzing tooth behavior. These tests, however, are limited with regard to obtaining information about the internal behavior of the structures studied. Furthermore, biomechanics are not only of interest at the limits of fracture or failure, but biomechanics are also important during normal function, for understanding property-structure relationships, and for tissue response to stress and strain. For a more precise interrogation of oral biomechanical systems, analysis by means of computational techniques is desirable.

When loads are applied to a structure, structural strains (deformation) and stresses are generated. This is normal, and is how a structure performs its structural function. But if such stresses become excessive and exceed the elastic limit, structural failure may result. In such situations, a combination of methodologies will provide the means for sequentially analyzing continuous and cyclic failure processes (Soares et al., 2008). Stresses represent how masticatory forces are transferred through a tooth or implant structure (Versluis & Tantbirojn, 2011). These stresses cannot be measured directly, and for failure in complex structures it is not easy to understand why and when a failure process is initiated, and how we can optimize the strength and longevity of the components of the stomatognathic system. The relationship between stress and strain is expressed in constitutive equations

according to universal physical laws. When dealing with physically and geometrically complex systems, an engineering concept that uses a numerical analysis to solve such equations becomes inevitable. Finite Element Analysis (FEA) is a widely used numerical analysis that has been applied successfully in many engineering and bioengineering areas since the 1950s. This computational numerical analysis can be considered the most comprehensive method currently available to calculate the complex conditions of stress distributions as are encountered in dental systems (Versluis & Tantbirojn, 2009).

The concept of FEA is obtaining a solution to a complex physical problem by dividing the problem domain into a collection of much smaller and simpler domains in which the field variables can be interpolated with the use of shape functions. The structure is discretized into so called "elements" connected through nodes. In FEA choosing the appropriate mathematical model, element type and degree of discretization are important to obtain accurate as well as time and cost effective solutions. Given the right model definition, FEA is capable of computationally simulating the stress distribution and predicting the sites of stress concentrations, which are the most likely points of failure initiation within a structure or material. Other advantages of this method compared with other research methodologies are the low operating costs, reduced time to carry out the investigation and it provides information that cannot be obtained by experimental studies (Soares et al. 2008).

However, FEA studies cannot replace the traditional laboratory studies. FEA needs laboratory validation to prove its results. The properties and boundary conditions dentistry is dealing with are complex and often little understood, therefore requiring assumptions and simplifications in the modeling of the stress-strain responses. Furthermore, large anatomical variability precludes conclusions based on unique solutions. The most powerful application of FEA is thus when it is conducted together with laboratory studies. For example, the finite element method can be performed before a laboratory study as a way to design and conduct the experimental research, to predict possible errors, and serve as a pilot study for the standardization protocols. The use of this methodology can also occur after laboratory experimental tests in order to explain ultra-structural phenomena that cannot be detected or isolated. The identification of stress fields and their internal and external distribution in the specimens may therefore help answer a research hypothesis (Ausiello et al., 2001).

The complexity of a FEA can differ depending on the modeled structure, research question, and available knowledge or operator experience. For example, FEA can be performed using two-dimensional (2D) or three-dimensional (3D) models. The choice between these two models depends on many inter-related factors, such as the complexity of the geometry, material properties, mode of analysis, required accuracy and the applicability of general findings, and finally the time and costs involved (Romeed et al., 2004; Poiate et al., 2011). 2D FEA is often performed in dental research (Soares et al., 2008; Silva et al., 2009; Soares et al., 2010). The advantage of a 2D-analysis is that it provides significant results and immediate insight with relatively low operating cost and reduced analysis time. However, the results of 2D models also have limitations regarding the complexity of some structural problems. In contrast, 3D FEA has the advantage of more realistic 3D stress distributions in complex 3D geometries (Fig. 1). However, creating a 3D model can be considered more costly, because it is more labor-intensive and time-consuming and may require additional technology for acquiring 3D geometrical data and generation of models (Santos-Filho et al., 2008).
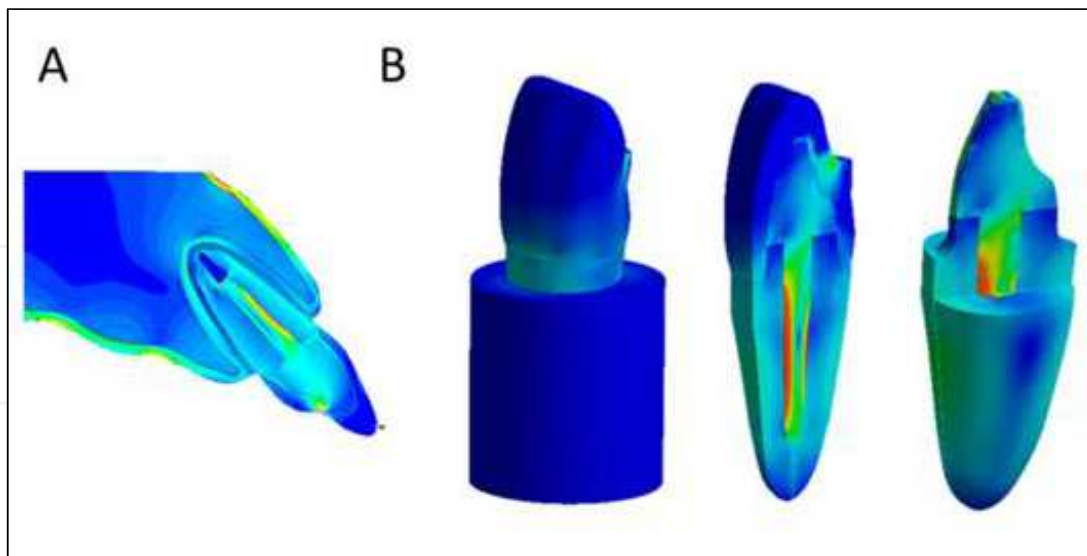
Fig. 1. Upper central incisor restored with cast post-and-cores. A) 2D FEA model, B) 3D FEA model with different cutting planes, showing internal stress distributions (ANSYS 12 Workbench - Ansys Inc., Houston, USA). (Santos-Filho, 2008).

In dental research, FEA has been used effectively in many research studies. For example, FEA has been used to analyze stress generation during the polymerization process of composite materials and stress analyses associated with different restorative protocols like tooth implant, root post canal, orthodontic approaches (Versluis et al., 1996; Versluis et al., 1998; Ausiello et al., 2001; Lin et al., 2001; Ausiello et al., 2002; Versluis et al., 2004; Misra et al., 2005; Meira et al., 2007; Witzel et al., 2007; Meira et al., 2010). This chapter will discuss the application and potential of finite element analysis in biomechanical studies, and how this method has been instrumental in improving the quality of oral health care.

2. Application of finite element analysis in dentistry - Modeling steps: Geometry, properties, and boundary conditions

The FEA procedure consists of three steps: pre-processing, processing and post-processing.

2.1 Pre-processing: Building a model

Pre-processing involves constructing the "model". A model consists of: (1) the geometrical representation, (2) the definition of the material properties, and (3) the determination of what loads and restraints are applied and where. Model construction is often difficult, because biological structures have irregular shapes, consist of different materials and/or compositions, and the exact loading conditions can have a large effect on the outcome. Therefore, the correct construction of a model to obtain accurate results from a FEA is very important. The development of FEA models can follow different protocols, depending on the aim of the study. Models used to analyze laboratory test parameters, like microtensile bond tests, flexural tests, or push-out tests usually have the simplest geometries and can be generated directly into the FEA software (Fig. 2.). Modeling of 2D and 3D biological structures are often more intricate, and may have to be performed with Computer Aided Design (CAD) or Bio-CAD software. This chapter mainly discusses 3D Bio-CAD modeling.
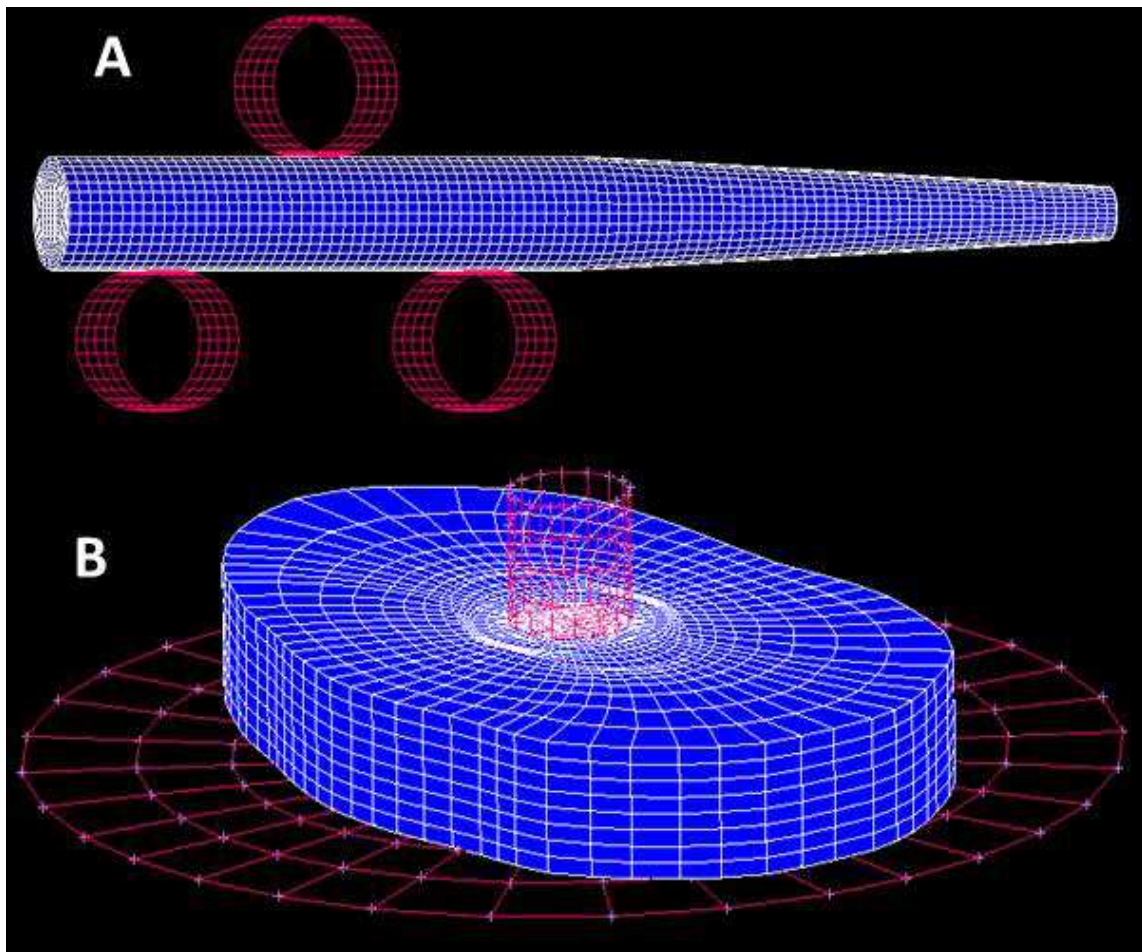
Fig. 2. Finite element models of test specimens made directly in FEA software.

2.2 Bio-CAD protocol for 3D modeling of organic structures

The modeling technique often used in bioengineering studies is called Bio-CAD, and consists of obtaining a virtual geometric model of a structure from anatomical references (Protocol developed in the Center for Information Technology Renato Archer, Travassos, 2010). The obtained geometrical model consists of closed volumes or solid shapes, in which a mesh distribution of discrete elements can be generated. The shape of the object of study can be reconstructed as close to reality as possible, for example, by reducing the size of the elements in regions that require more details. However, higher detail and thus reducing the element sizes will increase the total number of elements and consequently, the computational requirements. Modeling Bio-CAD involves the stages of obtaining the base-geometry, creation of reference curves, construction of surface areas, union of surfaces for generation of solids and exportation of the model to FEA software.

2.2.1 Obtaining the base-geometry

References for model creation, whether 2D or 3D, are images of the structure that is modeled. Modeling of biological structures for the finite element method usually requires CAD techniques. For 2D models, the modeling is made from the images or planar sections of a structure (photograph, tomography or radiograph) (Fig. 3).
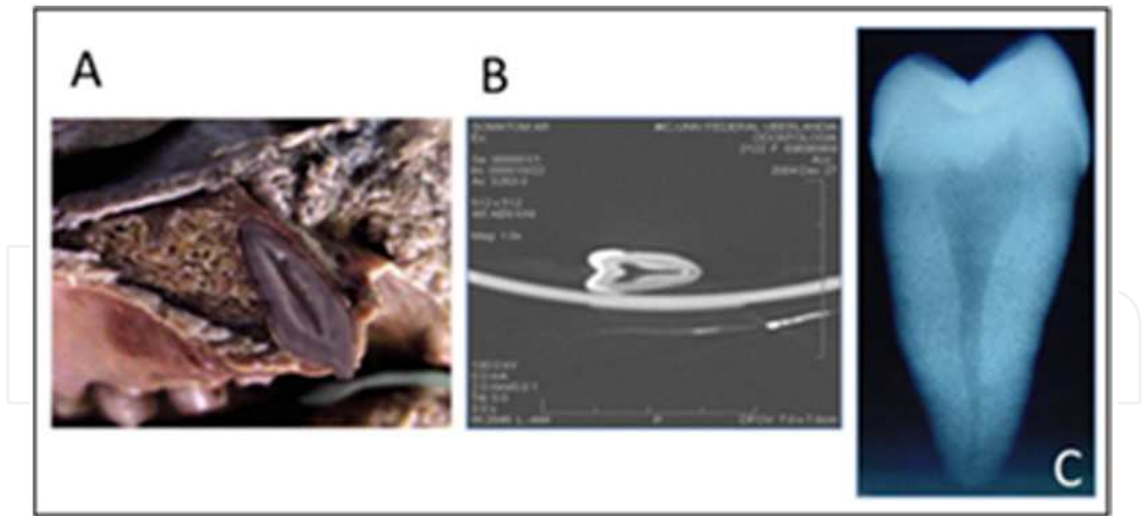
Fig. 3. Images used to build a two-dimensional model. A) Photographs, B) Section plane of computed tomography, C) Radiograph.

These images can be imported into different software programs that can digitize reference points of the structure, such as Image J (available at http://imagej.nih.gov). These points can be exported as a list of coordinates, which can subsequently be imported into a finite element program, for example MENTAT-MARC package (MSC. Software Corporation, Santa Ana, CA, USA), or CAD software, such as Mechanical Desktop (Autodesk, San Rafael, CA, USA) that can generate IGES-files that can be read by most FEA software. The imported reference points can be used to outline the shape of the modeled structure or materials, and hence the finite element mesh.

NURBS Modeling (Non Uniform Rational Bezier Spline) is one of several methods applied for building 3D models. This methodology involves a model creation from a base geometry in STL (stereolithography) format. Obtaining an STL-file, consisting of a mesh of triangular surfaces created from a distribution of surface points, is a critical step for 3D modeling. Several methods have been described in the literature (Magne, 2007; Soares et al., 2008). The
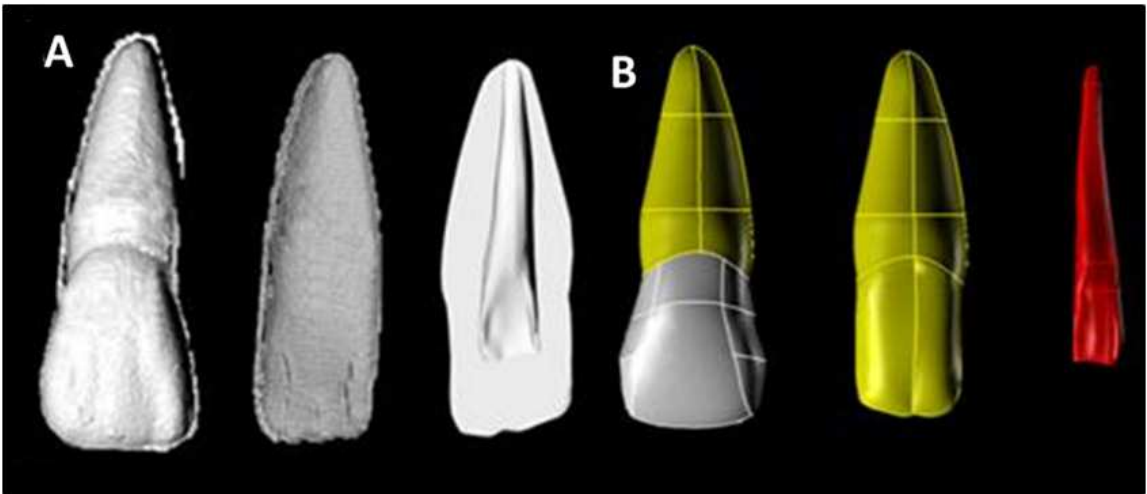

Fig. 4. NURBS modeling. A) STL model (stereolithography), B) NURBS-based geometry created from the STL (Santos-Filho, 2008).

STL file can be obtained by computed tomography, Micro-CT, magnetic resonance imaging (MRI) or optical, contact or laser scanning. Using CAD software, NURBS curves can be defined that follow the anatomical details of the structure. This transformation from surface elements to a NURBS-based representation allows for greater control of the shape and quality of the resulting finite element mesh (Fig. 4).

Our research group has used this strategy to create models of the tooth. First the outer shape of an intact tooth is scanned using a laser scanner (LPX 600, Roland DG, Osaka, Japan). Next, the enamel is removed by covering the root surface with a thin layer of nail polish, and immersing the tooth in 10% citric acid for 10 minutes in an ultra-sound machine. Using a stereomicroscope (40X) the complete removal of enamel can be confirmed. Then the tooth is scanned again and the two shapes (sound tooth and dentin) are fit using PixForm Pro II software (Roland DG, Osaka, Japan). The pulp geometry is generated by two X-ray images obtained from the tooth positioned bucco-lingually and mesio-distally. These images are exported to Image J software where the pulp shapes are traced and digitized, and eventually merged with the scanned tooth and dentin surfaces.

2.2.2 NURBS Modeling: Creation of the curves, surfaces and solids

NURBS Modeling or irregular surface modeling begins with planning the number and position of curves that will represent the main anatomic landmarks of the models, justifying the level of detail in each case. From these curves surfaces and volumes (solids) will be created. The NURBS curves will determine the quality of the model, and consequently, the quality of the finite element mesh. The modeling strategy begins with knowledge of the anatomy of the structure to be studied. The curves should be as regular as possible, and should not form a very small or narrow area with sharp angles, as this would hinder the formation of meshes. The boundary conditions, defined by external restrictions, contact structures and loading definition, must already be defined at the time of construction of lines and surfaces. The curves should provide continuity to ensure that the model will result in closed volumes. If models are made up of multiple solids, NURBS curves from adjacent solids should have the same point of origin to facilitate the formation of a regular mesh across the solid boundaries.

After curves have been defined, surfaces can be created using three or four curves each. The formation of surfaces should follow a chess pattern to prevent wrinkling of the end surfaces caused by the assigned tangency between the surfaces (Fig. 5). This makes it possible to choose the form of tangency between the surfaces and avoid creases in the models that would become areas of mesh complications and consequently locations of erroneous stress concentrations in the final finite element model. It is recommended that there is continuity of curvature between the surfaces. Finally the surfaces should be joined to form a closed NURBS volume.

Most cases involve more than one solid, with different materials and contact areas defined, among other features. In these cases, a classification is assigned to *multi-bodies*. Another important requirement is that there can be no intersection between bodies. There should also be no empty spaces between solids in contact, which in contact analysis would cause single contacts with associated stress peaks, or would cause gaps for intended bonded interfaces. In order to avoid these problems, it is recommended that the contact surfaces of

both bodies are identical and coincide using commands such as *Boolean Operations*, or copying common surfaces.
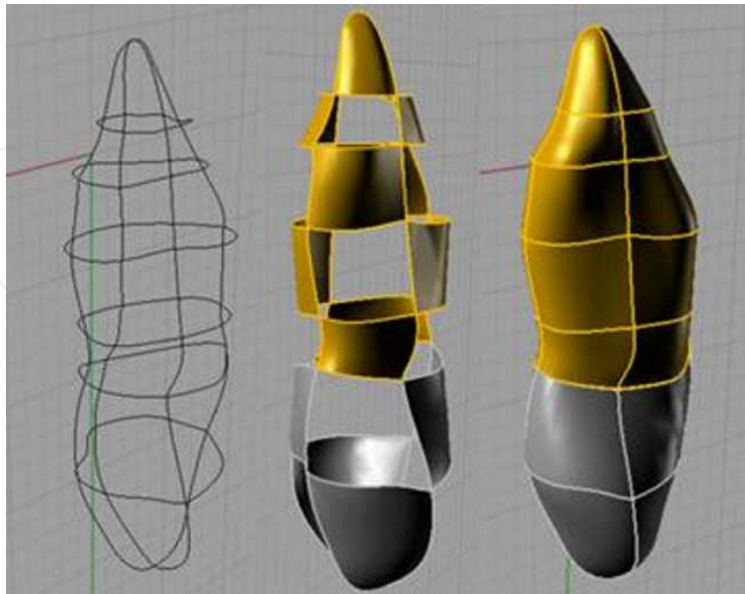

Fig. 5. Creation of surfaces and solids from reference curves in Rhinoceros 3D (Robert McNeel & Associates, USA).

2.2.3 Exporting the solids

The export model is usually saved in STEP (STP) format or IGES (IGS) format. The choice of format depends on the compatibility with the pre-processing software of the FEA program. Also be aware of the units (chosen at the beginning of modeling, usually in millimeters) before importing the model into the pre-processing software. Before exporting, it is recommended to carefully re-check the model to avoid rework: ensure that solids are closed, check for acute angles in surfaces or discontinuities, check for very short edges, check for surfaces that are too narrow or small, and inspect intersections between solids (Travassos, 2010).

2.3 Material properties

Material properties can be determined by means of mechanical tests and applied for any material with the same characteristics. Specimens and procedures can be carried out following agreed testing standards (ASTM - American Society for Testing and Materials). The minimum properties required for most linear elastic isotropic finite element analyses are the elastic modulus and Poisson's ratio.

2.3.1 Methods for obtaining material properties used in FEA

The elastic modulus (E) represents the inherent stiffness of a material within the elastic range, and describes the relationship between stress and strain. The elastic modulus can thus be determined from the slope of a stress/strain curve. Such relationship can be acquired by means of a uniaxial tensile test in the elastic regime (Chabrier et al., 1999). The modulus of elasticity is defined as:

$$E = \sigma/\varepsilon \qquad (1)$$

where (σ) is the stress and (ε) is the strain (ratio between amount of deformation and original dimension).

Various methods have been used to measure the elastic modulus (Chung et al., 2005; Vieira et al., 2006; Boaro et al., 2010; Suwannaroop et al., 2011). For dental materials and tissues, the classical uniaxial tensile test is often problematic due to small specimen dimensions dictated by size, cost, and/or manufacturing limitations. Therefore, other methods such as 3-point bending, indentation, nanoindentation and ultrasonic waves have been used to determine the elastic modulus. Using a Knoop hardness setup, the elastic modulus of composites can be estimated with an empirical relationship, yielding a simple and low cost method (Marshall et al., 1982). Using the dimensions of the short and long diagonals of the indentation, the elastic modulus (GPa) can be determined using the following equation:

$$E = 0.45 \; KHN/((0.140647\text{-}d/D) \; 100) \qquad (2)$$

where KHN is the Knoop Hardness (kg/mm^2), d is the short diagonal of the indentation, D is the long diagonal of the indentation, and 0.140647 is the ratio of the short and long diagonals of the Knoop indenter (1/7.11). Nanoindentation systems have also been used for this purpose. The elastic modulus from nanoindentation is obtained from the data generated in the load-displacement curve by means of the equation (Suwannaroop et al., 2011):

$$1/E^* = (1\text{-}v2)/E + (1\text{-}v2)/E' \qquad (3)$$

where E* is the reduced modulus from the nanoindenter, E is the modulus of the Berkovich diamond indenter (1,050 GPa), E' is the modulus of the specimen, υ is the Poisson's ratio for the indenter (0.07)[28], and υ' is the Poisson's ratio for the specimens.

The ISO 4049 (Dentistry - Resin based dental fillings) provides a standard for the use of three-point bending tests for determining the flexural modulus (elastic modulus) for composites. Generally, the preparation of specimens for microindentation tests is easier, specimen size is smaller and it has been suggested that their results are more consistent than with an ISO 4049 three-point bending test (Chung et al., 2005).

The analysis of anisotropic materials (i.e., materials with different stress-strain responses in different directions) requires the application of elastic moduli and Poisson's ratios in 3 directions (2 in case of orthotropy), as well as shear moduli in those directions. It is well accepted that enamel is not isotropic, but the anisotropy of dentin is less well established. Analyzing the effect of anisotropy in dentin, the presence and direction of dentinal tubules were not found to affect the mechanical response, indicating that dentin behaved homogeneous and isotropic (Peyton et al., 1952). More recently, some heterogeneity and anisotropy was demonstrated for dentin. However, the stiffness response seems to be only mildly anisotropic (Wang & Weiner, 1998; Kinney et al., 2004; Huo, 2005). Therefore, dentin properties in FEA are usually assumed to be isotropic. Potential simplifications such as the assumption of linear-elastic isotropic material behavior may be necessary in FEA simulations due to the difficulty of obtaining the correct directional properties, or the need to reduce the complexity of an analysis. As in other research approaches, some simplifications and assumptions are also common in FEA, and are permissible provided that their impact on the conclusions is carefully taken into account. It has been shown, for

example, that the assumption of isotropic properties for enamel did not change the conclusions of a shrinkage stress analysis (Versluis & Tantbirojn, 2011).

The other mandatory property for a FEA, the Poisson's ratio, is the ratio of lateral contraction and longitudinal elongation of a material subjected to a uniaxial load (Chabrier et al., 1999). Among the static methods are tensile and compression tests, in which a uniaxial stress is applied to the material and the Poisson's ratio is calculated from the resulting axial and transverse strains. Another method uses ultrasound (resonance), where the Poisson's ratio is obtained from the speed or natural frequency of the generated longitudinal and transverse waves.

2.3.2 Type of structural analysis: Linear and nonlinear analysis

The type of structural analysis depends on the subject that is being modeled. Depending on the model, the FEA can be linear or nonlinear. Linear or nonlinear analysis refers to the proportionality of the solutions. A solution is linear if the outcome is independent of its loading history. For example, an analysis is linear if the outcome will be the same irrespective of if the load is applied in one or multiple increments. Some conditions are inherently nonlinear, such as nonlinear material responses (e.g., rate-dependent properties or viscoelasticity, plastic deformation), time-dependent boundary conditions (e.g., contact analysis where independent bodies interact), or geometric instabilities (e.g., buckling). Sometimes linear conditions become nonlinear when general assumptions become invalid. For example, the stress-strain responses are generally based on the assumption of small displacements. When large deformations occur, the numerical solution procedures must be adjusted. Most high-end FEA software programs have the capability to resolve nonlinear equation systems. For the end-user, the difference between submitting a linear or nonlinear analysis is minimal, and usually only involves the prescription of multiple increments or invoking an alternative solver for the ensuing nonlinear solution. Since nonlinear systems potentially have multiple solutions, nonlinear analyses should also be checked more thoroughly for the convergence to the correct solution. Nonlinear solutions require more computational iterations to converge to a final solution, therefore nonlinear analyses are more costly in terms of computation and time.

Nonlinear FEA is a powerful tool to predict stress and strain within structures in situations that cannot be simulated in conventional linear static models. However the determination of elastic, plastic, and viscoelastic material behavior of the materials involved requires accurate mechanical testing prior to FEA. The experimental determination of mechanical properties continues to be a major challenge and impediment for more accurate FEA. For example, periodontal ligament (PDL) is a dental tissue structure with significant viscoelastic behavior, and simulation using nonlinear analysis would be more realistic. However, due to its complex structure; the exact mechanical properties of PDL must still be considered poorly understood. It such case it can be argued that using incorrect or questionable nonlinear mechanical properties in a FEA may be more obscuring than a well defined and understood simplification.

An example of the need for a nonlinear analysis is the simulation of the mechanical behavior across an interface. Interfacial areas are among the most important areas for the performance of materials or structures. Interfaces between different materials can often be

modeled as a perfect bond, where nodes are shared across the interface (Wakabayashi et al., 2008). The simulations of such interface can normally be conducted in a linear analysis. However, depending the actual conditions at a simulated interface, such perfect fusion can occasionally lead to unrealistic results. Fig. 6 shows a 2D FEA model of a root filled tooth restored with a cast post and core and a fiberglass post and composite core. When perfect bonding was assumed in a linear analysis, the stress distribution indicated higher stress concentrations in the cast post and core compared to the fiberglass post, while the stress distribution in the root dentin was nearly identical between these two models. Experimental failure data showed, however, that the failure modes of the cast post and core group were more catastrophic and involved longitudinal root fractures while all fractures of the root with fiberglass post were coronal fractures. Simulating the interface more realistically with friction between resin cement and cast post and core (requiring a nonlinear analysis) rather than a perfect fusion, the stress distribution changed substantially between the two post types (Fig. 6), and yielded more realistic results when compared with the experimental observations.
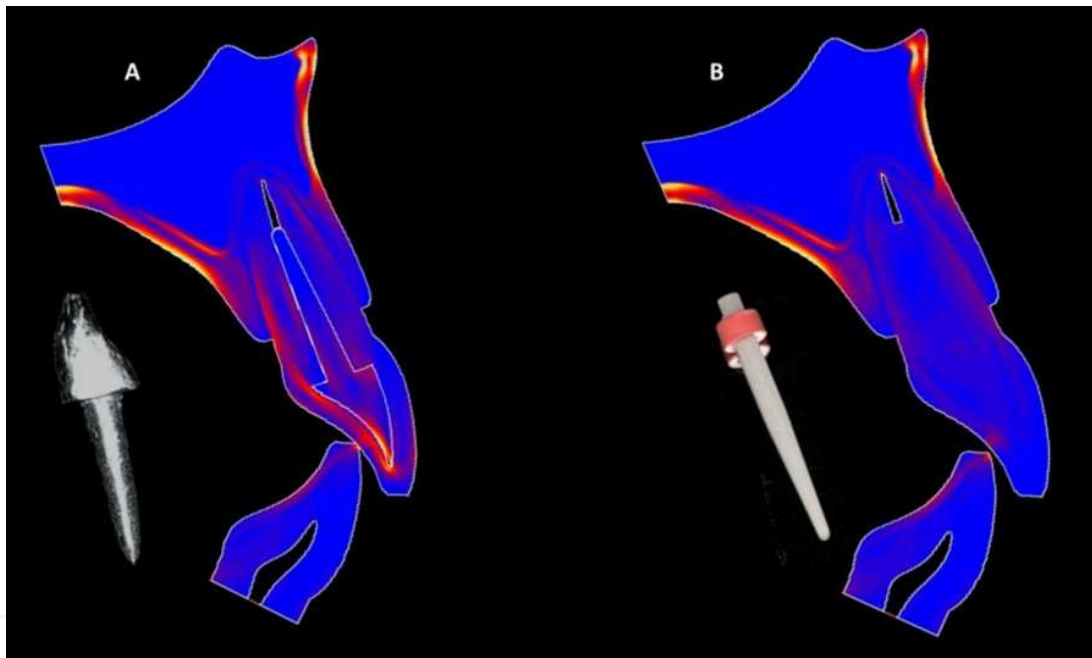

Fig. 6. Nonlinear FEA of endodontic treated tooth restored with A. Cast post and core and B. Fiberglass post.

2.4 Mesh generation

In FEA the whole domain is divided into smaller elements. The collection and distribution of these elements is called a mesh. Elements are interconnected by nodes, which are thus the only points though which elements interact with each other. The process of creating an element mesh is referred to as "discretization" of the problem domain (Geng et al., 2001).

There are many different types of elements. One of the differences can be their basic shape, such as triangular, tetrahedral, hexahedral, etc. Triangular or tetrahedral elements are popular because automatic meshing software routines are easier to develop and thus more advanced for those shapes. Automatic generation of element distributions is especially

useful in bioengineering, which often deals with irregular geometries. Since few modeled geometries have perfectly square dimensions or even straight edges, element shapes must be adapted to fit. Note that the accuracy of elements deteriorates the further they are distorted from their ideal basic shape. Besides their basic geometrical shapes, elements can differ in the way they are solved, such as linear or quadratic interpolation. This refers to how stress and strain is interpolated within an element.
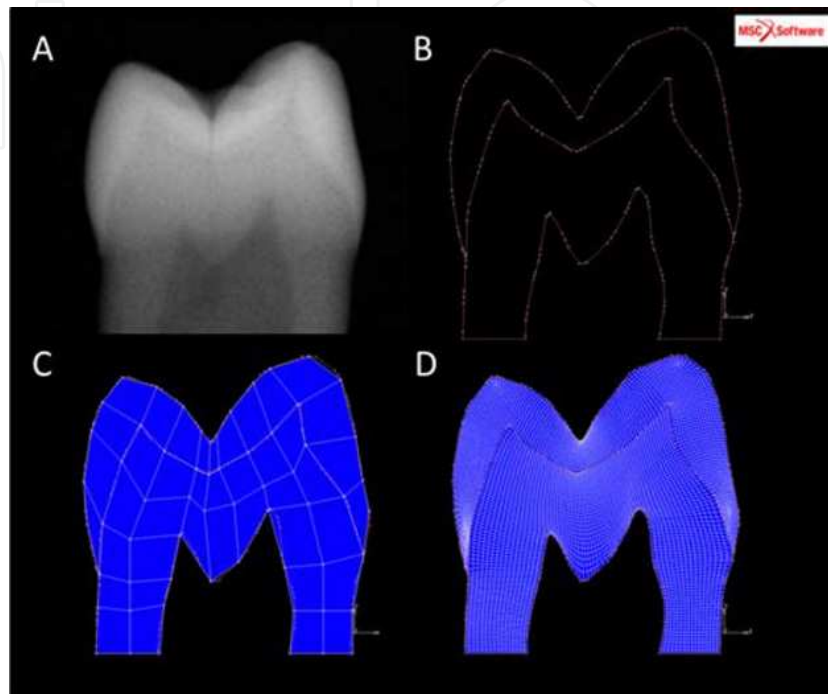

Fig. 7. Radiography of maxillary premolar (A); Lines plotted in the MARC/MENTAT software (B); Manual creation of mesh (C); Final subdivision for improving the mesh quality (D).

Most FEA software provides *automesh* or automatic mesh generation options. The program may suggests the size and number of elements or allow manual control for generating the element meshes. Manual mesh generation can give good results for 2D models (Fig. 7). However, most 3D models rely on automated mesh generators because manual creation of 3D models is very time-consuming. Still, various aspects of the 3D automeshing need to be controlled manually, such as the number of elements required in a given pre-selected area, the distribution of elements, the range of element sizes within a model, uniting or dividing elements, etc. The manual controls also allows selective distribution of elements, for example, more refined meshes in special regions of interest (contact interfaces, geometric discontinuities) while creating coarser mesh distributions in regions of less interest (Fig. 8).

In finite element modeling, a finer (denser) mesh should allow a more accurate solution. However, as a mesh is made finer and the element count increases, the computation time also increases. How can you get a mesh that balances accuracy and computing resources? One way is to perform a convergence study. This process involves the creation and analysis of multiple mesh distributions with increasing number of elements or refinements. When results with the various models are plotted, a convergence to a particular solution can be found. Based on this convergence data, an estimation of the error can be made for the

various mesh distributions. A mesh convergence study can thus be used to find a balance between an efficient mesh distribution and an acceptably accurate solution within the limitations of the computing resources. Moreover, a convergence test can verify if an obtained solution is true or if it was an artifact of a particular element distribution.

2.5 Boundary conditions

The boundary conditions define the external influences on a modeled structure, usually loading and constraints. Boundary conditions are associated with six degrees of freedom (DOF). The combination of all boundary conditions of a FEA model must represent the procedural conditions to which the actual structure that is simulated is subjected. The choice and application of boundary conditions is extremely important, because they determine the outcome of the FEA.

2.5.1 Prescribed displacement – Fixation and symmetry

In a simple way, restrictions can be summarized as the imposition of displacements and rotations on a finite element model, which can be either null or have fixed values or rates. These restrictions concern three rotations (around X, Y, Z-axes) and three translations (in X, Y, Z-directions). Static analysis requires sufficient fixation of a model to remain in place. Insufficient fixation will lead to instability and failure to reach a numerical solution in the FEA. Since nodes are the points through which elements communicate, boundary conditions are usually applied to nodes, where in a 3D model each free node has 6 degrees of freedom (3 translations and 3 rotations). Although some FEA software may allow application of boundary conditions to element edges or surfaces, they are extrapolated to the associated nodes. To achieve the fixation mimicking a support system in real life, for example complete immobilization of a modeled specimen in a test fixture, displacement constraints can be applied to nodes located in a region equivalent to those of the real support system. Symmetry can be viewed as a form of fixation. Since all displacements are mirrored, the displacement across the symmetry-axis is zero.

2.5.2 Load application

The application of loads in a FEA model must also represent the external loading situations to which the modeled structure is subjected. These loads can be tensile, compressive, shear, torque, etc. To simulate the masticatory forces, loads have been applied using different methods, for example point loads, distributed loads across a specific area, and by means of a simulated opposing cusp of the antagonist tooth (Fig. 9). A point load application may result in high stress concentrations around the loaded nodes, creating unrealistic stress concentrations. In reality, a masticatory contact force is likely to be distributed across certain contact areas on both the buccal and lingual cuspal inclines. However, the most realistic load application is not always the best choice for all research questions. Contact areas move depending on stiffness and thus deformation of both opposing teeth. If contact areas change, contact loads change also, which can have significant effect on the stress distribution. When a research question requires well-defined load conditions, point loads or prescribed distributed loading may be better choices than the seemingly more realistic simulated tooth contact.
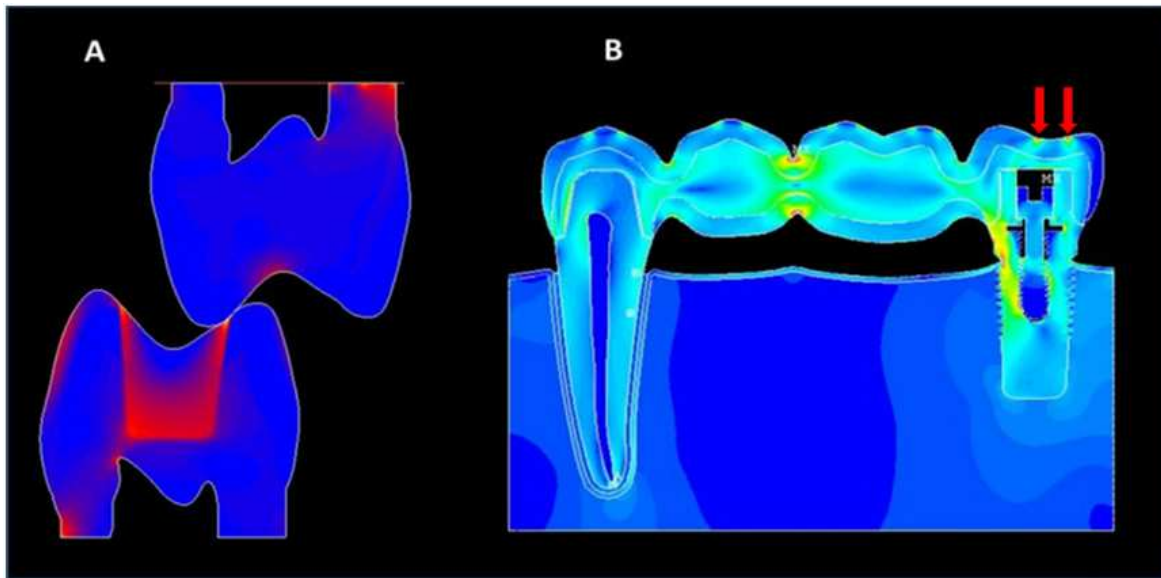
Fig. 9. Load application A. Contact analysis using antagonist tooth (Marc/Mentat software); B. Point load application (Ansys Inc., Houston, USA).

3. Evaluation of finite element analysis

FEA results can be intimidating, the amount of generated data (displacements, strains, stresses, temperatures, etc) is almost unlimited. However, one of the most powerful features of FEA is that results can be easily visualized and made accessible. Visualization of the results can be done by showing data distributions using a colour scale, where each colour corresponds to a range of values. Furthermore, deformations and displacements can be shown by comparing the original unstressed model outlines with the outline of the model under stress. Based on this immediate visualized output, the operator can investigate the displacement of the structure, the type of movement that was performed, which region has a higher dislodgement, or how to redistribute the stresses in the analyzed structure in either three dimensions or in two-dimensions. Such a structural analysis allows the determination of stress and strain resulting from external force, pressure, thermal change, and other factors. This section discusses how results from a finite element analysis should be evaluated, starting with a check of the model, followed by checking the outcome. Then the relationship between finite element and experimental will be discussed with respect to the limitations of either method.

3.1 Analysis of coherency

All finite element analyses should first be checked for coherence (or sanity). The first step of a coherence analysis is to visualize the displacements and deformations to verify that the simulated model moves in the expected direction. The second step of a coherence analysis is to analyze if the distribution stresses is as expected.

3.2 Validation of the outcome

After the model definition is confirmed to be sound, the validity of the outcome still has to be validated. A finite element analysis is modelled based on geometric, property, and

boundary conditions, each of which may have required significant assumptions or simplifications. The purpose of validation is thus to confirm that the general response of the model is realistic. Unlike stress, which cannot be measured directly, deformation or displacement can be directly measured. Therefore, displacement is often a good choice for comparing the simulated behaviour with the behaviour observed in reality, even if the simulation was a "stress" analysis. Validation can be achieved by comparing the outcome with published results from validated analyses or with laboratory measurements. Examples are strain gauge measurement, cuspal flexure, bending displacements, etc. Effects of stresses can also be indirectly validated through the observation of their expected effects, such as crack initiation and fractures. It is not realistic to expect an exact fit between experimental and numerical results, because even between experimental results there will not be an exact fit due to natural and experimental variations. Therefore it is important to remember that it is not an exact fit that validates a finite element analysis but rather the similarity in general tendencies (Versluis et al., 1997).

3.3 Interpretation of the results

After a finite element analysis has been checked and validated, it can be used to interpret the research question. Most finite element analysis results should be interpreted qualitatively. Quantitative interpretation can sometimes be justified, provided all input is verifiable and quantitatively validated. The current state of the art of the use of finite element modelling in dentistry indicates that predictive results are still best viewed in a qualitative manner. It is the search for optimal balance between the objectives of a study, computational efforts (accuracy and efficiency), and practical limitations that ultimately determines the value of a finite element model. Since most finite element models are linear, errors in magnitudes of the loads will not have a direct effect on qualitative predictions. However, small changes in types of boundary conditions such as the location of the loading can substantially alter even the qualitative performance predictions.

Biomechanical performance involves efficient function as well as failure. One of the failure mechanisms is loss of structural integrity, which can eventually result in loss of function. FEA can be used in the determination of fracture mechanics parameters, and examination of experimental failure test methods. In the dental FEA literature, failure is usually extrapolated from maximum stress values, where stress concentrations are identified as possible locations for failure initiation and relative concentration values are interpreted as related to the failure risk. When using this process for interpreting failure behaviour, it is important to carefully assess the stress concentration locations because they may depend heavily on the chosen modelling and boundary condition options (Korioth & Versluis, 1997). Furthermore, stress distributions change when a crack propagates. Therefore, researchers should be extremely cautious about extrapolating crack behaviour based on the distribution of stress concentrations from a static analysis.

Interfacial stress was previously noted as an important area that needs careful interpretation in FEA. For example, stress analyses of the tooth–restoration complex have been performed to predict failure risks at the interfaces as well as stresses transferred across such interfaces. Usually such interfaces are modelled as perfectly bonded, where tooth and restoration elements share the same node. Depending on the accuracy of this assumption, this may lead to erroneous interpretation of the results of a finite element analysis (Srirekha & Bashetty,

2010). Sometimes interfacial interactions are more complex, such as areas where two materials may contact but do not bond. In such cases contact analysis needs to be simulated which will require FEA software that can perform nonlinear analyses.

Most research protocols, including finite element analyses, will have limitations. Some limitations in finite element modelling are a deliberately choice. For example, although teeth are 3D structures, often they are modelled in 2D. Two-dimensional models offer excellent visual access for pre- and post-processing, improving their didactic potential. Furthermore, because of the reduced dimensions, more computational capacity can be preserved for improvements in element and simulation quality or functional processes such as masticatory movements. On the other hand, 3D models, although geometrically more realistic, may give a false impression of accuracy, because they are generally more coarse, contain elements with compromised shapes, and examination or improvement of the model is far more difficult (Korioth & Versluis, 1997).

3.4 Relationship between finite element and experimental analysis

The finite element method is sometimes viewed as a less time-consuming process than experimental research, and therefore could minimize laboratory testing requirements. For some applications, finite element analysis may provide faster solutions, for example for the testing of parameters, which can be changed more easily in FEA than in laboratory experiments. However, due to the complexity of shape, properties, and boundary conditions of dental structures, comprehensive modelling can also quickly becomes very complex and time-consuming. Finite element analysis should be viewed in combination with experimental methods, not as a substitute. Finite element analysis can provide information that would be difficult or impossible to obtain with experimental observations, but at the same time, finite element analysis cannot be performed without experimental input and validation.

Compared with experiments, FEA has clear limitations. These limitations are mainly due to the many factors that contribute to the mechanical response but are still poorly understood. Such lack of understanding usually does not affect experiments, if their outcomes are simply considered as phenomena. However, FEA is the compilation of our understanding of physical laws and material properties, expressed in a theoretical model that describes the interactions between the various factors. Therefore, phenomena are no acceptable input for a theoretical model. Limitations in FEA therefore most often refer back to our own lack of understanding the reality. In other words, our own limitations in understanding are the cause of limitations in FEA. Our inability to accurately describe and simulate biomechanical dynamics and properties of a tooth and its supporting structures limits the accuracy of our FEA models. Fortunately, even imperfect experimental or FEA testing methods can improve our insight and continuously expand our understanding of reality. Therefore, although certain differences may remain between reality and the analyses we conduct using the finite element method, the numerical approach can approximate, for example, otherwise inaccessible stress distributions within a tooth-restoration complex. Furthermore, the ability to visualize many of the results from finite element analyses has also undoubtedly helped researchers to more clearly convey their data, and helped expand the discussion and dissemination of research findings that have contributed to improve oral health.

4. Impact of finite element analysis on dentistry - How FE analyses have contributed to improved oral health

Oral health is important to an individual's well-being and overall health. In dentistry, most oral diseases are neither self-limiting nor self-repairing (Vargas & Arevalo, 2009). Therefore, prompt professional care is fundamental, given that oral diseases follow a downward spiral: incipient diseases requiring minimum dental care, if untreated, progress into diseases that require increasingly more complex and expensive treatments; increases in complexity and cost usually make the treatment even more out of reach for a large proportion of the population (Vargas & Ronzio, 2002). In this context, finite element analysis has been applied in various areas in dentistry (1) to improve the understanding of these complex processes and (2) to help to design better procedures.

4.1 Non-Carious Cervical Lesions (NCCL)

FEA has been used in the investigation of NCCL (Michael et al., 2009). Although the etiology of NCCL remains a controversial subject, there is a general consensus that the process is multi-factorial, and that stress can be one of the factors. Goel et al. (1991) investigated stresses arising at the dentino-enamel junction during function and noted that the shape of the dentino-enamel junction was different under working cusps than non-working cusps. Tensile stresses were elevated toward cervical enamel where the mechanical inter-locking between enamel and dentin is weaker than in other areas of the tooth, making it susceptible to cracking, which could contribute to cervical caries (Goel et al., 1991). Finite element analyses have usually assumed the NCCL across the CEJ (Fig, 10). A recent study, however, did not find clinical evidence of enamel loss above the occlusal margin of NCCL,
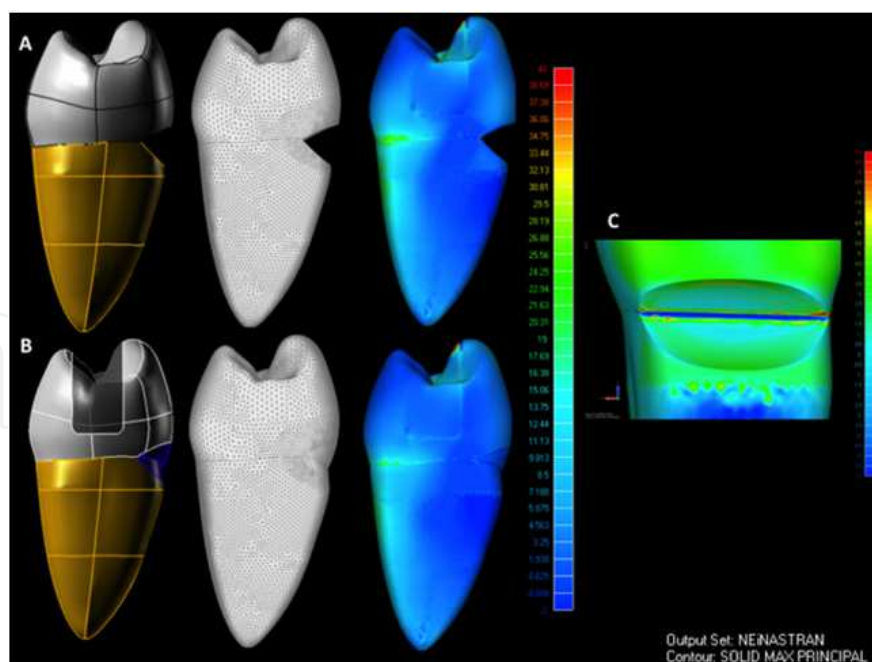

Fig. 10. A. 3D Model of FEA analysis of a non-carious cervical lesions not restored; B. 3D Model of FEA analysis of a non-carious cervical lesions restored with composite resin; C. Maximum principal stress distribution at the unrestored non-carious cervical lesion (Pereira FA, 2011).

except for fracture of enamel that was undermined by the NCCL (Hur et al., 2011). Since the location of the lesion will affect the stress conditions, combining clinical observations and finite element modeling will be essential to determine the stress factor in the initiation and development of NCCL.

4.2 Endodontic treatment

In the case of dental caries, the decay process can continue until the destruction of the tooth and the compromise of adjacent tissues. As the caries process progresses without some type of intervention, the pulp ultimately becomes involved and the root canal therapy is required (Vargas & Arevalo, 2009). One of the steps in root canal treatment is to completely fill the root canal system. During root canal preparation, many variables are outside the control of the clinician (natural root morphology, canal shape and size, dentine thickness) other factors can be addressed during treatment to reduce fracture susceptibility. Using finite element analysis, Versluis et al. (2006) demonstrated that the potential for fracture susceptibility may be reduced by ensuring round canal profiles and smooth canal taper (Fig. 11). Even when fins were not contacted by the instrument, stresses within the root were lower and more evenly distributed than before preparation. Rundquist & Versluis (2006) also used FEA to demonstrate that with increasing taper, root stresses decreased during root filling but tended to increase slightly during a masticatory load. Based on the simulation of vertical warm gutta-percha compaction and a subsequent occlusal load, they suggested that root fracture originating at the apical third was likely initiated during filling, whilst fracture originating in the cervical portion was likely caused by occlusal loads. Gutta-percha is the
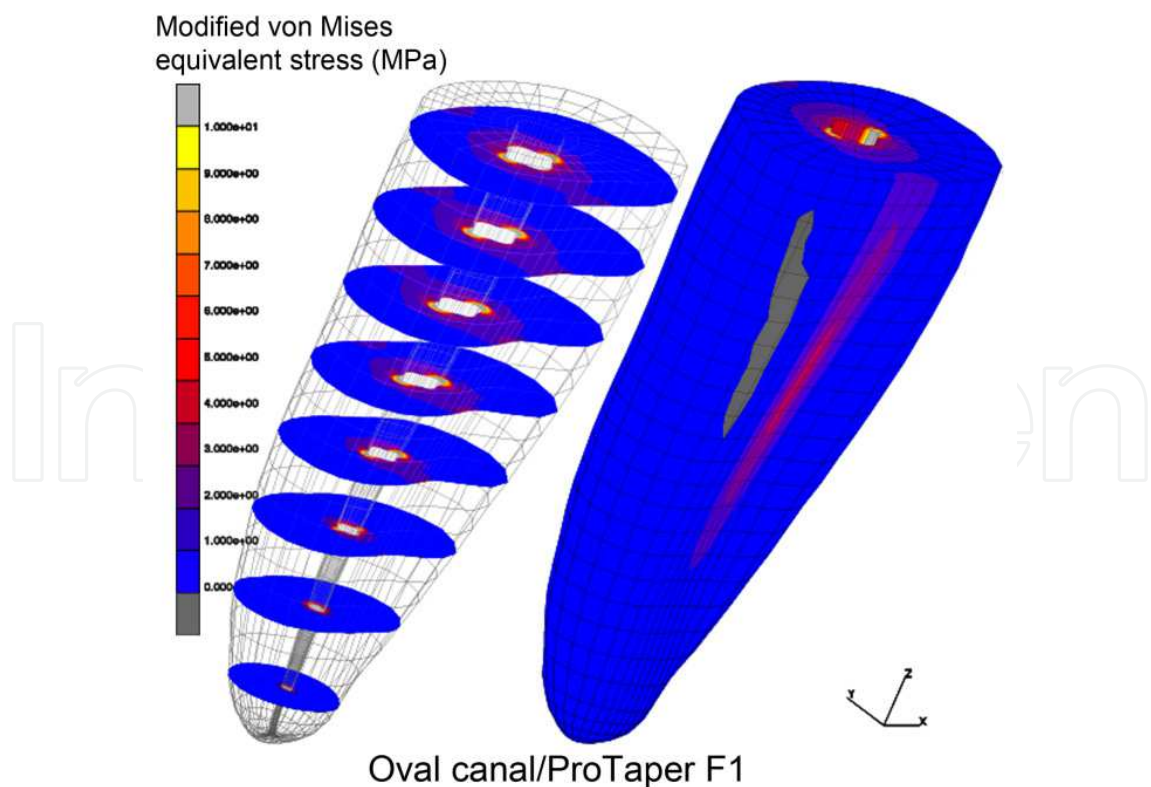

Fig. 11. Stress distribution during obturation pressure in a root with oval canal, cleaned with ProTaper F1 (Versluis et al., 2006).

most common core material used (Er et al., 2007). Although the softening of gutta-percha by heat is a widely used technique, the use of high levels of heat can lead to complications. When heat compaction techniques are used, the procedure should not harm the periodontal ligament (Budd et al., 1991). The use of the technique may result in an unintentional transmission of excessive heat to the surrounding periodontal tissues (Er et al., 2007). Excessive heat during obturation techniques may cause irreversible injury to tissues (Atrizadeh et al., 1971; Albrektsson et al., 1986). By using a three-dimensional thermal finite element analysis the distribution and temperatures were evaluated in a virtual model of a maxillary canine and surrounding tissues during a simulated continuous heat obturation procedure (Er et al., 2007).

4.3 Restoration of root filled teeth

Endodontically treated teeth are compromised by coronal destruction from dental caries (Ross, 1980), fractures (Soares et al., 2007), previous restorations (Schatz et al., 2001), and endodontic access (Soares et al., 2007). How these compromised teeth should be reconstructed to regain their original fracture resistance has been the subject of many studies investigating restoration types and benefits of posts (Fokkinga et al., 2005; Salameh et al., 2006; Salameh et al., 2007). It is not sufficient to only measure an endpoint such as fracture resistance to fully understand the effect of restoration type and post application. A more comprehensive analysis is thus needed to determine the optimal procedures for reconstructing endodontically treated teeth (Soares et al., 2008). The biomechanical conditions that lead to fracture are characterized by the stress state in a tooth, which can be assessed by finite element analysis (Fig. 12). Soares et al. (2008) therefore used FEA to investigate the stress distribution in an endodontically treated premolar restored with composite resin with or without a glass fiber post system and concluded that the use of glass fiber posts did not reinforce the tooth-restoration complex. Intraradicular retention should
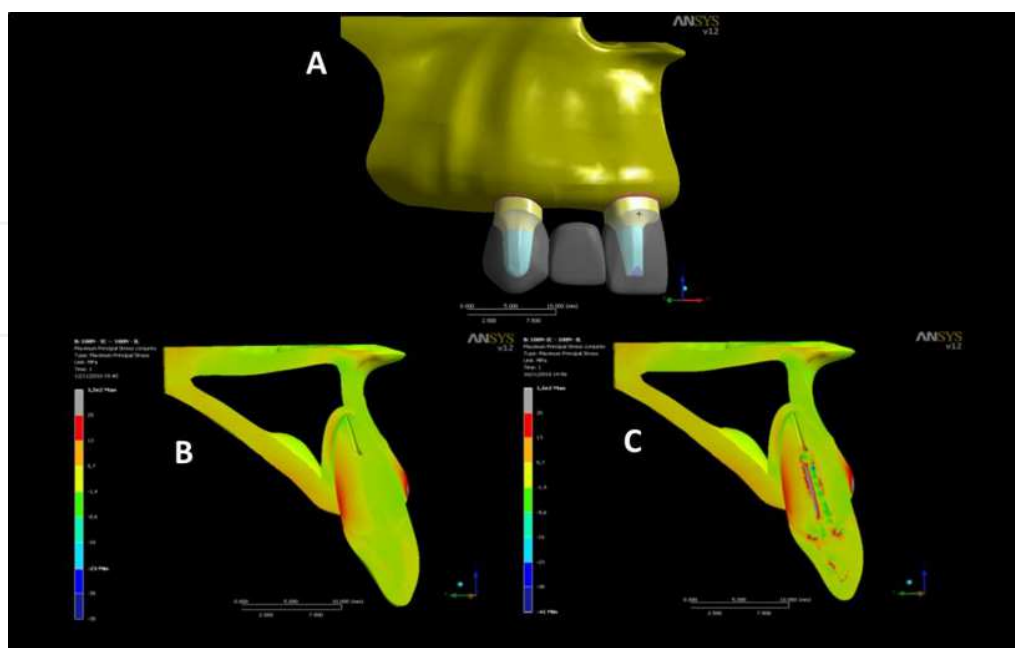

Fig. 12. A. 3D Model of FEA analysis of a 3 elements fixed prosthesis regarding the effect of post type (B. fiber glass post; C. Cast post and core) (Silva GR, 2011).

thus be indicated for endodontically treated teeth that have suffered excessive coronary structure loss (Yu et al., 2006). Research studies using FEA concluded that the use of post systems that have an elastic modulus similar to that of dentine result in a mechanically homogenous units with better biomechanical performance (Barjau-Escribano et al., 2006; Silva et al., 2009). Some studies have concluded that the attributes of carbon and glass fiber dowels make them suitable for dowel restoration (Glazer, 2000; Lanza et al., 2005). Dowel length, size, and design have also been shown to influence the biomechanics and stress distribution of restored teeth (Barjau-Escribano et al., 2006). Using finite element analysis it is possible to evaluate the influence of the type of material (carbon and glass fiber) and the external configuration of the dowel (smooth and serrated) on the stress distribution of teeth restored with varying dowel systems (Soares et al., 2009). Moreover, the difference in elastic modulus between dentin, intraradicular retainers, and cements could result in stress concentrations at the restoration interface when the tooth is in function (Soares et al., 2010, Silva et al., 2011).

4.4 Restorative procedures

In the field of operative dentistry, FEA seems to be an appropriate method for obtaining answers about the interferences caused by the restorative process in a complete structure, for optimizing the design of dental restorations and for evaluating stress distributions in relation to different designs. Many materials are available for dental restorations. The selection and indications for direct and indirect restorative materials involve esthetic, financial, and anatomic considerations, as well as important factors such as analysis of the biomechanical characteristics of the restorative materials, and the amount and state of remaining tooth structure (Soares et al., 2008). In recent years, the demand for nonmetal dental restorations has grown considerably. Metal-free reinforced restorative systems have become popular because of the less favorable esthetic appearance of metal ceramic crowns (Gardner et al., 1997). The primary advantages of nonmetal alternatives (composite resins and ceramics) are improved esthetics, the avoidance of mercury, and cost effectiveness (Stein et al., 2005). Composite resin and ceramic restorations retained with an adhesive resin are the most popular restorations currently used. Composite resins have mechanical properties similar to dentin (Willams et al., 1992) while ceramic has an elastic modulus similar to that of enamel (Albakry et al., 2003).

The conservation of dental structure is crucial to offering fracture resistance, since the removal of dentin reduces the structural integrity of a tooth and causes alteration in stress distributions (Soares et al., 2008b). In this context, the use of adhesive restorations is recommended for reinforcing remaining dental structure (Soares et al., 2008b, Versluis & Tantbirojn, 2011). By using the finite element analysis, stress distributions could be accessed within endodontically treated maxillary premolars that lost tooth structure and the effect of the type of restorative material used for restorations could be studied (Soares et al., 2008). The use of directly placed adhesive restorative materials, such as composite resin, and indirectly placed restorations, such as ceramic inlays, cemented with adhesive materials, generally reduced stress concentrations in comparison with amalgam restorations (Soares et al., 2008). Although indirect restorations may be recommended, the dentist still faces to the choice of geometric configuration of the cavity preparation (Soares et al., 2003).

Inlays and onlays are the 2 technical choices for indirect restorations (Fig. 13). Some studies have shown that after endodontic treatment, teeth restored with intracoronal restorations show more severe fracture patterns (Hannig et al., 2005; Soares et al., 2008c). However, it is unclear whether bonded intracoronal restorations should be used for large defects and which material is the most indicated. In this context, Soares et al. (2003) evaluated the cavity preparation influence on the stress distribution of molar teeth restored with esthetic indirect restorations. The stress distribution pattern of the sound tooth was compared to several different extensions of preparation for inlay, onlay and overlay restored with ceramic or ceromer materials. The cavity preparation extension was significant only for onlays covering one cusp and for overlays. Ceramic restorations had higher stress concentrations, while ceromer restorations caused higher stresses in the tooth structure (Soares et al., 2003).
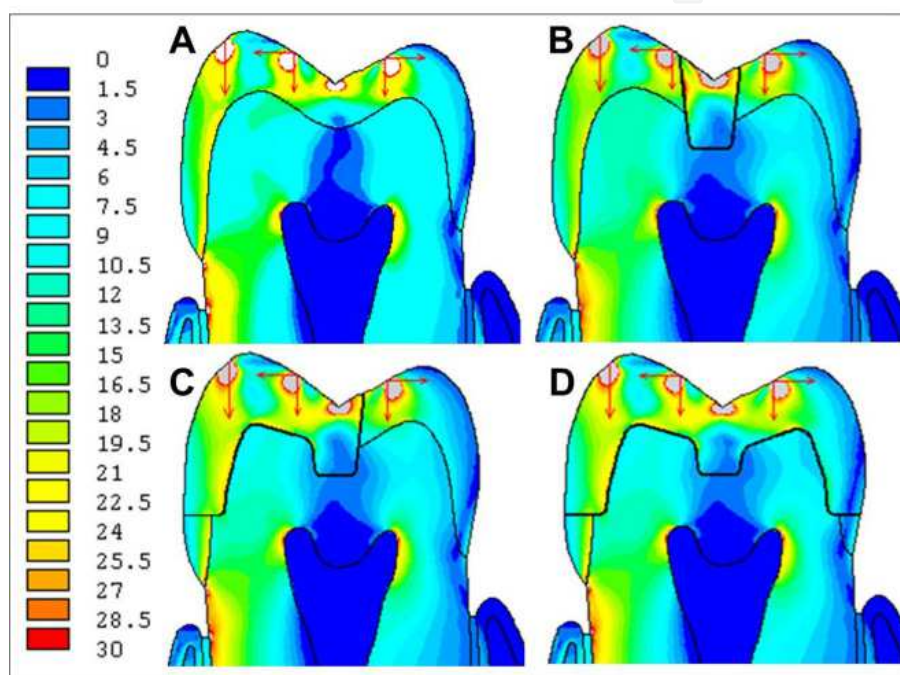

Fig. 13. FEA of different cavity restoration designs for ceramic indirect restorations. A. Intact tooth, B. Inlay restoration; C. Onlay convering buccal cusps; D. Overlay ceramic. (Von mises Stress distribution)

The routine use of metal-free crowns has resulted in an increasing number of fractured restorations (Bello & Jarvis, 1997). Increased fracture resistance of ceramic systems when metal reinforcement was eliminated, has been obtained by the addition of chemical components such as aluminum oxide, leucite, and lithium disilicate (Mak et al., 1997; Drummond et al., 2000). Considering that any restoration has a risk of fracture, the finite element analysis provides a method to evaluate stress distributions in different ceramic systems under occlusal forces. Various studies investigating the performance of ceramic restorations have been performed. Using the finite element analysis method, some investigators (Hubsch et al., 2000; Magne et al., 2002; Magne, 2007; Dejak & Mlotkowski, 2008) demonstrated that ceramic inlays reduced tension at the dentin-adhesive interface and may offer better protection against debonding at the dentin restoration interface, compared with the composite resin inlay. In this context, Reis et al. (2010) investigated, through a 3D finite element analysis, the biomechanical behavior of indirect restored maxillary premolars

based on type of preparation (inlay or onlay), and restorative material (composite resin, resin laboratory, reinforced ceramic with lithium disilicate or reinforced ceramic with leucite). Materials with higher modulus of elasticity transfer less stress into the tooth structure. However, materials with modulus of elasticity much larger than the dental structure caused more severe stress concentrations. The models that used reinforced ceramics with leucite showed a behavior that was biomechanically closest to healthy teeth (Reis et al., 2010).

4.5 Composite and resin cement shrinkage

Resin-composite materials have been widely and increasingly used today in adhesive dental restorative procedures (Fagundes et al., 2009). An important advantage over metallic filling materials is the well-known possibility of bonding the restoration to dental tissues (Marques de Melo et al., 2008) and a significant disadvantage of many of these materials are still the polymerization shrinkage (Pereira et al., 2008). The clinical concern about polymerization shrinkage is evident from the large number of publications and large number of controversial opinions about this topic (Versluis et al., 2004). Shrinkage stress has been associated with various clinical symptoms, including fracture propagation, microleakage and post-operative sensitivity, none of which are direct measures of shrinkage stress. Since stress cannot be measured directly, the presence of shrinkage stresses can only be quantified through indirect manifestations, in particular tooth deformation (Tantbirojn et al., 2004).

Various methods have been used to estimate residual shrinkage stresses, ranging from extrapolated shrinkage or load measurements in vitro to stress analyses in tooth shaped anatomies using photoelastic or finite element methods (Kinomoto et al., 1999; Ausiello et al., 2001). Determination of shrinkage stress is difficult, because it is a transient and nonlinear process. The amount of stress after polymerization therefore depends on the correct description of all changes in mechanical properties and their sequence. Moreover, stress is not a material property or even a structural value, because stress is a three-dimensional local tensor (system of related vectors) that is determined by the combination of multiple material properties and local conditions. Since finite element analysis performs its calculation based on such input (mechanical properties, geometry, boundary conditions), it is eminently suitable for studying residual shrinkage stress in dental systems. On the other hand, as the input for especially the mechanical properties remains to be determined more comprehensively, any polymerization shrinkage predicted by finite element analysis should be validated experimentally using indirect factors that can be measured, such as displacement.

Using such validated finite element analyses, shrinkage stresses in restored teeth (enamel and dentin) were found to increase with increasing restoration size, while stresses in the restoration and along the tooth-restoration interface decreased (Versluis et al., 2004). This outcome was explained by the change in tooth stiffness: removal of dental hard tissue decreases the stiffness of the tooth, causing the tooth to be deformed more by the shrinkage stresses (higher stress in the tooth) and causing less resistance to the composite shrinkage (lower stress in the composite). As this example shows, shrinkage stresses are generated in the adhesive interface as well as in the composite and in the residual tooth structure (Versluis et al., 2004; Ausiello et al., 2011).

Restoration placement, techniques are widely recognized as a major factor in the modification of shrinkage stresses. Various techniques, ranging from incremental composite placement to light-exposure regimes, have been advocated to reduce shrinkage stress effects on a restored tooth. Using finite element analysis, it was shown that even during restoration, cavities deform, and thus that incremental application of composite may end up with a higher tooth deformation than a bulk filling (Versluis et al., 1996). Recently the interaction between incremental filling technique, elastic modulus, and post-gel shrinkage of different dental composites was investigated in a restored premolar. Sixteen composites, indicated for restoring posterior teeth, were analyzed. Two incremental techniques, horizontal or oblique, were applied in a finite element model using experimentally determined properties. The calculated shrinkage stress showed a strong correlation with post-gel shrinkage and a weaker correlation was found with elastic modulus. The oblique incremental filling technique resulted in slightly lower residual shrinkage stress along the enamel/composite interface compared to the horizontal technique. However horizontal incremental filling resulted in slightly lower stresses along the dentin/composite interface compared to the oblique technique (Soares et al., 2011). FEA has been used also to analyze the residual shrinkage stress of resin cement used to cement a ceramic inlay, recently we proved that resin cement polymerized immediately after cementation produced significantly more residual stress than when was delayed for 5 minutes after setting ceramic inlay and polymerization (Fig. 14).
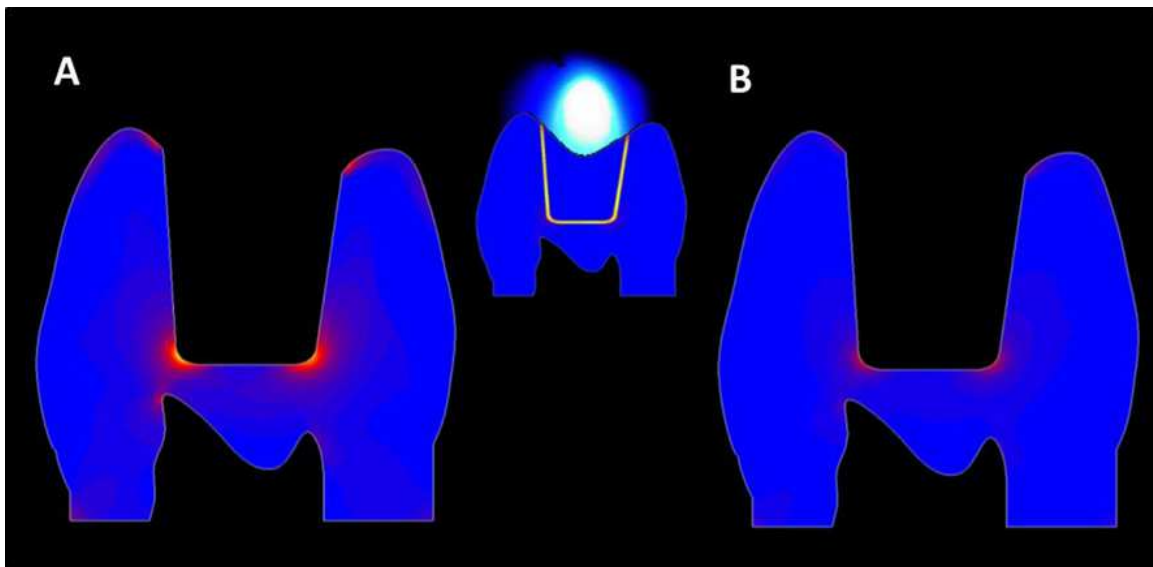

Fig. 14. FEA of residual shrinkage stress of resin cement used to cement a ceramic inlay. A. resin cement polymerized immediately after cementation; B. Reduction of shrinkage stress with delay for 5 minutes after setting ceramic inlay and polymerization.

An often used experimental test for measuring shrinkage forces uses a cylindrical composite specimen bonded between two flat surfaces of steel, glass, composite, or acrylic rods. Even for such seemingly simple experimental tests, understanding the outcome can be difficult. Although one may expect that for a specific experimental set-up, differences in the measured force could be attributed to the composite properties, particularly shrinkage and elastic modulus, it was found that the relative ranking of a series of materials was affected by differences in system compliance. As a result, different studies may show different

rankings and may draw contradictory conclusions about polymerization stress, shrinkage or modulus (Meira et al., 2011). Finite element analysis can help to better understand the test mechanics that cause such divergences among studies. Using an FEA approach, a commonly used test apparatus was simulated with different compliance levels defined by the bonding substrate (steel, glass, composite, or acrylic). The authors showed that when shrinkage and modulus increased simultaneously, stress increased regardless of the substrate. However, if shrinkage and modulus were inversely related, their magnitudes and interaction with rod material determined the stress response (Meira et al., 2011).

4.6 Periodontology and implantology

Another oral problem with high prevalence, mainly in adults, is periodontal disease. ''Periodontal disease'' is a generic term describing diseases affecting the gums and tissues that support the teeth (Thomson et al., 2004). A periodontal compromised tooth can be diagnosed from probing depth, mobility, supporting bone volume, crown-to-root ratio, and root form (Grossmann & Sadan, 2005). It is generally accepted that a reduction of periodontal support worsens the prognosis of a tooth. However, the morphology of the periodontum with reduced structural support has not been well understood in relation to clinical functions, such as load-bearing capability (Ona & Wakabayashi, 2006). To determine the interaction of reduced periodontal support with mechanical function, one must determine the stress and strain created in the periodontum in accordance with the morphologic alteration of the structures (Ona & Wakabayashi, 2006). Finite element analysis can be used for such assessment, and of the influence of progressive reduction of alveolar support on stress distributions in periodontal structures (Ona & Wakabayashi, 2006). The stress in the periodontum could also predict the potential pain and damage that may occur under functional bite force (Kawarizadeh et al., 2004).
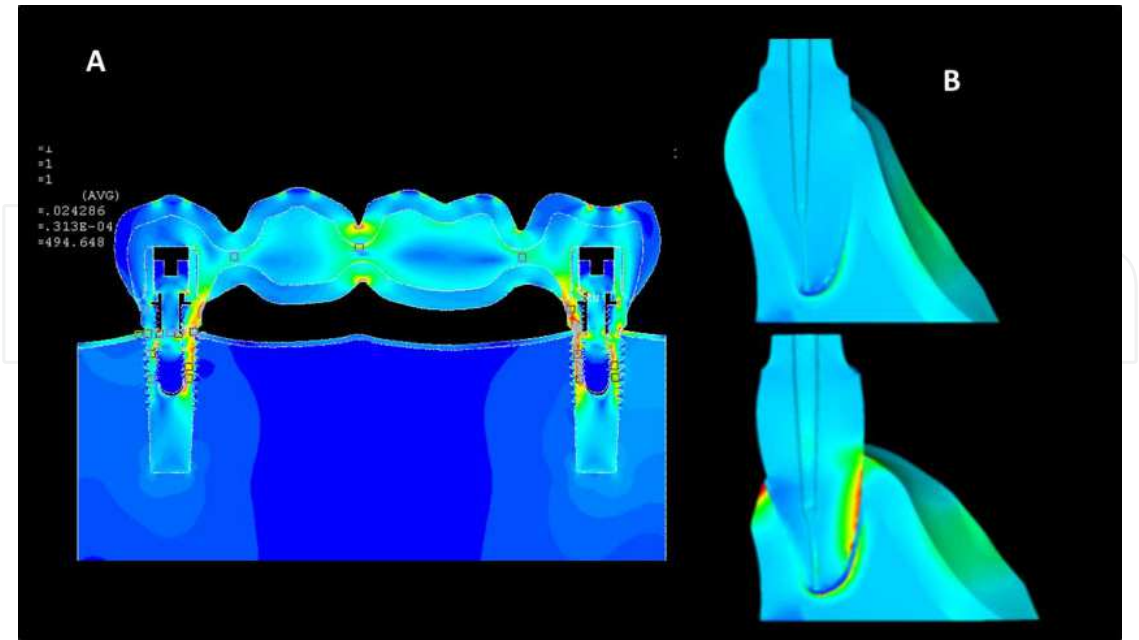

Fig. 15. FEA analysis of implant prosthesis demonstrating the stress concentration on the mesial region of the interface between implant and prosthesis. B. FEA analysis of canine restored with fiber glass post and its effect on bone loss (Roscoe MG, 2010).

Historically, periodontal disease is one of the main causes of tooth loss (Deng et al., 2010). Traditionally, patients with severe periodontitis have ultimately had all teeth removed due to severe alveolar bone resorption and high risks for systemic infections (Deng et al., 2010). In this context implant therapy has been applied successfully for three decades, and proven to be a successful means for oral rehabilitation (Albrektsson et al., 1986). The knowledge of physiologic values of alveolar stresses provides a guideline reference for the design of dental implants and it is also important for the understanding of stress-related bone remodeling and osseointegration (Srirekha & Bashetty, 2010). Stiffness of the tissue-implant interface and implant-supporting tissues is considered the main determinant factor in osseointegration (Ramp & Jeffcoat, 2001; Turkyilmaz et al., 2009). Finite element analysis has been used extensively in the field of implant research over the past 2 decades (Geng et al., 2001). It has been used to investigate the impact of implant geometry (Himmlova et al., 2004), material properties of implants (Yang & Xiang, 2007), quality of implant-supporting tissues (Petrie & Williams, 2007), fixture-prosthesis connections (Akca et al., 2003), and of implant loading conditions (Natali et al., 2006).

4.7 Trauma and orthodontics

Beyond caries and periodontal disease, orofacial trauma is also considered a public health problem (Ferrari & Ferreria de Mederios, 2002). Finite element analysis has also been widely used for dental trauma analysis (Huang et al., 2005). In the real world traumatic injuries to teeth typically result from a dynamic force (Huang et al., 2005). Therefore, for traumatic analysis of a tooth, it has been recommended to simulate time-dependent behavior and analyze different rates of loading (Natali et al., 2004). Finite element analysis can provide insight into the process of impact stresses and fracture propagation in teeth subjected to dynamic impact loads in various directions.
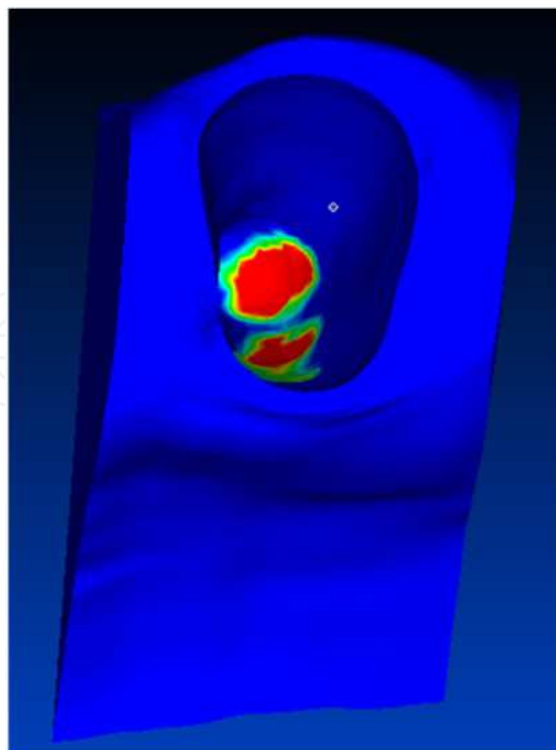

Fig. 16. FEA analysis of orthodontic intrusion movement of a maxillary canine.

Finite element analysis has also been used to study the biomechanics of tooth movement, which allows accurate assessment of appliance systems and materials without the need to go to animal or other less representative models (Srirekha & Bashetty, 2010). Orthodontic tooth movement is a biomechanical process, because the remodeling processes of the alveolar support structures that result in the tooth movement are triggered by orthodontic forces and moments and their consequences for the stress / strain distribution in the periodontium. The redistribution of stresses and strains causes site-specific resorption and formation of the alveolar bone and with it the translation and rotation of the associated tooth (Cattaneo et al., 2009). Finite element analysis can provide insight into the stress and strain distributions around teeth with orthodontic loading to help orthodontists define a loading regime that results in a maximal rate of tooth movement with a minimum of adverse side-effects. The main challenges for the application of finite element analysis in orthodontics has been the definition of the mechanical properties of the periodontal ligament (Toms et al., 2002) and to move beyond the currently most common practice of static finite element models.

4.8 Summary: How FE analysis contributes to improve oral health

It is often commented that finite element analysis is a powerful tool for the interpretation of complex biomechanical systems. Yet, all clinicians and dental researchers are acutely aware of the complexity of oral tissues and their interactions, and hence of the limitations of any theoretical model that depends on input from our incomplete knowledge. The reason why FEA is nonetheless considered such a powerful tool is that it does not need perfect input to be already extremely useful. FEA helps researchers and clinicians formulate the right research questions, design appropriate experiments, and through the underlying universal physics that form the basis of FEA it provides an almost instant insight into complex biomechanical relationships (cause and effect) that cannot be easily obtained or communicated with any other method. The expanded insight and understanding of mechanical responses have undeniably been of direct significance for justifying experimental questions and improving clinical treatments.

As the preceding examples show, finite element analysis not only offers solutions for the engineering problems, but it has been instrumental in the progress in many areas of dentistry. Finite element analysis has improved the understanding of complex processes and has assisted researchers and clinicians in designing better procedures to maintain oral health. Finite element simulation provides unique advantages for dental research, such as its precision and its ability to solve complex biomechanical problems for which other research methods are too cumbersome or even impossible (Ersoz, 2000). Finite element simulation allows more comprehensive prediction and analysis of medical processes or treatments because in a process where many variables need to be considered, it allows for manipulation of single parameters, making it possible to isolate and study the influence of each parameter with more precision (Sun et al., 2008). Thanks to the highly graphic pre- and post-processing features, finite element analysis has also brought researchers and clinicians closer together. It can be argued that without such visualization, stress and strain development would remain mostly academic. The visual interface has improved the communication and collaboration between clinical and research expertise, and is likely to have had a significant impact on the current state of the art in dentistry.

Finite element analysis is not perfect. But we should not expect our theoretical models to be perfect because our understanding of dental properties and processes is still developing. Finite element analysis, however, will continue to improve along with our own understanding about reality. Such continuous improvement will happen as long as we keep comparing reality with theory, and use the insight we gain from these comparisons for improving the theory. The past decades have shown how finite element simulation, which is an expression of our theoretical understanding of biomechanics, has moved from mainly static and linear conditions to more dynamic or transient and nonlinear conditions (Wakabayashi et al., 2008; Srirekha & Bashetty, 2010), thus reflecting the gains that were made in dental science with support from finite element analysis.

5. Acknowledgment

6. References

Akca K, Cehreli MC, Iplikcioglu H. Evaluation of the mechanical characteristics of the implant-abutment complex of a reduced-diameter morse-taper implant. A nonlinear finite element stress analysis. Clin Oral Implants Res. 2003;14(4):444-54.

Albakry M, Guazzato M, Swain MV. Biaxial flexural strength, elastic moduli, and x-ray diffraction characterization of three pressable all-ceramic materials. J Prosthet Dent. 2003;89(4):374-80.

Albrektsson T, Zarb G, Worthington P, Eriksson AR. The long-term efficacy of currently used dental implants: a review and proposed criteria of success. Int J Oral Maxillofac Implants. 1986;1(1):11-25.

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

Atrizadeh F, Kennedy J, Zander H. Ankylosis of teeth following thermal injury. J Periodontal Res. 1971;6(3):159-67.

Ausiello P, Apicella A, Davidson CL, Rengo S. 3D-finite element analyses of cusp movements in a human upper premolar, restored with adhesive resin-based composites. J Biomech. 2001;34(10):1269-77.

Ausiello P, Apicella A, Davidson CL. Effect of adhesive layer properties on stress distribution in composite restorations--a 3D finite element analysis. Dent Mater. 2002;18(4):295-303.

Ausiello P, Franciosa P, Martorelli M, Watts DC. Numerical fatigue 3D-FE modeling of indirect composite-restored posterior teeth. Dent Mater. 2011;27(5):423-30.

Barjau-Escribano A, Sancho-Bru JL, Forner-Navarro L, Rodriguez-Cervantes PJ, Perez-Gonzalez A, Sanchez-Marin FT. Influence of prefabricated post material on restored teeth: fracture strength and stress distribution. Oper Dent. 2006;31(1):47-54.

Bello A, Jarvis RH. A review of esthetic alternatives for the restoration of anterior teeth. J Prosthet Dent. 1997;78(5):437-40.

Boaro LC, Goncalves F, Guimaraes TC, Ferracane JL, Versluis A, Braga RR. Polymerization stress, shrinkage and elastic modulus of current low-shrinkage restorative composites. Dent Mater. 2010;26(12):1144-50.

Budd CS, Weller RN, Kulild JC. A comparison of thermoplasticized injectable gutta-percha obturation techniques. J Endod. 1991;17(6):260-4.

Cattaneo PM, Dalstra M, Melsen B. Strains in periodontal ligament and alveolar bone associated with orthodontic tooth movement analyzed by finite element. Orthod Craniofac Res. 2009;12(2):120-8.

Chabrier F, Lloyd CH, Scrimgeour SN. Measurement at low strain rates of the elastic properties of dental polymeric materials. Dent Mater. 1999;15(1):33-8.

Chung SM, Yap AU, Tsai KT, Yap FL. Elastic modulus of resin-based dental restorative materials: a microindentation approach. J Biomed Mater Res B Appl Biomater. 2005;72(2):246-53

Dejak B, Mlotkowski A. Three dimensional finite element analysis of strength and adhesion of composite resin versus ceramic inlays in molars. J Prosthet Dent. 2008;99(2):131-40.

Deng F, Zhang H, Shao H, He Q, Zhang P. A comparison of clinical outcomes for implants placed in fresh extraction sockets versus healed sites in periodontally compromised patients: a 1-year follow-up report. Int J Oral Maxillofac Implants. 2010;25(5):1036-40.

Drummond JL, King TJ, Bapna MS, Koperski RD. Mechanical property evaluation of pressable restorative ceramics. Dent Mater. 2000;16(3):226-33.

Er O, Yaman SD, Hasan M. Finite element analysis of the effects of thermal obturation in maxillary canine teeth. Oral Surg Oral Med Oral Pathol Oral Radiol Endod. 2007;104(2):277-86.

Ersoz E. Evaluation of stresses caused by dentin pin with finite elements stress analysis method. J Oral Rehabil. 2000;27(9):769-73.

Fagundes TC, Barata TJ, Carvalho CA, Franco EB, van Dijken JW, Navarro MF. Clinical evaluation of two packable posterior composites: a five-year follow-up. J Am Dent Assoc. 2009;140(4):447–54.

Ferrari CH, Ferreria de Mederios JM. Dental trauma and level of information: mouthguard use in different contact sports. Dent Traumatol. 2002;18(3):144-7.

Fokkinga WA, Le Bell AM, Kreulen CM, Lassila LV, Vallittu PK, Creugers NH. Ex vivo fracture resistance of direct resin composite complete crowns with and without posts on maxillary premolars. Int Endod J. 2005;38(4):230 –7.

Gardner FM, Tillman-McCombs KW, Gaston ML, Runyan DA. In vitro failure load of metal-collar margins compared with porcelain facial margins of metal-ceramic crowns. J Prosthet Dent. 1997;78(1):1-4.

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.

Glazer B. Restoration of endodontically treated teeth with carbon fiber posts: prospective study. J Can Dent Assoc. 2000;66(11):613-8.

Goel VK, Khera SC, Ralston JL, Chang KH. Stresses at the dentinoenamel junction of human teeth--a finite element investigation. J Prosthet Dent. 1991;66(4):451-9.

Grossmann Y, Sadan A. The prosthodontic concept of crown-to-root ratio: a review of the literature. J Prosthet Dent. 2005;93(6):559-62.

Hannig C, Westphal C, Becker K, Attin T. Fracture resistance of endodontically treated maxillary premolars restored with CAD/CAM ceramic inlays. J Prosthet Dent. 2005;94(4):342-9.

Himmlova L, Dostalova T, Kacovsky A, Konvickova S. Influence of implant length and diameter on stress distribution: a finite element analysis. J Prosthet Dent. 2004;91(1):20-5.

Huang HM, Ou KL, Wang WN, Chiu WT, Lin CT, Lee SY. Dynamic finite element analysis of the human maxillary incisor under impact loading in various directions. J Endod. 2005;31(10):723-7.

Hubsch PF, Middleton J, Knox J. A finite element analysis of the stress at the restoration-tooth interface, comparing inlays and bulk fillings. Biomaterials. 2000;21(10):1015-9.

Huo B. An inhomogeneous and anisotropic constitutive model of human dentin. J Biomech. 2005;38(3):587-94.

Hur B, Kim HC, Park JK, Versluis A. Characteristics of non-carious cervical lesions--an ex vivo study using micro computed tomography. J Oral Rehabil. 2011. Jun;38(6):469-74. doi: 10.1111/j.1365-2842.2010.02172.x. Epub 2010 Oct 19.

Kawarizadeh A, Bourauel C, Zhang D, Gotz W, Jager A. Correlation of stress and strain profiles and the distribution of osteoclastic cells induced by orthodontic loading in rat. Eur J Oral Sci. 2004;112(2):140-7.

Kinomoto Y, Torii M, Takeshige F, Ebisu S. Comparison of polymerization contraction stresses between self- and light- curing composites. J Dent. 1999;27(5):383–9.

Kinney JH, Balooch M, Marshall GW, Marshall SJ. A micromechanics model of the elastic properties of human dentine. Archives of Oral Biology 2004; 44: 813–822.

Korioth TW, Versluis A. (1997). Modeling the mechanical behavior of the jaws and their related structures by finite element (FE) analysis. Critical Reviews in Oral Biology & Medicine, Vol.8, No.1, (January 1997), pp. 90-104, ISSN 1544-1113

Lanza A, Aversa R, Rengo S, Apicella D, Apicella A. 3D FEA of cemented steel, glass and carbon posts in a maxillary incisor. Dent Mater. 2005;21(8):709-15.

Lin CL, Chang CH, Ko CC. Multifactorial analysis of an MOD restored human premolar using auto-mesh finite element approach. J Oral Rehabil. 2001;28(6):576-85.

Magne P, Perakis N, Belser UC, Krejci I. Stress distribution of inlay-anchored adhesive fixed partial dentures: a finite element analysis of influence of restorative materials and abutment preparation design. J Prosthet Dent. 2002;87(5):516-27.

Magne P. Efficient 3D finite element analysis of dental restorative procedures using micro-CT data. Dent Mater. 2007;23(5):539-48.

Mak M, Qualtrough AJ, Burke FJ. The effect of different ceramic materials on the fracture resistance of dentin-bonded crowns. Quintessence Int. 1997;28(3):197-203.

Marques de Melo R, Galhano G, Barbosa SH, Valandro LF, Pavanelli CA, Bottino MA. Effect of adhesive system type and tooth region on the bond strength to dentin. J Adhes Dent. 2008;10(2):127–33.

Marshall DB, Noma T, Evans AG. A simple method for determining elastic-modulus-to-hardness ratios using Knoop indentation measurements. J Am Ceram Soc 1982;65: C175-6.

Meira JB, Braga RR, de Carvalho AC, Rodrigues FP, Xavier TA, Ballester RY. Influence of local factors on composite shrinkage stress development--a finite element analysis. J Adhes Dent. 2007;9(6):499-503.

Meira JB, Braga RR, Ballester RY, Tanaka CB, Versluis A. Understanding contradictory data in contraction stress tests. J Dent Res. 2011;90(3):365-70.

Michael JA, Townsend GC, Greenwood LF, Kaidonis JA. Abfraction: separating fact from fiction. Aust Dent J. 2009;54(1):2-8.

Misra A, Spencer P, Marangos O, Wang Y, Katz JL. Parametric study of the effect of phase anisotropy on the micromechanical behaviour of dentin-adhesive interfaces. J R Soc Interface. 2005;2(3):145-57.

Natali AN, Pavan PG, Ruggero AL. Analysis of bone-implant interaction phenomena by using a numerical approach. Clin Oral Implants Res. 2006;17(1):67-74.

Natali AN, Pavan PG, Scarpa C. Numerical analysis of tooth mobility: formulation of a non-linear constitutive law for the periodontal ligament. Dent Mater. 2004;20(7):623-9.

Ona M, Wakabayashi N. Influence of alveolar support on stress in periodontal structures. J Dent Res. 2006;85(12):1087-91.

Pereira RA, Araujo PA, Castaneda-Espinosa JC, Mondelli RF. Comparative analysis of the shrinkage stress of composite resins. J Appl Oral Sci. 2008;16(1):30–4.

Pereira FA, Influência do material restaurador, lesão cervical e tipo de contato no comportamento biomecânico de pré-molares superiores tratados endodonticamente. [Faculdade de Odontologia – UFU] Dissertação de Mestrado 2011

Petrie CS, Williams JL. Probabilistic analysis of peri-implant strain predictions as influenced by uncertainties in bone properties and occlusal forces. Clin Oral Implants Res. 2007;18(5):611-9.

Peyton FA, Mahler DB, Hershenov B. Physical properties of dentin. J Dent Res. 1952;31(3):366-70.

Poiate IA, Vasconcellos AB, Mori M, Poiate E, Jr. 2D and 3D finite element analysis of central incisor generated by computerized tomography. Comput Methods Programs Biomed. 2011;

Ramp LC, Jeffcoat RL. Dynamic behavior of implants as a measure of osseointegration. Int J Oral Maxillofac Implants. 2001;16(5):637-45.

Reis BR. Influência da configuração cavitária e tipo de material restaurador no comportamento biomecânico de pré-molar superior. Análise por elementos finitos. [Dissertação]. Uberlândia: Universidade Federal de Uberlândia; 2010. Mestrado em Odontologia.

Romeed SA, Fok SL, Wilson NH. Finite element analysis of fixed partial denture replacement. J Oral Rehabil. 2004;31(12):1208-17.

Roscoe MG, Influência da perda óssea, tipo de retentor e presença de férula no comportamento biomecânico de caninos superiores tratados endodonticamente. [Dissertação de Mestrado], Universidade Federal de Uberlândia, Uberlândia, MG, 2010.

Ross IF. Fracture susceptibility of endodontically treated teeth. J Endod. 1980;6(5):560 –5.

Rundquist BD, Versluis A. How does canal taper affect root stresses? Int Endod J. 2006 Mar;39(3):226-37. PubMed PMID: 16507077.

Salameh Z, Sorrentino R, Ounsi HF, Goracci C, Tashkandi E, Tay FR, Ferrari M. Effect of different all-ceramic crown system on fracture resistance and failure pattern of endodontically treated maxillary premolars restored with and without glass fiber posts. J Endod. 2007;33(7):848-51.

Salameh Z, Sorrentino R, Papacchini F, Ounsi HF, Tashkandi E, Goracci C, Ferrari M. Fracture resistance and failure patterns of endodontically treated mandibular molars restored using resin composite with or without translucent glass fiber posts. J Endod. 2006;32(8):752-5.

Santos-Filho PC. Biomecânica restauradora de dentes tratados endodonticamente – Análise por elementos Finitos. Piracicaba: Faculdade de Odontologia da Universidade Estadual de Campinas; 2008.

Schatz D, Alfter G, Goz G. Fracture resistance of human incisors and premolars: morphological and patho-anatomical factors. Dent Traumatol. 2001;17(4):167–73.

Silva NR, Castro CG, Santos-Filho PC, Silva GR, Campos RE, Soares PV, et al. Influence of different post design and composition on stress distribution in maxillary central incisor: Finite element analysis. Indian J Dent Res. 2009;20(2):153-8.

Silva GR, Efeito do tipo de retentor e da presença de férula na distribuição de tensões em prótese fixas em cerâmica pura – Análise por elementos finitos [Tese de Doutorado]. Piracicaba: Faculdade de Odontologia da Universidade Estadual de Campinas; 2011.

Soares CJ. Influência da configuração do preparo cavitário na distribuição de tensões e resistência à fratura de restaurações indiretas estéticas. Piracicaba: Universidade Estadual de Campinas; 2003. Doutorado em Clínica Odontológica, área de concentração em dentística.

Soares CJ, Martins LR, Fonseca RB, Correr-Sobrinho L, Fernandes-Neto AJ. Influence of cavity preparation design on fracture resistance of posterior Leucite-reinforced ceramic restorations. J Prosthet Dent. 2006;95(6):421–9.

Soares CJ, Santana FR, Silva NR, Preira JC, Pereira CA. Influence of the endodontic treatment on mechanical properties of root dentin. J Endod. 2007;33(5): 603– 6.

Soares CJ, Soares PV, Santos-Filho PC, Armstrong SR. Microtensile specimen attachment and shape--finite element analysis. J Dent Res. 2008 Jan;87(1):89-93.

Soares CJ, Castro CG, Santos Filho PC, Soares PV, Magalhaes D, Martins LR. Two-dimensional FEA of dowels of different compositions and external surface configurations. J Prosthodont. 2009;18(1):36-42.

Soares CJ, Raposo LH, Soares PV, Santos-Filho PC, Menezes MS, Soares PB, et al. Effect of different cements on the biomechanical behavior of teeth restored with cast dowel-and-cores-in vitro and FEA analysis. J Prosthodont. 2010;19(2):130-7.

Soares CJ, Versluis A, Tantbirojn D. Polymerization shrinkage stresses in a premolar restored with different composites and different incremental techniques. Dent Mater., In press.

Soares PV, Santos-Filho PC, Gomide HA, Araujo CA, Martins LR, Soares CJ. Influence of restorative technique on the biomechanical behavior of endodontically treated maxillary premolars. Part II: strain measurement and stress distribution. J Prosthet Dent. 2008;99(2):114-22.

Soares PV, Santos-Filho PC, Martins LR, Soares CJ. Influence of restorative tech- nique on the biomechanical behavior of endodontically treated maxillary premolars. Part I: fracture resistance and fracture mode. J Prosthet Dent. 2008;99(1):30-7.

Soares PV, Santos-Filho PC, Queiroz EC, Araújo TC, Campos RE, Araújo CA, Soares CJ. Fracture resistance and stress distribution in endodontically treated maxillary premolars restored with composite resin. J Prosthodont. 2008;17(2):114-9.

Soares PV. Análise do complexo tensão-deformação e mecanismo de falha de pré-molares superiores com diferentes morfologias radiculares e redução seqüencial de estrutura dental. [Tese de Doutorado]. Piracicaba: Faculdade de Odontologia da Universidade Estadual de Campinas; 2008.

Srirekha A, Bashetty K. Infinite to finite: an overview of finite element analysis. Indian J Dent Res. 2010;21(3):425-32.

Stein PS, Sullivan J, Haubenreich JE, Osborne PB. Composite resin in medicine and dentistry. J Long Term Eff Med Implants. 2005;15(6):641-54.

Sun X, Witzel EA, Bian H, Kang S. 3-D finite element simulation for ultrasonic propagation in tooth. J Dent. 2008;36(7):546-53.

Suwannaroop P, Chaijareenont P, Koottathape N, Takahashi H, Arksornnukit M. In vitro wear resistance, hardness and elastic modulus of artificial denture teeth. Dent Mater J. 2011. 30(4):461-8.

Tantbirojn D, Versluis A, Pintado MR, DeLong R, Douglas WH. Tooth deformation patterns in molars after composite restoration. Dent Mater. 2004; 20: 535-542.

Thomson WM, Slade GD, Beck JD, Elter JR, Spencer AJ, Chalmers JM. Incidence of periodontal attachment loss over 5 years among older South Australians. J Clin Periodontol. 2004;31(2):119-25.

Toms SR, Lemons JE, Bartolucci AA, Eberhardt AW. Nonlinear stress-strain behavior of periodontal ligament under orthodontic loading. Am J Orthod Dentofacial Orthop. 2002;122(2):174-9.

Travassos AB. Protocolo BioCAD para modelagem de estruturas orgânicas. Campinas: Centro de Tecnologia da Informação Renato Archer; 2010.

Turkyilmaz I, Sennerby L, McGlumphy EA, Tozum TF. Biomechanical aspects of primary implant stability: a human cadaver study. Clin Implant Dent Relat Res. 2009;11(2):113-9.

Vargas CM, Arevalo O. How dental care can preserve and improve oral health. Dent Clin North Am. 2009;53(3):399-420.

Vargas CM, Ronzio CR. Relationship between children's dental needs and dental care utilization: United States, 1988-1994. Am J Public Health. 2002;92(11):1816-21.

Versluis A, Douglas WH, Cross M, Sakaguchi RL. Does an incremental filling technique reduce polymerization shrinkage stresses? J Dent Res. 1996;75(3):871-8.

Versluis A, Tantbirojn D, Douglas WH. Distribution of transient properties during polymerization of a light-initiated restorative composite. Dent Mater. 2004;20(6):543-53.

Versluis A, Tantbirojn D, Douglas WH (1997). Why do shear bond tests pull out dentin? Journal of Dental Research 76: 1298-1307.

Versluis A, Tantbirojn D, Douglas WH. Do dental composites always shrink toward the light? J Dent Res. 1998;77(6):1435-45.

Versluis A, Tantbirojn D, Pintado MR, DeLong R, Douglas WH. Residual shrinkage stress distributions in molars after composite restoration. Dent Mater. 2004;20(6):554-64.

Versluis A, Tantbirojn D. Filling cavities or restoring teeth? J Tenn Dent Assoc. 2011;91(2):36-42; quiz 42-3.

Versluis A, Tantbirojn D. Relationship between shrinkage and stress. A, D, editor. Hershey, PA: IGI Global; 2009.

Versluis A, Messer HH, Pintado MR. Changes in compaction stress distributions in roots resulting from canal preparation. Int Endod J. 2006 Dec;39(12):931-9.

Vieira AP, Hancock R, Dumitriu M, Limeback H, Grynpas MD. Fluoride's effect on human dentin ultrasound velocity (elastic modulus) and tubule size. Eur J Oral Sci. 2006;114(1):83-8.

Wakabayashi N, Ona M, Suzuki T, Igarashi Y. Nonlinear finite element analyses: advances and challenges in dental applications. J Dent. 2008;36(7):463-71.

Wang R, Weiner S. Human root dentin: Structural anisotropy and Vickers microhardness isotropy. Connective Tissue Research 1998; 39, 269–279.

Willams G, Lambrechts P, Braem M, Celis JP, Vanherle G. A classification of dental composites according to their morphological and mechanical characteristics. Dent Mater. 1992;8(5):310-9.

Witzel MF, Ballester RY, Meira JB, Lima RG, Braga RR. Composite shrinkage stress as a function of specimen dimensions and compliance of the testing system. Dent Mater. 2007;23(2):204-10.

Yang J, Xiang HJ. A three-dimensional finite element study on the biomechanical behavior of an FGBM dental implant in surrounding bone. J Biomech. 2007;40(11):2377-85.

Yu WJ, Kwon TY, Kyung HM, Kim KH. An evaluation of localized debonding between fibre post and root canal wall by finite element simulation. Int Endod J. 2006;39(12):959-67.
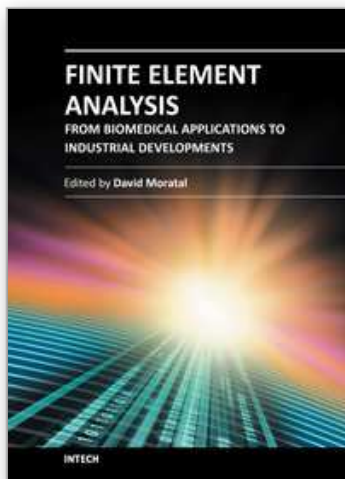
Finite Element Analysis - From Biomedical Applications to Industrial Developments

Edited by Dr. David Moratal

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:

Carlos José Soares, Antheunis Versluis, Andréa Dolores Correia Miranda Valdivia, Aline Arêdes Bicalho, Crisnicaw Veríssimo, Bruno de Castro Ferreira Barreto and Marina Guimarães Roscoe (2012). Finite Element Analysis in Dentistry - Improving the Quality of Oral Health Care, Finite Element Analysis - From Biomedical Applications to Industrial Developments, Dr. David Moratal (Ed.), ISBN: 978-953-51-0474-2, InTech, Available from: http://www.intechopen.com/books/finite-element-analysis-from-biomedical-applications-to-industrial-developments/finite-element-analysis-in-dentistry-improving-the-quality-of-oral-health-care

INTECH
open science | open minds