Int J Clin Exp Med 2014;7(4):817-825
www.ijcem.com /ISSN:1940-5901/IJCEM1402025

*Original Article*

# FEM evaluation of cemented-retained versus screw-retained dental implant single-tooth crown prosthesis

Marco Cicciu[1], Ennio Bramanti[2], Giada Matacena[2], Eugenio Guglielmino[3], Giacomo Risitano[3]

*[1]Department of Human Pathology, School of Dentistry, University of Messina, ME, IT; [2]Department of Odontostomatology, School of Dentistry, University of Messina, ME, IT; [3]Department of Electronic Engineering, Chemistry and Industrial Engineering, University of Messina, ME, IT*

Received February 7, 2014; Accepted April 14, 2014; Epub April 15, 2014; Published April 30, 2014

**Abstract:** Prosthetic rehabilitation of partial or total edentulous patients is today a challenge for clinicians and dental practitioners. The application of dental implants in order to recover areas of missing teeth is going to be a predictable technique, however some important points about the implant angulation, the stress distribution over the bone tissue and prosthetic components should be well investigated for having final long term clinical results. Two different system of the prosthesis fixation are commonly used. The screw retained crown and the cemented retained one. All of the two restoration techniques give to the clinicians several advantages and some disadvantages. Aim of this work is to evaluate all the mechanical features of each system, through engineering systems of investigations like FEM and Von Mises analyses. The FEM is today a useful tool for the prediction of stress effect upon material and biomaterial under load or strengths. Specifically three different area has been evaluated through this study: the dental crown with the bone interface; the passant screw connection area; the occlusal surface of the two different type of crown. The elastic features of the materials used in the study have been taken from recent literature data. Results revealed an adequate response for both type of prostheses, although cemented retained one showed better results over the occlusal area.

**Keywords:** FEM analysis, bone tissue, load, dental implant

## Introduction

Finite element analysis (FEM) has been adopted in solving complicated geometric problems, for which it is very difficult to achieve an analytical solution. This method is performed to indicate mechanical aspects of biomaterials and human tissues that can be measured in vivo. During the last 30 years, FEM was widely utilizing for the prediction of stress effect on the implant and its surrounding bone. The method in which stresses are transferred to the surrounding bone play a key for long-term success of dental implant. Many factors plays an important role in load transfer from dental implant to bone: loading modality, bone-implant interface, length and diameter of the implants, shape and characteristics of the implant surface, prosthesis type, quantity and quality of the surrounding bone. Human bone tissue and its response to the mechanical forces distribution are difficult to simulate. The complexities of the mechanical characterization of bone and its interaction with implant systems have forced researchers to make major simplifications. Osseo-integration is biological process well described in literature. Firstly Branemark showed predictable options for treatments ranging from the replacement of single tooth to complete arch restorations in titanium implant connected with the bone tissue [1, 2]. The long-term clinical success of the implant therapy is connected to their connection into newly formed bone and to a correct biomechanical placement of those devices in the atrophic jaws. The normal events of bone healing may failure if the implant is loaded before the new bone formation or if the stress distribution follows not balanced strength. In that case, a fibrous scar repair tissue with hard tissue resorption and consequent

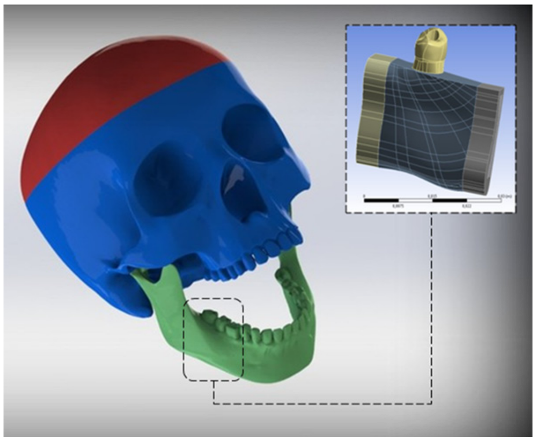


**Figure 1.** 3D CAD model after the reverse engineering process.

clinical failure may happen. Bone resorption process affects mainly the implant neck region and can be activated by surgical trauma, bacterial infection, clustering effect or overloading of the bone-implant interface [2].

Recent studies [3, 4] have shown that technical-mechanical problems are the most common cause of dental implant failure of partial prosthesis supported by implants, which recorded in the 38.7% of cases. Among these, the 32% are late prosthetic complications such as fracture of the prosthetic veneering, screw fracture or loss of cemento retention.

Moreover, the patient does not perceive the occlusal overload on an implant structure with painful symptoms because of the lack of the natural teeth and its mechanical receptors in the periodontal ligament. Nerve fibers in the bone, mucosa, periosteum and muscle trigger a compensatory mechanism defined 'boneperception' [4] that can only partially compensate the mechanical negative feedback of the periodontal ligament. For this reason, every prosthetic implant system is exposed to a greater risk of biomechanical overload. Therefore, the biomechanical implant system and the evaluation of masticatory loads in the maintenance of tissue physiology have a relevant, both epidemiological and clinical, value. This experimental study has the objective to compare the two types of implant prosthetic connection, screwed vs. cemented, from a biomechanical and engineering point of view [1, 5]. Authors analysed the static tension upon the prosthetic-implant system after the application of the same 400 N axial force with a direction from the top to the bottom, using the finite element method which allowed to performance a comparative analysis of the stress distribution in the whole system in relation to the type of prosthetic connection used.

## Material and methods

The relationship between chewing/deglutition is connected to the quality and quantity of the masticatory loads. Direction, intensity and frequency (working cycles) of loads have to be considered for a correct evaluation of the force distribution. The analysis was performed on single tooth dental implant and prosthetic crowns in order to point out possible failure related to the fracture of structural components or to overload on bone tissue (**Figures 1** and **2**).

Tow informatics programs to recreate the virtual three-dimensional CAD model have been used: (1) Siemens NX: it is an integrated product design solution that simplifies and accelerates the development process, providing the highest level of integration between different disciplines. (2) Creo Elements/Pro: a three-dimensional parametric CAD modeler created by Parametric Technology Corporation (PTC). It is a system-oriented approach that uses a mechanical feature-based and offers modeling capabilities of hybrid and solid parts, assembly modeling, and creation of engineering drawings. The analysis process was then divided into two stages: Pre-processing: construction phase of the finite element model; and Post-processing: processing and representation of the solutions.

*Phase pre-processing*

Represents the modelling phase, in which the information passed from the physical system to a mathematical model, extrapolating from the same number of variables and "filtering out" the remaining ones. The physical system, though complex, can be broken down into subsystems. The subsystem will then be divided into finite elements to which is applied a mathematical model. Unlike analytical treatises, it is sufficient that the mathematical model chosen is appropriate to the simple geometry of finite elements.

The choice of an element type in a software program is equivalent to an implicit choice of the mathematical model that is the basis.

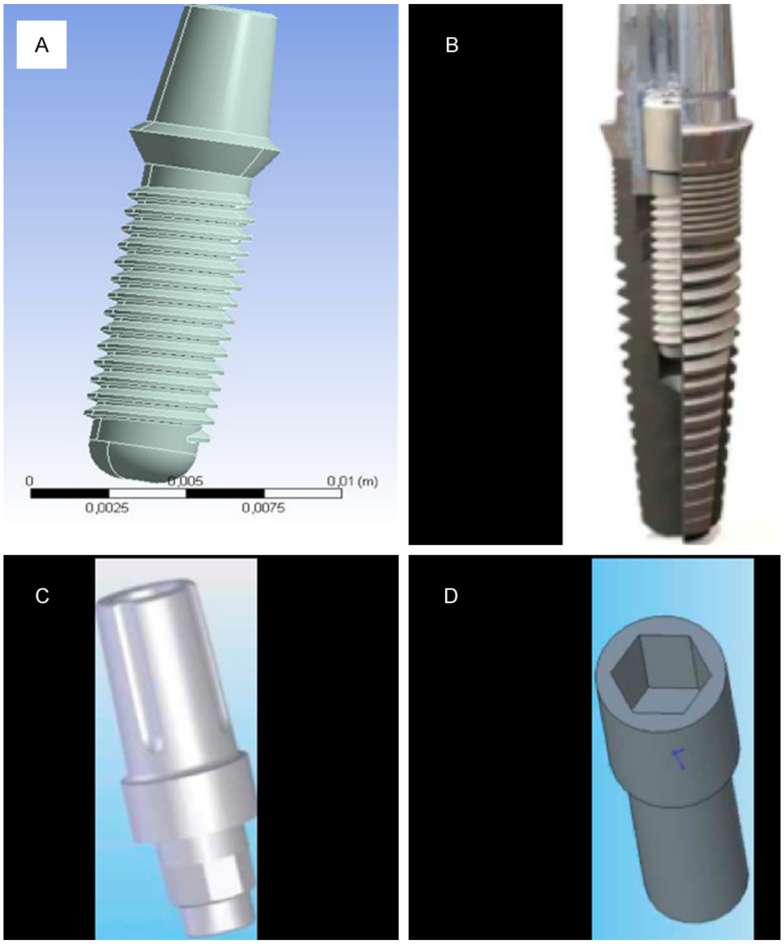



Figure 2. Model of the implant screw and other component. A: Virtual Model; B: Original Model; C: Abutment; D: Passant connection screw.

To get to the finite element model several basic steps should be followed, in order to involve the insertion of errors in the final solution.

*Reverse engineering*

The technique aimed at the creation of the CAD model of the physical object to be achieved, by digitizing the surface, editing and filtering of measured data, segmentation and creation of the corresponding mathematical models.

You can also reconstruct specific parts of an object, which you want to particular analyses.

In our study, in fact, both the dental implant with all its components was recreated. Moreover, the system is isolated with single screw passing through the abutment and the prosthetic crown (**Figures 3** and **4**).

The main dimensions are deducted from the implant-prosthetic components and made real by the small details of their physic-chemical characteristics provided by the scientific literature and catalogues.

The objective is to frame and, therefore, represent a physical system in a very clear and correct shape, so as to build a 3D model that is as realistic as possible.

*CAD import of using FEM*

Then, after obtaining these models three-dimensional CAD, the FEA jaw-implant-prosthesis was performed with ANSYS WORKBENCH 14.5®, program characterized by a bi-directional connectivity CAD, by high productivity and by an innovative design vision that binds the entire simulation process.

The finite element analysis showed the relation (stress and strain) between bone and implant surface cemented prosthetic crown/screwed.

For the simulation of bone were taken from the literature [1-6] some values and benchmarks: Modulus of elasticity: 17.300 MPa; Poisson's ratio: 0.3; Bone density: 1,800 kg/m$^3$; and Compressive strength: 190 MPa.

These values imply, at the start, a considerable approximation (**Tables 1**-**4**), since the characteristics of the bone tissue are so variable and dependent on the individual biotype, which are neither standardized nor classifiable by any investigation FEM.

*Choice of materials*

In this experimental study we chose titanium grade 4 for the construction of the plant, the abutment screw and abutment. For prosthetic crowns the choice fell on feldspathic porcelain.

The properties of materials have been specified in terms of Young's modulus, Poisson's ratio and density. The different physical behav-

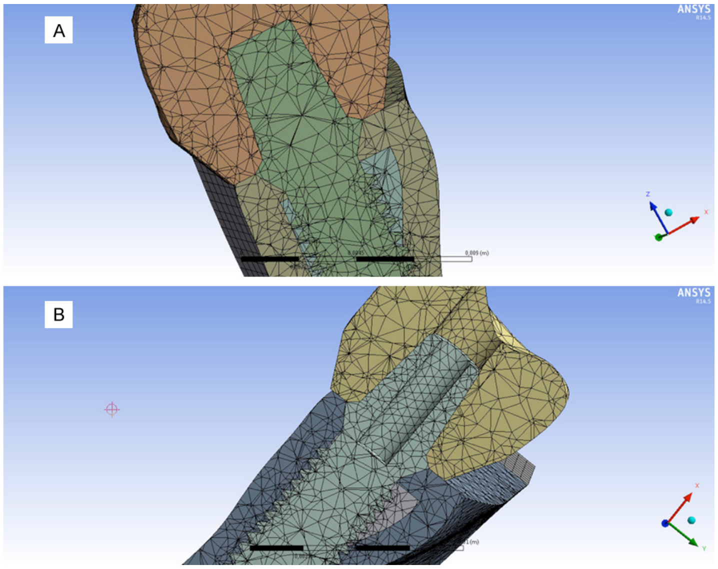



**Figure 3.** Virtual Model analysed in the system representing bone dental implant and prosthetic crown. A: Cemented retained prosthesis; B: Screw retained prosthesis.

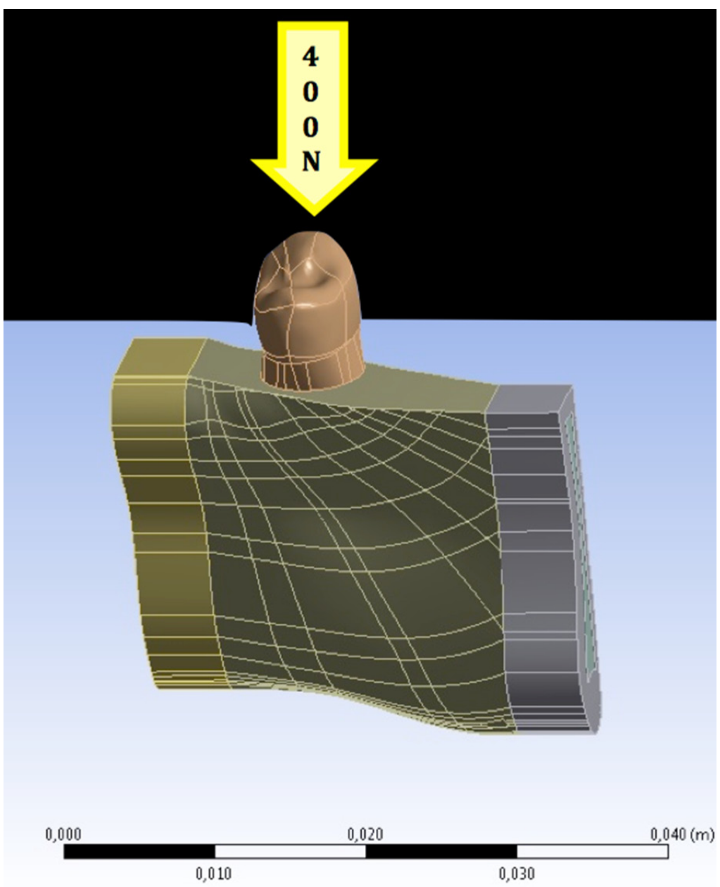



**Figure 4.** Axial load over the 3D model.

iour of materials with respect to the loading forces has been considered.

In our case, the alloys of titanium have a plastic behaviour, thanks to which are resistant. Titanium, in fact, is able to absorb loads, even intense, possibly meeting to a permanent deformation but without tending to fracture.

Titanium alloys have a limit of resistance at least 5 times greater than that of the ceramic, it can be subjected to a voltage of up to 1000 MPa (equivalent to 1000 kg on each $mm^2$ of surface) and do not involve rupture of the crash, or fractures per pulse. For this reason, in our 3D model, the more resistant component within the implant-prosthesis system is precisely the dental implant.

The ceramic instead assumes an elastic behaviour and when subjected to the forces of the load does not deform but it will fracture directly just after passing its threshold limit which is about 200-400 MPa (200-400 then N $mm^2$ of surface. The mere chewing of nougat produces a force load of about 2000 N, and it is immediately evident that the ceramic is a very fragile material.

It was also demonstrated that the mechanical resistance to bending of the ceramic decreases with increasing roughness of its surface, whereby also the manufacture of the material and the internal quality control during processing are of utmost importance to obtain good performance mechanical [1, 7, 8].

*Creating the correct MESH*

The discretization in space and in time has the aim to obtain a discrete model consisting of a finite number of freedom degrees (situation of the mesh). A polygon mesh is a collection of vertices, edges and faces that defines the shape of a polyhedral object in 3D computer graphics and solid modelling (**Figures 3** and **4**).

**Table 1.** Titanium features accordingly with the literature [1-5, 7, 16-18]

| Density [kg·m$^3$] | Young's module [GPa] | Poisson's module [/] | Strength of dilatation [MPa] | Tension of fracture [MPa] |
|---|---|---|---|---|
| 4620 | 96 | 0.36 | 930 | 1070 |

**Table 2.** Ceramic characteristic accordingly with the literature [1-5, 7, 16-18]

| Density [kg·m$^3$] | Young's module [GPa] | Poisson's module [/] |
|---|---|---|
| 2300 | 83 | 0.33 |

**Table 3.** Mechanical features of marrow bone accordingly with the literature [1-5, 7, 16-18]

| Density [kg·m$^3$] | Young's module direction x [MPa] | Young's module direction x [MPa] | Young's module direction x [MPa] | Poisson's module direction x [/] | Poisson's module direction y [/] | Poisson's module direction z [/] |
|---|---|---|---|---|---|---|
| 1200 | 294 | 883 | 294 | 0.236 | 0.345 | 0.236 |

**Table 4.** Mechanical features of cortical bone accordingly with the literature [1-5, 7, 16-18]

| Density [kg·m$^3$] | Young's module direction x [GPa] | Young's module direction x [GPa] | Young's module direction x [GPa] | Poisson's module direction x [/] | Poisson's module direction y [/] | Poisson's module direction z [/] |
|---|---|---|---|---|---|---|
| 1800 | 13.5 | 20.4 | 13.5 | 0.236 | 0.345 | 0.236 |

The basic unit of a mesh is the voxels (volume pixels): in geometric modelling it is divided the volume that contains the object for modelling it in a three-dimensional grid of positions. Each position determines the presence or absence of material. More get closer to the areas of interest; more the mesh increases in number.

In this experimental study, the constraints interlocking quite distant places extreme bones were fixed to affect the results derived from the bone-implant contact.

*Conditions with bone-implant interface*

The bone-implant boundary conditions established in this FEM analysis are those of a perfect interlocking gear. In this study it was chosen to simulate "osseointegration perfect" with a contact type defined bounded that, by definition, is characterized by a total contact surface between the implant and the bone, with no possibility scroll between the two. Although this represents a choice of approximation and simplification, as it does not exist in reality a clinical condition in which there is a 100% contact between the implant surface and bone margin.

*During the post-processing*

In the phase of post-processing, the function-solution is processed and represented, both in fundamental quantities directly represented in discrete form in the FEM method (for example, shifts in structural), both in the quantities derived from them (for example, voltages, in a compatible approach to structural problems).

The post-processing phase is constituted by: (1) Stress analysis: This phase involves the comparison between the high and low values of mechanical stress, and the forces applied on a unit area. The mechanical stresses are formed by a normal component and a tangential component to the surface. The normal component and the tensional force perpendicular to the surface, can rise to a tensile stress if the body, subject to such a force, undergoes elongation, or to a compression stress if the body, on the contrary suffers shortenings (**Figure 5**). The tangential force to the surface, however, rises to a shear stress and/or torsion. In this FEM analysis, the evaluating tensional forces caused by axial loads of 400 N were performed in two dynamic virtual models: one to load the cemented prosthesis implant, the other to load the prosthesis type screwed.

It analyses the evolution of the voltage and the ideal von Mises stresses on critical parts of the implant-prosthesis system modelled. The analysis of the von Mises yield criterion is about tensile and compression related to ductile

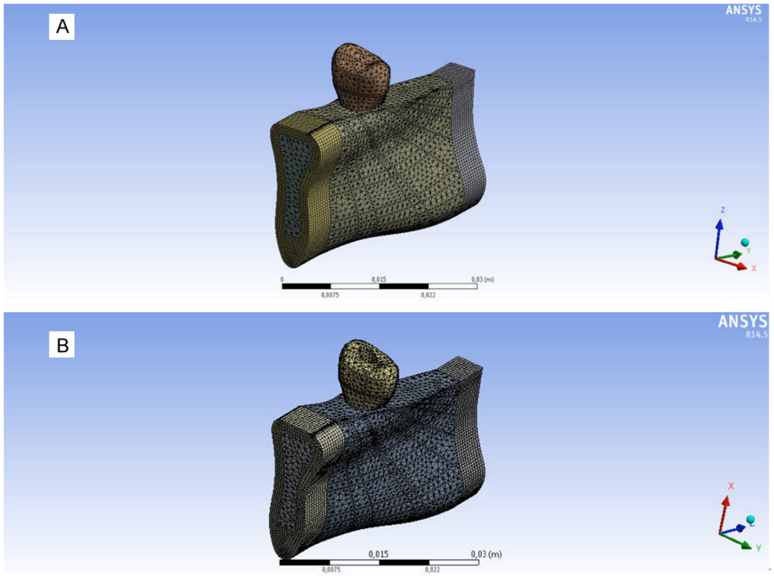



**Figure 5.** Dental implant screw-crown system. A: Cemented retained prosthesis; B: Screw retained prosthesis with the hole in the occlusal area.

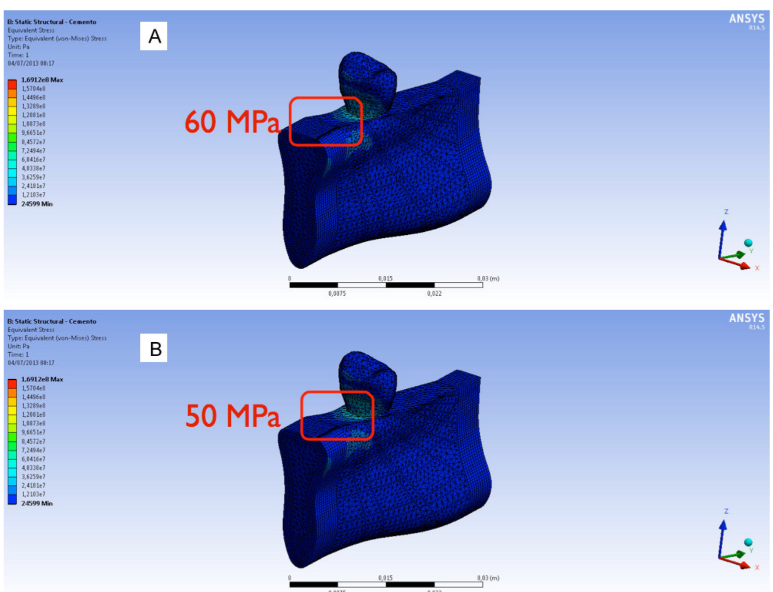



**Figure 6.** Zone 1 margin between crown and bone tissue. A: Cemented retained prosthesis; B: Screw retained prosthesis. MPa Mega Pascal.

materials and isotropic. According to this criterion, the yield strength of the material is reached when the strain energy distorting (deformation of shape but not volume) reaches a limit value. (2) Analysis of deformation: Through simulations elastic-linear finite element analysis allows to evaluate the linear deformations faced by individual components of the virtual model. The deformation of a continuous body is a change in the geometrical configuration of such a body, which leads to a change of its shape or its dimensions following the application of a stress [5, 14]. The study of the deformation of a continuous body expresses the characterization of the mechanical behaviour of the material constituting the body [3, 7, 11, 15]. To this end it is not so important to know the global deformation of the body but to arrive at a local characterization of the deformation, and to a description of the deformation that affects around a generic point of each of the body.

## Results and discussion

In this experimental study a comparative analysis of two dynamic virtual models was carried out: an implant-prosthesis system with single-bonded prosthetic crown on the implant through hardening, compared with an implant-prosthesis with single prosthetic crown screwed on the system (**Figure 6**). The verification of resistance to the same static axial load of 400 N, directed downwards, the two systematic implant-prosthetic it was conducted. That is, it is going to compare, according to the hypothesis of von Mises, the ideal voltage with the voltage sustainable. The finite element analysis shows that the two connections implant-prosthetic, which differ in their physical structure and their constituent parts, are characterized by a different distribution and dissipation of mechanical stress [7-10].

Inside the virtual three-dimensional model, three anatomical areas of the implant-prosthesis system were extrapolated and compared with significant biomechanical results: Zone 1: crown margin-bone; Zone 2: the point of con-

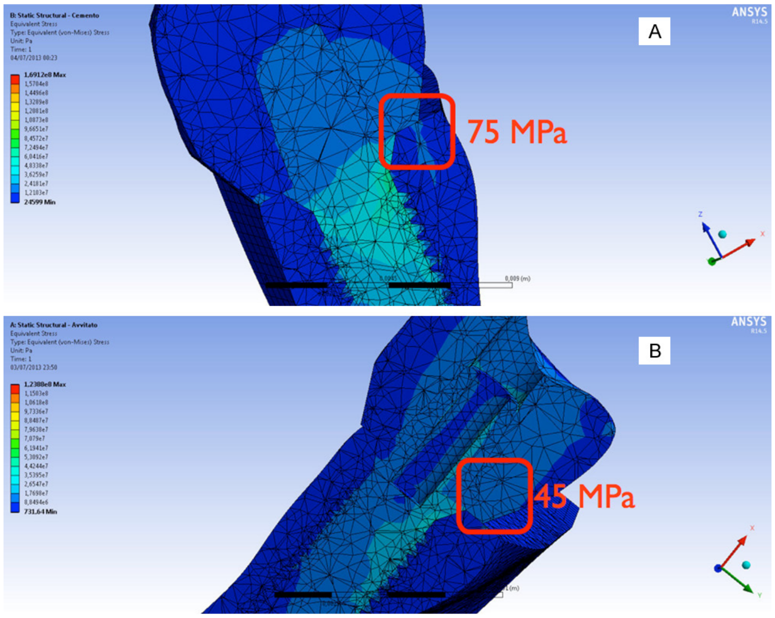



**Figure 7.** Zone 2 margin between bone and dental implant screws. A: Cemented retained prosthesis; B: Screw retained prosthesis. MPa Mega Pascal.

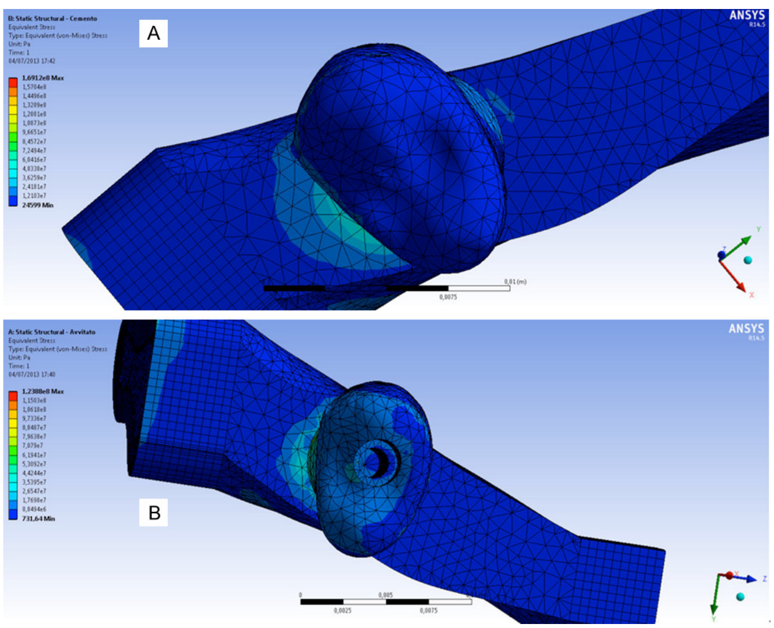



**Figure 8.** Zone 3 margin on the occlusal area. A: Cemented retained prosthesis; B: Screw retained prosthesis. The hole in the screw retained is clearly a wicked point of the structure.

nection of three components: the bone-implant-prosthesis; and Zone 3: occlusal surface.

The load determines a distribution of Von Mises stress fairly uniform with values of 50 MPa to 60 MPa and screwed prosthesis for cemented prosthesis. The higher voltages are recorded at the point of contact between the apical margin of the prosthesis and the bone (**Figures 7** and **8**).

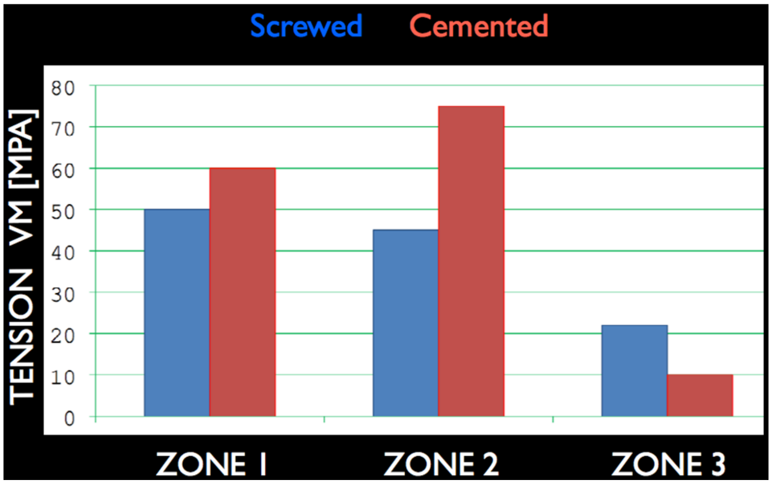
In this case the cemented prosthesis is further subjected to tensional forces. However, the mechanical stress of 70 MPa that is going to dissipate dental implant is absolutely irrelevant because the titanium alloy implant has far greater strength. So if this area were subjected to mechanical stress even more, the implant-prosthesis system would not meet the biomechanical negative effects.

The cemented retained prosthesis is not made up of any area of structural discontinuity. It is integral and coherent throughout its volume. Form the other side, it is clearly evident the access hole through screw connection between the implant and the screwed retained prosthesis (**Table 5**).

From a physical point of view and mechanical, any body discontinuous, interrupted in its structure, appears to be less resistant. The evaluation of the FEM allows highlighting an accumulation of tensional forces in the boundary surrounding the screw hole. These mechanical stresses of 22 MPa discharged only on the screw connection and on the prosthesis tend to fracture easily without passing through an intermediate deformation step because being particularly burdensome for the prosthetic structure, which consists of a ceramic material from fragile elastic behaviour.

In the histogram below there is a summary of the relative maxima values of effort achieved in three zones described by two models, screwed vs. cemented. In the present study it was not

**Table 5.** Tension and Stress distribution over the three different area analysed




performed the analysis of the deformation, since the bodies considered not undergo deformation of the linear type for the following reasons: (1) The ceramic prosthetic only undergoes to deformation in the elastic range, and it returns to its initial shape as soon as the loading force is interrupted; (2) The titanium implant does not meet linear permanent deformation under tensional forces of small intensity.

In this experimental study, numerous approximations were carried out, since an implant-prosthesis system is characterized by an infinite number of variables, which are not objectively reproducible in an investigation FEM. The obtained results cannot be corrected in their absolute value, that is, from a quantitative point of view. In fact, the finite element method applied to dental rehabilitation is intended to give a qualitative indication to the operator in its therapeutic choices. To get the absolute values it should be quantitatively correct the biomechanical conditions of each individual patient, considering all the variables of the individual case. This would be possible only with deep and invasive operations, for example by placing strain gauges in the alveolar bone, connected to force sensors that record all individual biomechanical value (**Table 3**).

This parametric analysis with finite element evaluation of the implant-prosthetic connections cemented and screw retained put in place an engineering and biomechanical comparison between two different types of implant-supported prostheses, concluding that a screw retained prosthesis on an implant seems to be less durable and tends to fracture more than a cemented prosthesis. As well as other types of dental implants, the screw connection seems to be the actual weak link in the chain, resulting in values related with fatigue and consequent failure of the prosthesis [10-18].

This is valid only in qualitative terms, which means that the FEM investigation has shown that a fracture involved firstly the screwed retained prosthesis of the "same" cemented prosthesis that rehabilitate the same dental element in the same area of the same patient buccal.

In fact, thanks to their uniformity of the surface, the cemented retained prostheses offer a better and more homogeneous distribution of the load forces compared to prostheses screwed one.

Ultimately, the relative maximum voltage calculated in the vicinity of the access hole to the screw connection, are to be interpreted qualitatively as an index of the critical point, and not quantitatively as an estimate of the actual voltages present in the area.

This study, however, has the following limitations: (1) Play approximate clinical reality through a virtual simplified mathematical model that cannot reproduce the infinite variability and biotype that characterize a system implant-prosthesis. (2) Further approximation of modelling the bone-implant, considering a type of contact "bounded" that does not reflect reality. (3) Reproduction of an axial force only, without taking into account the horizontal and oblique loading forces. (4) Failure analysis of dynamic fatigue loading on the question. This experimental model is in fact a preliminary study that will be integrated, overcoming the above biomechanical limitations. The approximations value, however, are inevitable in these very complex physical bio systems. We exclude the attempt to validate this FEA model, increasing the type and number of simulations and by modelling a biomechanical system very likely, as reliable as possible and reflecting the clinical reality.

**Address correspondence to:** Marco Cicciù, Department of Human Pathology, Dental School, Messina University, Via Consolare Valeria 98100, Messina, Italy. Tel: 0039-090 221 6911; E-mail: acromarco@yahoo.it; Ennio Bramanti, Department of Odontostomatology, School of Dentistry, University of Messina, ME, IT. E-mail: enniobramanti@hotmail.it

## References

[1] de Cos Juez FJ, Sánchez Lasheras F, García Nieto PJ, Álvarez-Arenal A. Non-linear numerical analysis of a double-threaded titanium alloy dental implant by FEM. Applied Mathematics and Computation 2008; 206: 952-967.
[2] Bozkaya D, Muftu S. Mechanics of the tapered interference fit in dental implants. J Biomech 2003; 36: 1649-1658.
[3] Pietrabissa R, Contro R, Quaglini V, Soncini M, Gionso L, Simion M. Experimental and computational approach for the evaluation of the biomechanical effects of dental bridge misfit. J Biomech 2000; 33: 1498-1495.
[4] Cicciù M, Beretta M, Risitano G, Maiorana C. Cemented-retained vs screw-retained implant restorations: an investigation on 1939 dental implants. Minerva Stomatol 2008 Apr; 57: 167-79.
[5] Michailidis N, Karabinas G, Tsouknidas A, Maliaris G, Tsipas D, Koidis P. A FEM based endosteal implant simulation to determine the effect of peri-implant bone resorption on stress induced implant failure. Biomed Mater Eng 2013; 23: 317-27.
[6] Covani U, Ricci M, Tonelli P, Barone A. An evaluation of new designs in implant-abutment connections: a finite element method assessment. Implant Dent 2013; 22: 263-7.
[7] Roychowdhury A, Pal S. A 3-D FEM analysis of single and multiple screw-root dental implant fixed in a mandible. Crit Rev Biomed Eng 2000; 28: 405-10.
[8] Moon HJ, Lee JH, Choi K, Choi JB, Koh CS. Homogenized stress analysis in a dental implant system. J Med Eng Technol 1997; 21: 233-40.
[9] Sano M, Ikebe K, Yang TC, Maeda Y. Biomechanical rationale for six splinted implants in bilateral canine, premolar, and molar regions in an edentulous maxilla. Implant Dent 2012; 21: 220-4.
[10] Lan TH, Huang HL, Wu JH, Lee HE, Wang CH. Stress analysis of different angulations of implant installation: the finite element method. Kaohsiung J Med Sci 2008; 24: 138-43.
[11] Bergkvist G, Simonsson K, Rydberg K, Johansson F, Dérand T. A finite element analysis of stress distribution in bone tissue surrounding uncoupled or splinted dental implants. Clin Implant Dent Relat Res 2008; 10: 40-6.
[12] van Staden RC, Guan H, Johnson NW, Loo YC, Meredith N. Step-wise analysis of the dental implant insertion process using the finite element technique. Clin Oral Implants Res 2008; 19: 303-13.
[13] Van Staden RC, Guan H, Loo YC. Application of the finite element method in dental implant research. Comput Methods Biomech Biomed Engin 2006; 9: 257-70.
[14] Al-Sukhun J, Lindqvist C, Helenius M. Development of a three-dimensional finite element model of a human mandible containing endosseous dental implants. II. Variables affecting the predictive behavior of a finite element model of a human mandible. J Biomed Mater Res A 2007; 80: 247-56.
[15] Cicciù M, Risitano G, Maiorana C, Franceschini G. Parametric analysis of the strength in the "Toronto" osseous-prosthesis system. Minerva Stomatol 2009; 58: 9-23.
[16] Boschian Pest L, Guidotti S, Pietrabissa R, Gagliani M. Stress distribution in a post-restored tooth using the three-dimensional finite element method. J Oral Rehabil 2006; 33: 690-7.
[17] Kitagawa T, Tanimoto Y, Odaki M, Nemoto K, Aida M. Influence of implant/abutment joint designs on abutment screw loosening in a dental implant system. J Biomed Mater Res B Appl Biomater 2005; 75: 457-63.
[18] Roychowdhury A, Pal S, Saha S. Stress analysis of an artificial temporal mandibular joint. Crit Rev Biomed Eng 2000; 28: 411-20.