Mechanical Analysis

By Numerical Simulation

01 Mechanical properties

Mechanical analysis by numerical simulation or the finite elements method is a very valuable tool when evaluating biomechanical responses to different load conditions.

At AVINENT we have performed these tests on our dental implants by evaluating their behavior after placement and their evolution under masticatory loads. These studies have helped us to develop the appropriate design for our dental implants and appliances by achieving minimal dimensional tolerances while taking into account implant-bone stresses for improved osseointegration.

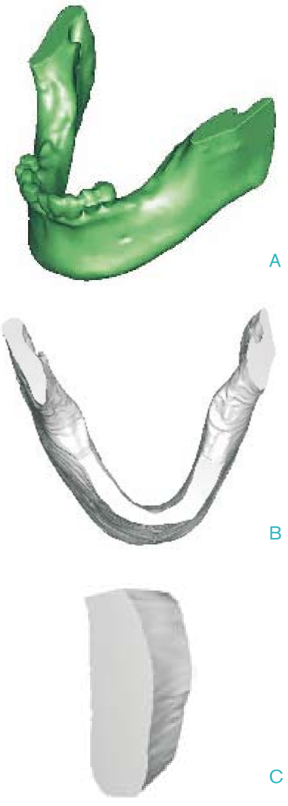

Figure 1. (A) Image of the mandible supposed in this endeavor, obtained with a scanner image treatment program. (B) Geometry of the mandible from the numerical calculation program used in this endeavor. Implant area is indicated therein. (C) Part of the bone that is simulated in this endeavor into which the implant combination is incorporated.

PURPOSE

The basic purpose of this endeavor has been an analysis of the process of placing an implant by numerical methods, and the evolution of its behavior over time under masticatory loads.

A complex three-dimensional model has been defined for representing the fundamental aspects of the system to be analyzed, and the influences of type of load, bone quality and degree of osseointegration have been evaluated in the distribution of stresses and deformations in the bone-implant combination. Osseointegration has been taken into account in a simple fashion by defining some regions of transition between the implant and the cortical and trabecular bones which have been assigned different properties to represent different stages of the process.

GEOMETRIC MODEL

The anatomy of the mandible and quality of the bone vary from patient to patient, so it is clear that an ideal numerical model should be based on each individual’s bone geometry and distribution (thickness of the cortical bone and density of the trabecular bone). The methodology to be followed would be to analyze the mandible to be treated through a scan. The images obtained would be translated into a CAD program, with which a three-dimensional geometric model would be generated representing the actual mandible into which the implant system being studied would be placed.

Document prepared by the ***scientific committee of AVINENT Implant System S.L.*** *[1] with the collaboration of* ***CREB*** *(Centre de Recerca en Enginyeria Biomèdica) [2] and the* ***CTM*** *(Centre Tecnològic) [3]. Both these centers belong to the* ***UPC*** *(Universitat Politècnica de Catalunya).*

[1] ***Scientific committee of AVINENT Implant System S.L.****, A. Cortina, C. Vendrell, E. Falcó, J. Serra*
[2] ***CREB****: A. Mestre*
[3] ***CTM****: J. Caro, Ma D. Riera, J. M. Prado*

However, in this case, analysis of the implant through the different stages (placement, osseointegration and behavior in service) has been performed in a setting that has been supposed to be representative of a healthy adult. The three-dimensional information has been obtained from an actual mandible (see fig. 1 - A), and a region of this mandible has been selected where implantation of the prosthesis has been made (see fig. 1- B and C), in such a way that the complete geometric model is as represented in figure 2.

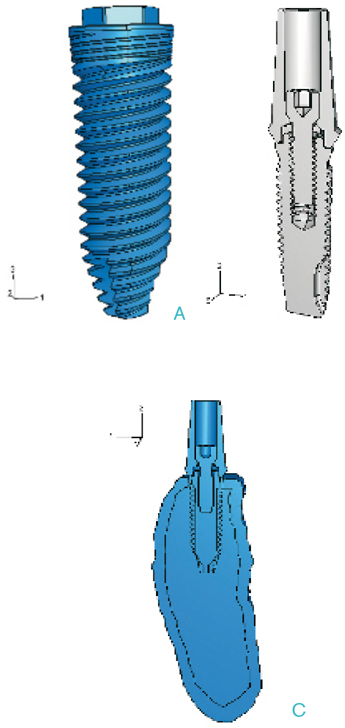

Figure 2. (A) Geometry of the implant; (B) Split view of the metal combination implanted; (C) It also shows the bone, where the cortical region has been differentiated from the trabecular as may be observed in this figure. Another aspect to be emphasized is a simplification of geometry in the area of contact between screw and implant, which is a requirement to resolve such a complex problem from the point of view of geometry.

MATERIALS AND METHODS

The mechanical behavior associated with all the elements of what is called here the implant combination corresponds to the behavior of a conventional isotropic metallic material having linear elasticity, with a Young's modulus equal to 115000 MPa and a Poisson ratio of 0.28 and perfectly plastic behavior, with an elastic limit equal to 380 MPa.

The complex architecture of cortical bone does not allow for simple micromechanical treatment at this time. In this endeavor, its mechanical behavior is simulated through an isotropic linear elasticity model. Concerning plastic behavior, isotropism in yielding and hardening due to deformation have been supposed, with an elastic limit equal to 120 MPa and a maximum strength of 133 MPa.

The mechanical response of trabecular bone depends on its density, which varies with the magnitude of the load applied and the orientation of the trabeculae in the direction of the load. In this case, values ranging between 3000 and 600 MPa have been assigned to the elastic modulus. The Poisson ratio in cellular materials depends completely on their cell geometry and no relationship with relative density is observed. The value customarily assigned to this parameter is 1/3.

The plastic behavior of trabecular bone is also becoming the subject of numerous studies relating to density and strength. However, sufficient data are not available for implementing a specific inelastic behavior model for this material. What has been supposed here is perfectly plastic behavior with an elastic limit from 5 to10 MPa.

Mechanical Analysis

By Numerical Simulation

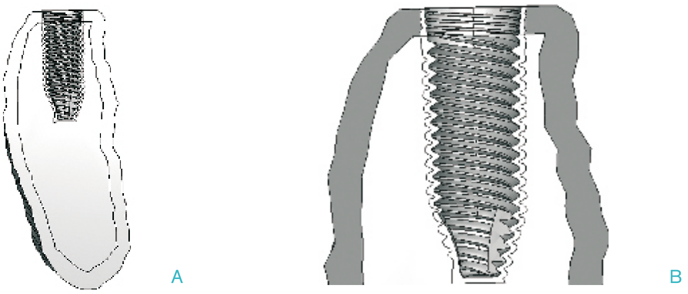

Figure 3. (A) An overall view of the sections assigned to the bone. (B) The different properties considered are detailed; in the basic calculation, which is for the time immediately after implantation, there is direct contact between the implant and the original cortical and trabecular bones; in this case, the transitional areas of the figure have been assigned the properties of these kinds of bone. The status of contacts at different stages of osseointegration have been detailed in the text.

The basic model supposes the osteal status of the mandible when the implantation is performed: cortical layer and trabecular bone having the above defined geometric and mechanical features. Nonetheless, one of the purposes of this simulation has been to study the effect that material produced by osseointegration has on the transmission and distribution of stresses in the implant-bone combination. In this, three different cases have been considered which would correspond to three different stages of the osseointegration process. Figure 3 shows a schematic representation of the assignment of sections with properties that are dependent on the amount of time since implantation.

Three different pairs of Young's modulus values have been assigned to the transitional regions between the implant and the cortical and trabecular bones in the three cases detailed below.

In these three cases, it has been supposed that these transitional areas have not yet developed any structure and therefore any anisotropic behavior.

Table 1. Elastic constants for the three stages of osseointegration supposed.

	YOUNG'S MODULUS (MPa) TRANSITION	
conditions	cortical bone implant	trabecular bone implant
case 1	500	500
case 2	1500	1500
case 3	5000	1500

Figure 4. A rendering the compressive load applied on the implanted combination. α is the angle of this load and it is one of the variables in calculation.

CALCULATIONS

The following calculations have been made:

a. Basic Calculation

Consisting of placing a 100 N load applied perpendicular to the upper surface of the pillar. Implant-bone contact without osseointegration. This refers to the situation after implanting the combination.

b. Effect of Load Inclination

Through three calculations, by varying the angle defined in figure 4, to which the values 10, 20 and 30° (b.1, b.2 and b.3, respectively) have been assigned.

c. Effect of Osseointegration

Three calculations have been made for an α value equal to 30° in cases 1, 2 and 3 (defined in table 1), which include a variation of properties in the area where there is implant-trabecular bone contact and implant-cortical bone contact with the osseointegration process (c.1, c.2 and c.3).

d. Effect of Bone Quality

Comparing the results for two thicknesses of cortical bone: basic thickness, 1.47 mm, and 0.7 mm (d.1, d.2).

Analysis has been considered to be static in all these cases. Calculations have been performed with the calculation program using the ABAQUS 6.5.1 (HKS, Inc., 2005) finite elements method.

Mechanical Analysis

By Numerical Simulation

RESULTS

The results presented herein are for displacement distributions for the combination and the equivalent stress developed in each case. In all these figures, displacements are shown in millimeters (mm) and stresses in mega-Pascals (MPa).

a. Basic Calculation - Displacements

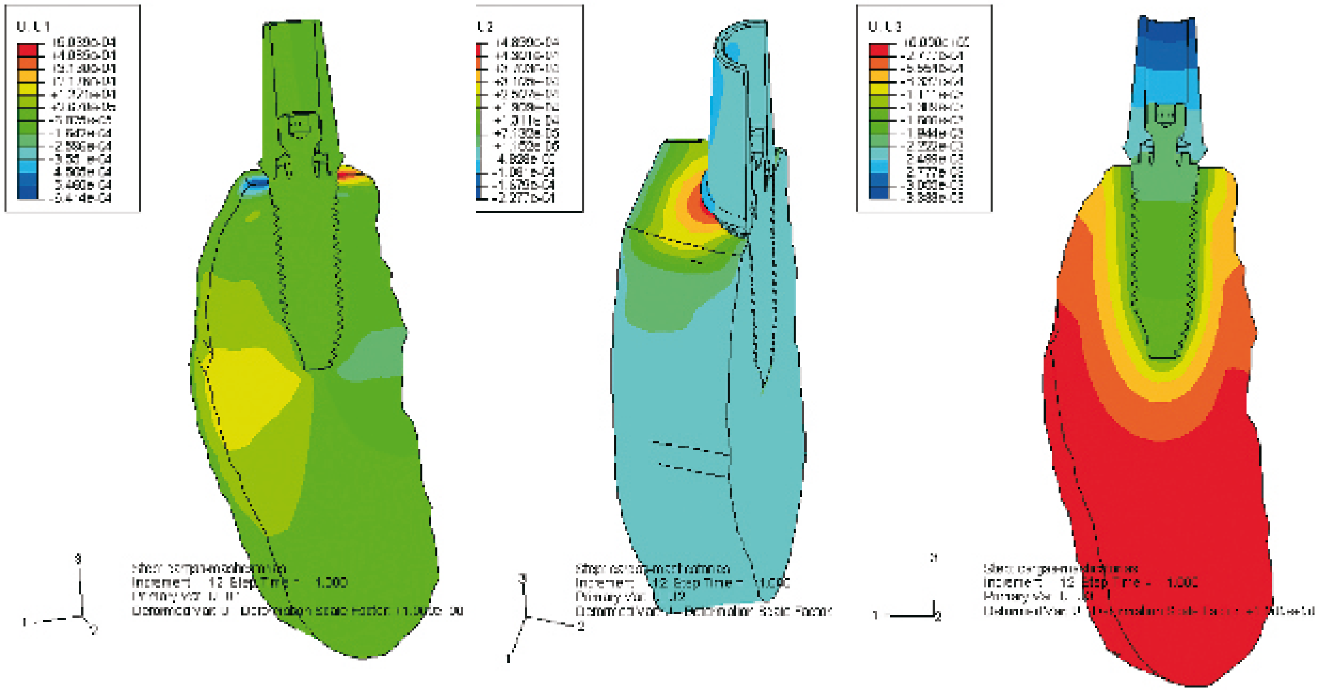

Figure 5a. Distribution of displacements (in millimeters) under load in all three directions for the combination analyzed. Basic calculation. Inclination of load, α = 0°.

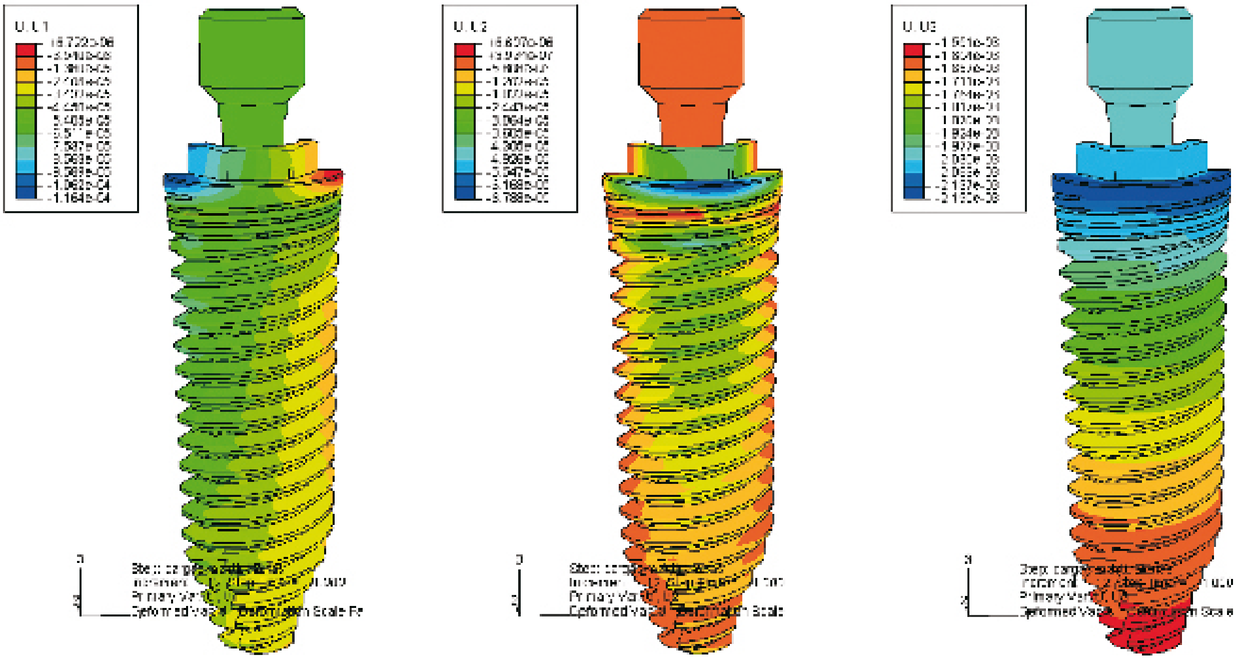

Figure 5b. Distribution of displacements (in millimeters) under load in all three directions in the implant-screw combination. Basic calculation. Inclination of load, $\alpha = 0°$.

Mechanical Analysis
By Numerical Simulation

b. Effect of Load Inclination on Displacement Distribution

Lateral displacement (U1) induced in the bone increases with the angle of inclination of the load.

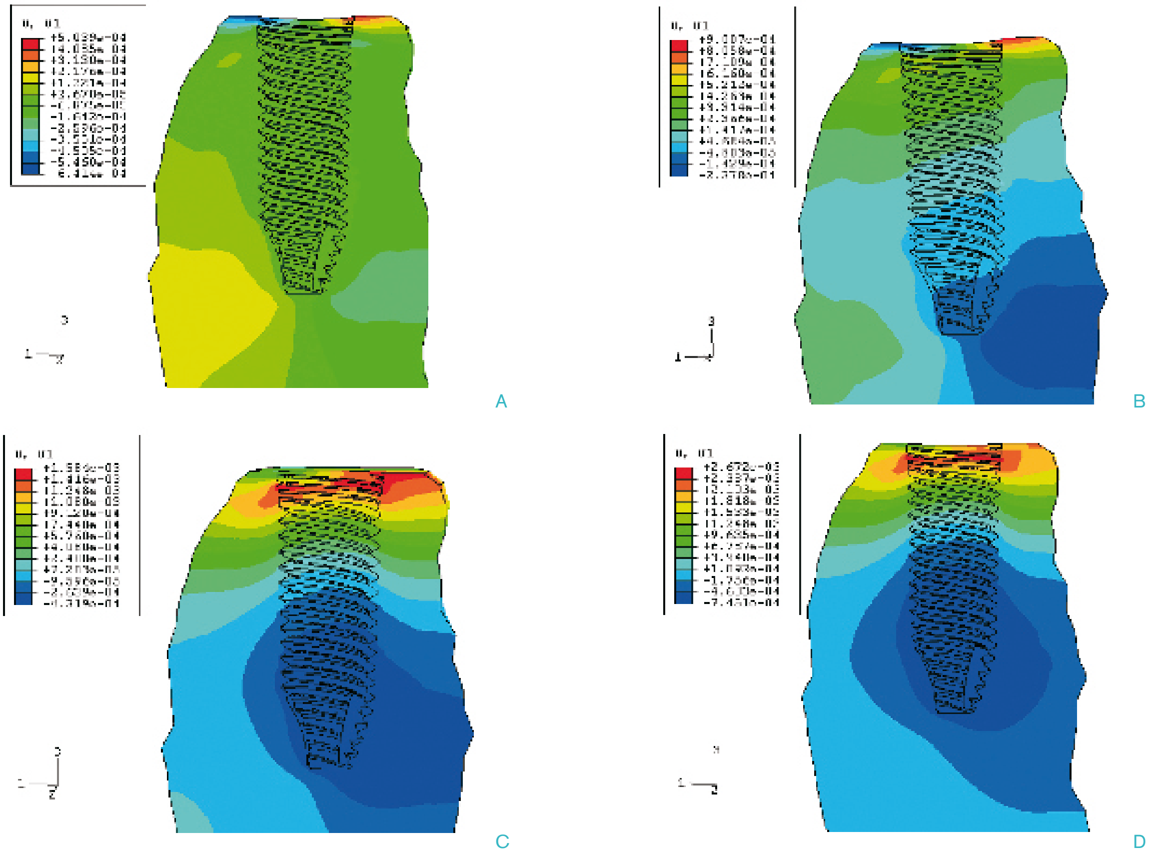

Figure 6. Distribution of displacement U1 in the bone for the different load inclinations studied. (A) 0°; (B) 10°; (C) 20° and (D) 30°.

c. Effect of Osseointegration Level on Displacement Distribution

In figure 7, the results obtained for displacement under load are compared for a 30° load application angle value and the properties defined in table 1 for three levels of osseointegration.

As the rigidity of the osseointegrated area increases, displacement in direction U1 under the same load decreases.

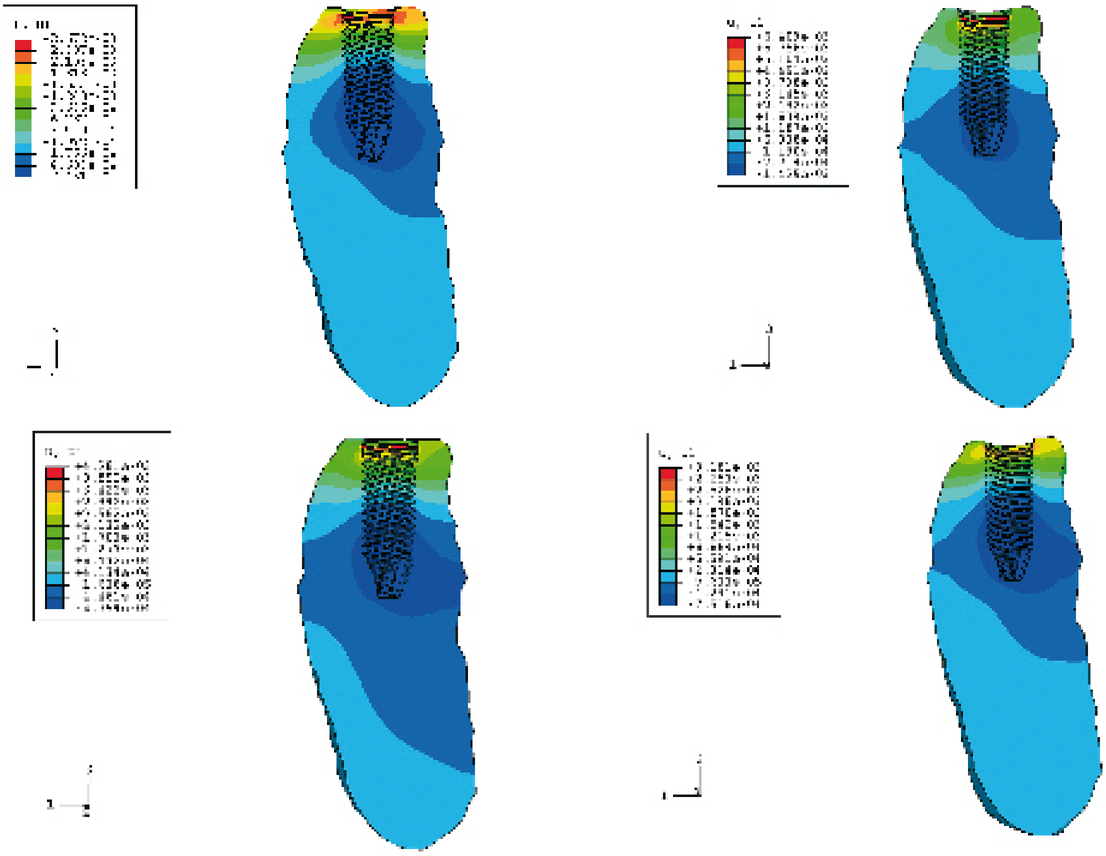

Figure 7. Distribution of displacement in direction 1 in the trabecular bone for: above, from left to right, cases b.3 and c.1-case I (general case with α = 300, and start of osseointegration, respectively) and below, also from left to right, cases 2 and 3 (c.2 and c.3 - intermediate and final stages of the phases of osseointegration considered).

Mechanical Analysis
By Numerical Simulation

d. Effect of Bone Quality on Displacement Distribution

As the thickness of the cortical bone (more rigid than the trabecular bone) decreases, displacement in direction U1 due to the effect of the load being applied increases, as may be seen by observing figure 8 which compares the results for cases b.3 (worst load case, $\alpha = 30°$), with cortical thickness equal to 1.47 mm and case d.2 (cortical bone thickness equal to 0.7 mm).

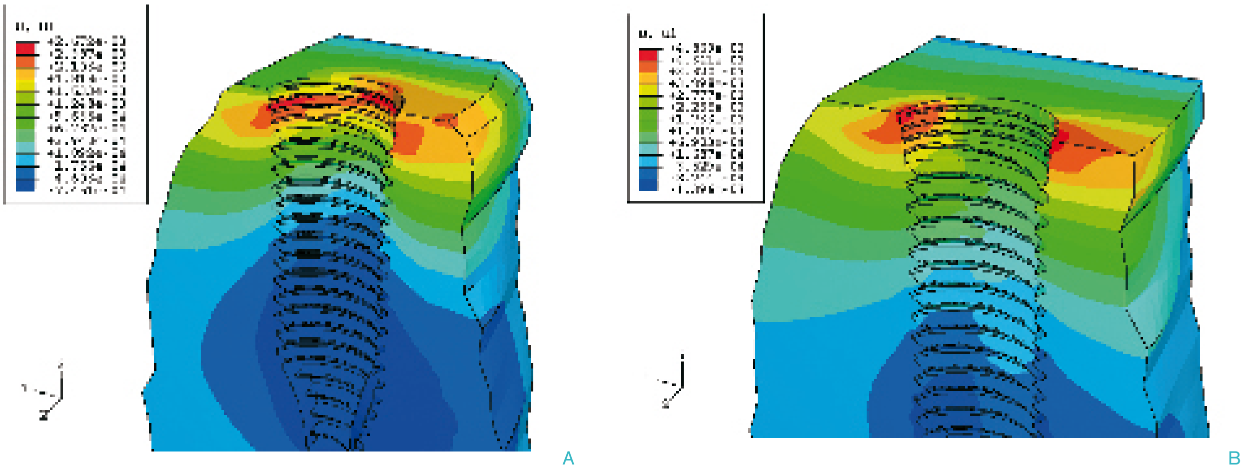

Figure 8. Distribution of displacement U1 in the bone for two different thicknesses of cortical bone. (A) 1.47 mm (case b.3); (B) 0.7 mm. In both cases, the load has been applied at an angle of 30° (case d.2).

a. Basic Calculation – Equivalent Stress

Equivalent stress distribution for the combination and the different elements of the model is shown in figures 9 through 12.

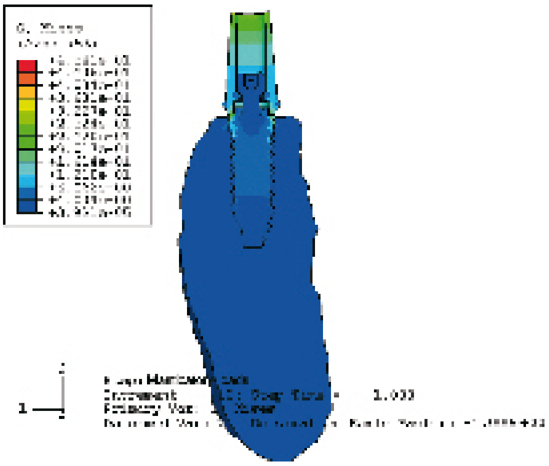

Figure 9. Distribution of von Mises stress (in MPa) under load in the combination analyzed.

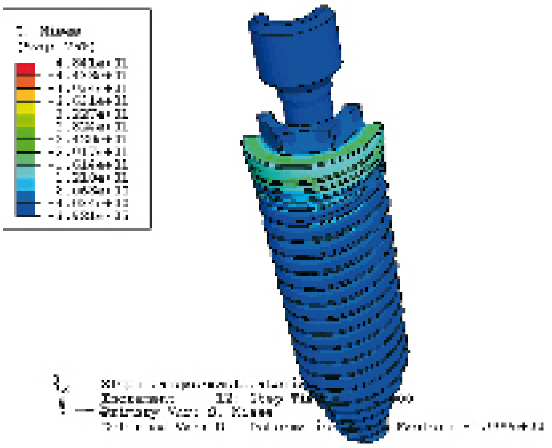

Figure 10. Distribution of von Mises stress (in MPa) under load in the implant.

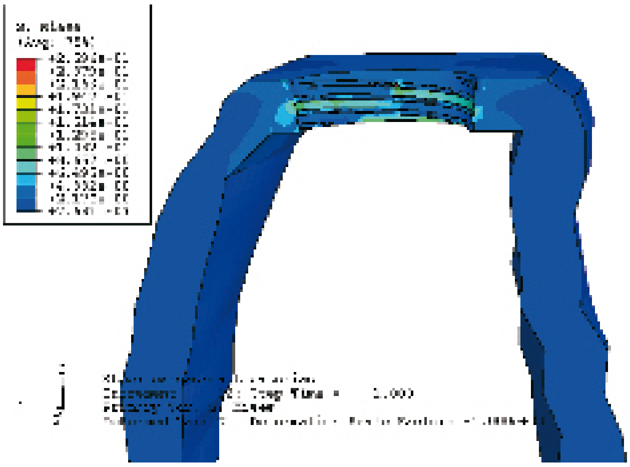

Figure 11. Distribution of von Mises stress (in MPa) in the cortical bone under load.

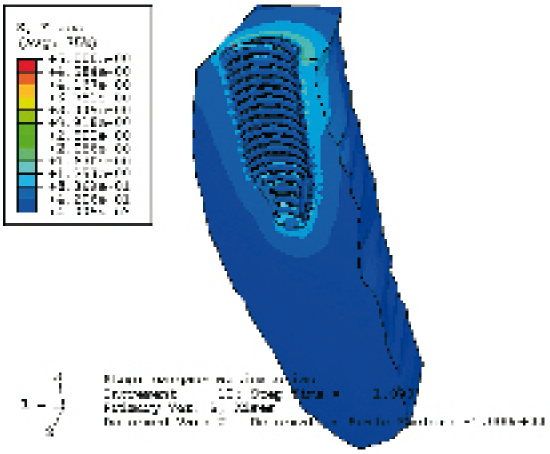

Figure 12. Distribution of von Mises stress (in MPa) in the trabecular bone under load.

Mechanical Analysis

By Numerical Simulation

b. Effect of Load Inclination on Equivalent Stress Distribution

Distributions of equivalent stress (von Mises) in the implant are compared for different inclinations in figure 13. Equivalent stress also increases with the angle at which the masticatory force under consideration is being applied. As may be seen from the results in figure 13, the stresses in the implant can be considerable, with the highest value being found in the upper region at the contact with the cortical bone.

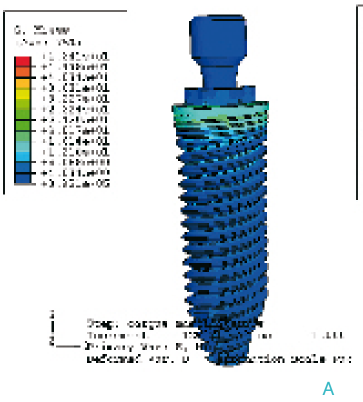

Figure 13a. Equivalent stress distribution in the implant for the different load inclinations studied. In this part of the figure, (A) 0°; (B) 10°.

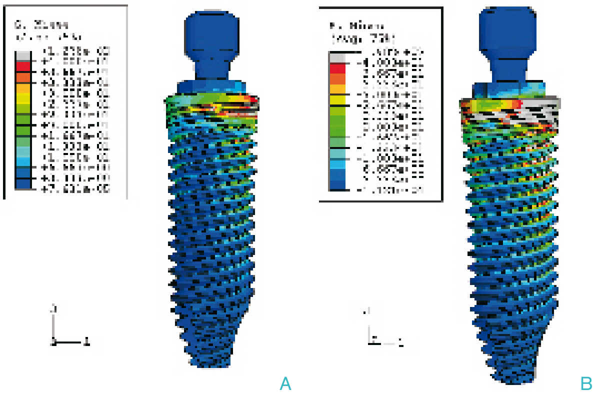

Figure 13b. Equivalent stress distribution in the implant for the different load inclinations studied: (A) 20°, (B) 30°.

c. Effect of Degree of Osseointegration on Stress Distribution

Figure 14 shows a comparison of equivalent stress distributions for the four combinations of materials that have been supposed.

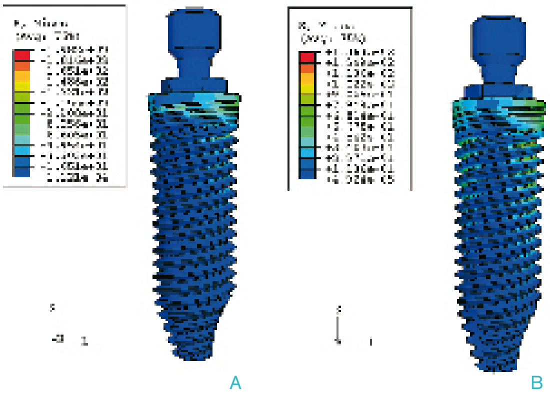

Figure 14a. Equivalent stress distribution for case b.3 (A) and case 1 (a very early stage in the osseointegration process).

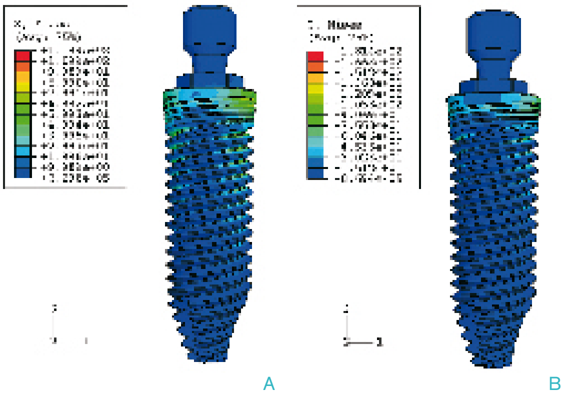

Figure 14b. Equivalent stress distribution for cases 2 (A) and 3 (intermediate and final stages of those considered) in the implant.

Mechanical Analysis
By Numerical Simulation

d. Effect of Bone Quality on Equivalent Stress Distribution

The degrees of stress in the implant for the two bone qualities considered (d.1 and d.2) may be compared in figure 15.

As may be observed in this figure, as the amount of cortical bone decreases, equivalent stress in the upper region of the implant becomes greater.

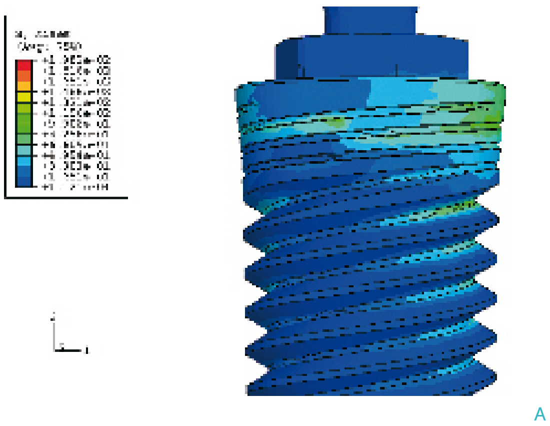

Figure 15a. Equivalent stress distribution in the implant for the two cortical bone thicknesses: 1.47 mm (A) and 0.7 mm.

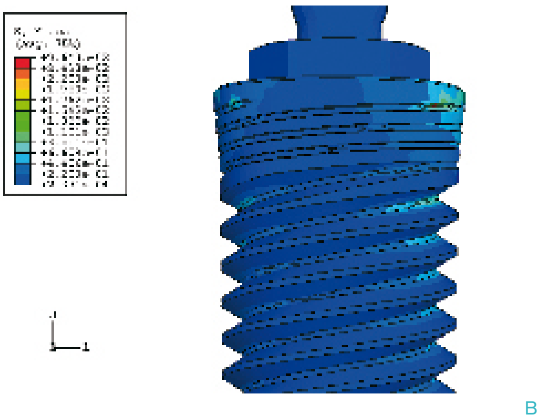

Figure 15b. Equivalent stress distribution for cases 2 (A) and 3 (intermediate and final stages of those considered) in the implant.

CONCLUSIONS

The finite elements method is a very good tool for evaluating the biomechanical response of dental implants under very different load conditions. During a relatively short time an analysis can be made of the effects of bone quality, anatomical morphology and the appropriateness of different kinds of implants for a certain individual.

However, in this case, use of this technique still involves serious limitations that are basically due to the difficulty in comparing the results it provides with actual data, plus a limited knowledge of some of the phenomena governing the implantation and osseointegration processes.

The mechanical properties of the cortical bone, which are quite anisotropic, are variable among the different areas of the mandible itself. Though some information on this has been published, it is difficult to incorporate this variability into the numerical model. It is also known that the density and thus the properties of trabecular bone can vary according to the load applied, and this issue has not yet been included in numerical simulations.

Surgical practice in implantation, with or without mechanical interference, can notably change the stress status of the implant-bone combination, and can thus affect further osseointegration. Different situations have been analyzed and compared in this endeavor.

Perhaps the weakest point in simulating dental implantology in terms of the properties of materials is the difficulty in implementing the phenomenon of osseointegration. In order to take this phenomenon into account here, a transition area between the implant and the trabecular and cortical bones has been defined, which has been assigned properties that vary according to the phase being analyzed. This strategy at least makes it possible to perform an evaluation of the progressive change in the deformation status of the implant-bone system after implantation.

This system is also exacting in addition to being complex. The way this implant adjustment screw is manufactured through industrial processes always requires dimensional tolerances. In this endeavor a comparison has been made between the mechanical status caused by masticatory loads according to two suppositions: a) where the internal threads of the implant and the external threads of the screw mate perfectly, and b) by taking into account the worst case scenario, where the parts are in compliance with dimensional standards, but the threads do not mate perfectly.

As an overall conclusion, numerical simulation techniques can be a great help in designing dental prostheses without losing sight of a series of very important issues that are still difficult to include in analysis. However, as is also true in other engineering disciplines, simulation does at least enable a qualitative knowledge of the distribution of stresses and deformations in the different elements composing the implant area.