 CMES, vol.4, no.3&4, pp.381-396, 2003

Finite Element Analysis of the Jaw-Teeth/Dental Implant System: A Note About Geometrical and Material Modeling

Leone Corradi[1] and Francesco Genna[2]

Abstract: A critical comparison of several Finite Element models is presented, with reference to the analysis of the stress and strain states around a tooth or a fixed dental implant. Such an analysis, if performed on a full, three-dimensional geometry of the jaw-tooth/dental implant system, requires significant computational resources, and it is therefore often done on simplified models, whose validity can be questionable. On the other side, the use of simplified models is adequate — almost mandatory — when detailed results are needed, or when geometrical and material nonlinearities, as well as other complicating factors, are to be taken into account. The first issue addressed here is that of the correct boundary conditions to apply to simplified models in order to obtain results reasonably resembling those given by a full, three-dimensional numerical analysis. It is shown that several simplified models can give acceptable results in terms of stress distribution in the cortical bone, in the proximity of the studied tooth/implant, but that only the study of three-dimensional portions of the total jaw-tooth/implant system can lead to accurate enough results for stresses far from the tooth/implant or, in any case, in the trabecular bone.

A second issue is that of the importance, on the stress state around a loaded tooth or fixed implant, of the presence of the surrounding teeth. It is shown that such a presence has a non-negligible influence, and that, therefore, simplified models describing isolated teeth or implants have only comparative value. Finally, some comments are given about the material properties definition, with special reference to the inclusion of the periodontal ligament into the Finite Element model.

[1] Professor of Engineering
Department of Nuclear Engineering, Politecnico di Milano
Piazza Leonardo da Vinci, 32 — 20133 Milano, Italy
[2] Professor of Engineering (Corresponding Author)
Department of Civil Engineering, University of Brescia
Via Branze, 38 — 25123 Brescia, Italy

1 Introduction

The biomechanics of the human jaw has been analyzed by finite elements since the mid-seventies and, with the increasing computing power available, increasingly accurate geometrical models have been constructed, able to predict at least the global deformation patterns of the jaw under various loading conditions (see, for instance, Hart et al., 1992, Korioth et al., 1994a, 1994b, Meijer et al., 1993a, 1993b, van Zyl et al., 1995, and references quoted therein).

A more difficult problem, not yet fully solved, is the accurate prediction of the stress and strain distribution details in the surroundings of a tooth root, or of a fixed, osteointegrated dental implant. The ability of performing this task is crucial in view of the design of "optimal" fixed implants, i.e., implants that develop in their proximity, under loading, stress and strain fields matching as closely as possible those arising around a healthy tooth. The difficulty posed by this problem is almost unsurmountable, owing to the extreme complication of the geometry, of the loading, of the materials, and of the boundary conditions[3].

The practical relevance of this topic has pushed the research to proceed despite the difficulties. The approach followed in most cases is that of comparative analyses, performed on models based on various degrees of approximation, with the purpose of comparing to each other different implant designs, but with no pretence of furnishing realistic solutions in terms of stress and strain fields. Most of these models are two-dimensional, and based on the assumption of linear elastic behavior of the materials.

Unfortunately, it is easy to observe that several commonly adopted approximate Finite Element models involve a degree of simplification so high, with respect to

[3] Recently developed methods, such as the so-called "meshless" methods (see, for instance, Atluri and Shen, 2002a,b) might alleviate some of these computational difficulties.

reality, to remove any meaning from the obtained results, which, therefore, become rather dubious even in terms of comparison among different design solutions. The most common modeling "errors" concern the geometry, the boundary conditions and the loading (the issue of the material characterization is somewhat subtler, and will be treated here only for a very specific aspect), and arise from a series of objective difficulties, some of which can be itemized as follows:

1. reduction of a three-dimensional problem to a two-dimensional one. Quite a number of papers can be found, in the literature, dealing with the analysis of the stresses around both implants and teeth, performed on two-dimensional Finite Element meshes. The most commonly employed models are (i) plane stress/strain ones, describing cross-sections of the jaw (for instance, Lavernia et al., 1982, Middleton et al., 1996, Rees and Jacobsen, 1997, Vaillancourt et al., 1990); (ii) plane stress ones, describing a frontal view of the jaw (such as one of the models discussed in Meijer et al., 1993b); (iii) axisymmetric ones, which have little resemblance to reality, but avoid the problem of defining the thickness of the various parts of the model, and at least permit a reasonable description of a tooth or an implant subjected to a purely axial loading (examples to be found in del Valle et al., 1998, Rieger et al., 1989); (iv) "mixed" models, like that proposed in Richter et al. (1990), where an axisymmetric implant model immersed into a plane strain cross-section of the jaw is studied;

2. reduction of a full three-dimensional model of the jaw to a smaller, three-dimensional portion of it, such as done in Andersen et al. (1991a, 1991b), McGuinness et al. (1991), Tanne et al. (1987) and several others;

3. definition of the boundary conditions. This problem is particularly tough in the case of the plane/axisymmetric models, but even in the full three-dimensional case the simulation of the real boundary conditions of a human jaw under masticatory loading may require very sophisticated computing tools. In several papers no indication at all of the prescribed boundary conditions is given. In some cases, where use is made of a reduced three-dimensional mesh, the boundary conditions have been reconducted to a set of self-equilibrated loads derived from a full three-dimensional analysis (Andersen et al., 1991a, 1991b). This seems a correct approach, but it still suffers from the difficulty of properly treating the full model;

4. definition of the loading conditions. This problem has two aspects: the first one is related to the difficulty in defining, on three-dimensional models, the intensity, direction and application point of the loads applied by the jaw muscles during the mastication, while the second arises when simplified models are adopted, in which the masticatory actions are simply described in terms of a load directly applied on the tooth/implant. In this case, of very common use, it is quite difficult to assess whether the correct loading condition is due to applied forces or to applied displacements;

5. analysis of an isolated tooth/implant. Most of the analyses, performed either to compare different implant designs, or to analyze some feature of the human teeth or some operations done on them, describe an isolated tooth or implant. The validity of the results of such analyses is questionable, in view of the expected interaction of one tooth/implant with the surrounding teeth.

Plane stress models of the jaw section clearly lose the effect of several important stress components, and plane stress models describing frontal views of the jaw cannot take into account the boxing effect of the cortical bone surrounding the trabecular one; in both cases the definition of elastic moduli and thicknesses is quite arbitrary.

Plane strain models may be adequate for the jaw, but are of course meaningless for the tooth/implant; axisymmetric models suffer from the inverse problem, but they appear to be the most reasonable, among two-dimensional ones, for the prediction of stresses due to purely axial loading in the proximity of the studied tooth/implant. Mixing plane strain and axisymmetric elements is simply wrong.

In all these cases the definition of correct boundary conditions remains an open problem; in many papers the studied portion of the jaw is considered fixed in its bottom part, which may be a poor simulation of the presence of the insertion of the jaw muscles, but is wrong for the largest part of the jaw itself; in all cases, such a boundary condition predicts non-existing high stress concentration close to the interesting zone, around the tooth root or the implant fixture.

In terms of loading, the application of a force rather than of a displacement on the top of a tooth or an implant can make a big difference. As observed by Brunski (1992), the IMZ implant analyzed in Richter et al. (1990) appears to behave quite differently from a standard implant essentially because it has been studied under prescribed displacements; an analysis under prescribed forces, with the same geometry used in Richter et al. (1990), would have evidenced almost no differences between the standard and the IMZ designs, in terms of instantaneous stresses around the fixture.

In this work we try to give some suggestions for the definition of usable simplified models of the jaw-teeth/fixed implants system, in the same spirit of Meijer et al. (1993b). In that work the use of a simplified three-dimensional model of the zone surrounding an isolated implant, in an edentulous jaw, is recommended as the simplest possible model giving results comparable to those given by a complete model of the jaw. Here we address the same issue from a somewhat wider viewpoint, comparing to each other results given by two- and three-dimensional simplified meshes, both for the case of an isolated implant and for the case of an implant surrounded by other teeth. We examine several choices of geometry, boundary conditions and loading types, and try to reach a conclusion in terms of the effectiveness and accuracy of the various models; in the definition of the most simplified ones we try to suggest how to reduce the modeling errors.

Sections 2 and 3 deal with the analysis of the stress state in the jaw bone around an implant: Section 2 essentially deals with the topic of the reduction from a full model to partial two- and three-dimensional ones for an edentulous jaw; Section 3 treats the question of the effect of the presence of adjacent teeth.

As said, also the material characterization creates predictable difficulties. Here we don't want to go into details of this topic, too complex and ramified to be fully treated in this context. We only point out that by far the most important and complex component of the jaw-teeth system, in terms of material description, is the periodontal ligament, which, however, has received so far very little attention, mainly owing to the lack of reliable experimental evidence. A description of the presence of the periodontal ligament in terms of a nonlinear interface, based on some recent experimental results (Pini, 1999; Pini et al., 2000) has been attempted in Gei et al. (2002) and Fogazzi et al. (2000); in Section 4 we will briefly address this issue by switching to the analysis of the stress state around a healthy tooth, with main reference to the illustration of the importance of the presence and correct simulation of the periodontal ligament on the computed results.

We want to remark that in the following we do not intend to illustrate fully correct results from the clinical viewpoint, a task which would require extensive meshing, taking into account details neglected here (such as, for instance, the presence of the cementum around the tooth roots, or elastic anisotropy, and so on), and careful description of several other features outside the scope of the present paper. Our intent is only to give some basic guidelines to the definition of a model representing a good compromise between cost and accuracy; to this purpose, we have defined a basic jaw geometry and simple restraint and loading conditions, and, on their basis, we have constructed several models in order to compare their results, even if the starting data obviously don't fully correspond to a realistic situation from the medical viewpoint.

2 Two- vs. Three-Dimensional Models for an Edentulous Jaw

Here we refer, for the sake of brevity, to a single problem: that of an edentulous jaw in which a single, isolated fixed dental implant, replacing a lower incisor, is subjected to a static, instantaneous vertical load of 250 N. Note that by "edentulous" here we mean, as done usually, a jaw bone with a single alveolus, therefore a much stiffer jaw model than in reality.

It would be interesting to discuss also the effect of different loading conditions, but, for the purposes of this Section, one is felt to be sufficient. Here we wish to compare results given by several different finite element meshes, and to individuate, if possible, the best compromise between accuracy and effectiveness.

The reference results are obtained on the model of Figure 1. This is a full three-dimensional mesh of a jaw, constructed starting from a geometry acquired using an optical system. The basic layout of the optical head is made of a LCD projector that projects suitable patterns of

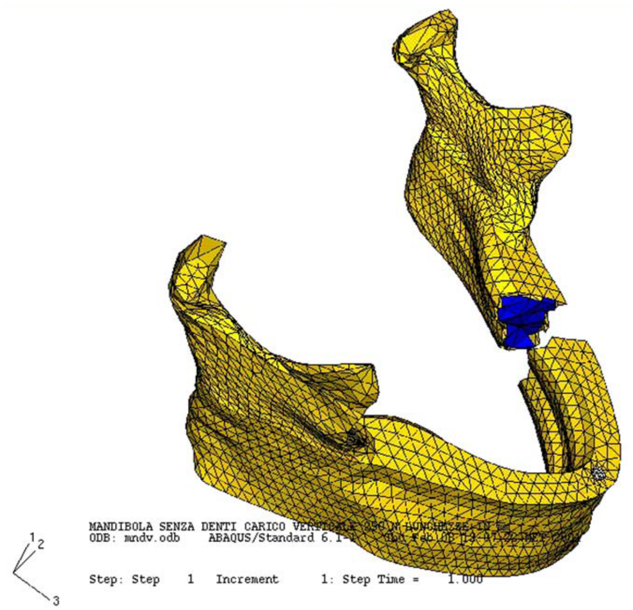


Figure 1 : Three-dimensional mesh of the complete edentulous jaw with a single implant.

Table 1 : Elastic properties of the jaw/implant models

Material	Young's Modulus E [MPa]	Poisson's Ratio ν [-]
Enamel	80,000	0.3
Dentine	18,000	0.25
Trabecular bone	500	0.3
Cortical bone	14,000	0.3
Titanium	103,400	0.35

structured light on the target object; these are acquired, along a different direction with respect to that of projection, by a video camera. The deformation induced on the patterns by the object shape allows suitable light coding: this coding solves in an easy way the problem of determining the correspondence between the directions of projection and of acquisition (Carini et al. (2000)). The mesh was constructed and solved by means of the commercial finite element code ABAQUS (Hibbitt et al., 2000). All the materials are taken as linear elastic and isotropic; since we focus the attention on the issues of geometrical modeling and of definition of boundary conditions, the presence of the periodontal ligament has been neglected. Table 1 summarizes the material parameters adopted, taken from the literature. The Table includes also the data for the teeth, used in the next Sections.

As said, the loading and boundary conditions are kept as simple as possible. The extremities of the condyles are fixed to the reference system, and the loading is given as a resultant force of 250 N acting on the implant in the direction of its axis.

Five simplified finite element discretizations for the study of the same problem are shown in figures 2 to 4.

The model of Figure 2 represents a frontal (mental) portion of the corpus of the jaw. Here two simplifications are introduced: first, the geometry of the bone is obtained from the extrusion of a single section of the jaw (this allows a very easy generation of the mesh); second, much more critical, the boundary conditions are prescribed by fully fixing the lateral sides of the mesh.

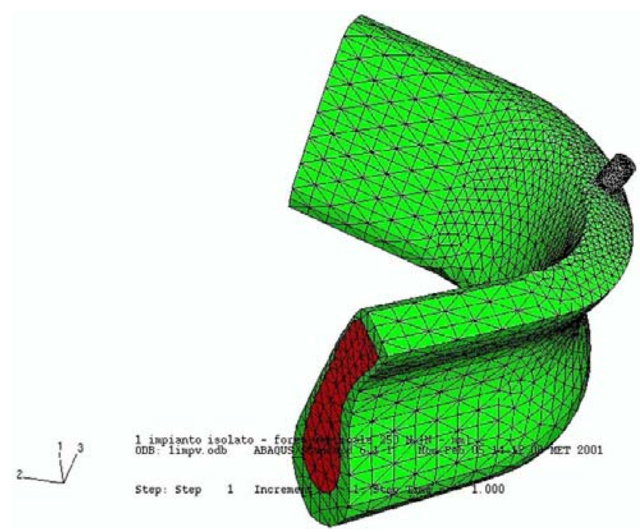
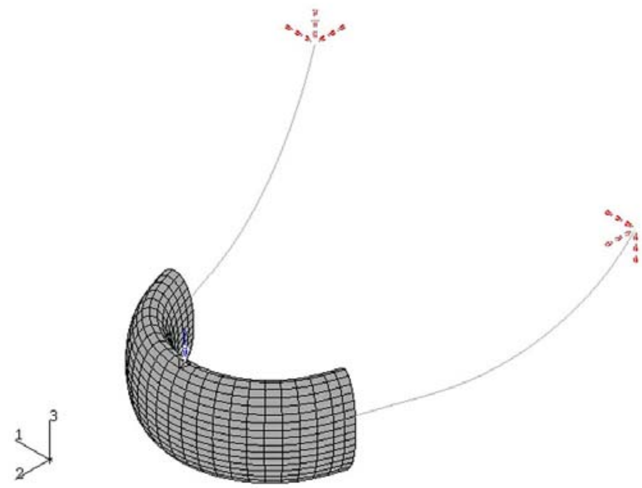


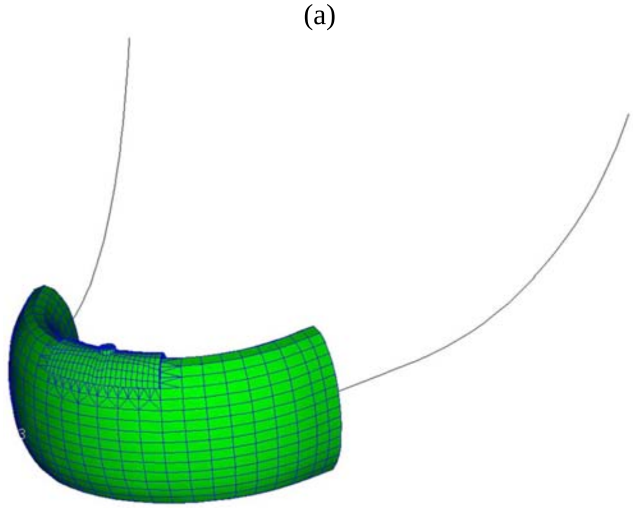
Figure 2 : Three-dimensional mesh of the frontal portion of the edentulous jaw with a single implant.

As for that of Figure 1, the finite element mesh has been obtained using 4-noded tetrahedral solid elements, integrated at a single Gauss point.

The two models of Figure 3 have been constructed somehow following the spirit of what proposed in Andersen et al. (1991a, 1991b) and in Meijer et al. (1993b). They are two greatly simplified three-dimensional models. In the first one (Figure 3a) a frontal (mental) portion of the jaw has been described as a shell of elliptical section, which corresponds to the cortical bone only. The rest of the jaw (the two rami, the coronoid processes and the condyles) has been reduced to two curved beams in space, whose stiffness has been computed in such a way as to approximately match that of the corresponding portion of the jaw (Capsoni et al., 1999). The frontal part of the jaw has been meshed by means of 4-noded shell elements with reduced integration, and the rest of the mesh is composed by 3-noded Timoshenko beam elements. Here, for further meshing simplicity, the implant has been simulated as a rigid inclusion into the shell elements forming the cortical bone.



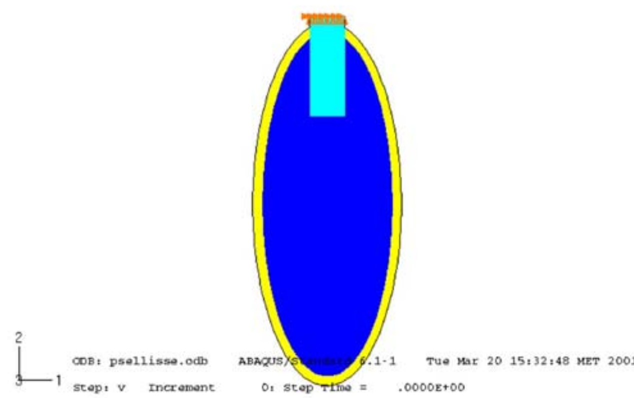
(a)



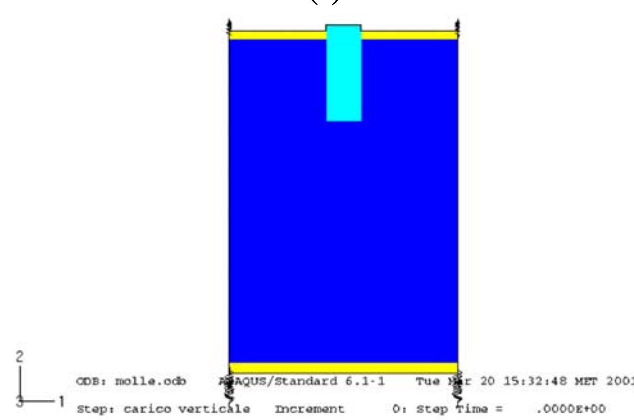
(b)

Figure 3 : Simplified three-dimensional meshes (from Ceruti et al., 1999): (a) shell/beam model; (b) solid/shell/beam model.

The second mesh (Figure 3b), also discussed in Capsoni et al. (1999), represents the integration of a detailed continuum (solid) discretization of the zone surrounding the implant with a relatively coarse shell/beam discretization of the remaining portion of the jaw. The solid zone is defined by of 8-noded brick elements and represents the essential part of the model, the additional shell/beam elements playing the role of an elastic support, providing reasonable boundary conditions. In fact, very similar results are obtained by considering this portion only, with boundary conditions defined as done in Andersen et al. (1991a, 1991b), i.e., by transferring on the reduced geometry the boundary forces stemming from the mesh of Figure 3a. This second procedure, however, requires a preliminary three-dimensional analysis, and is much more demanding computationally. For this reason, here we will comment only the results obtained from the model of Figure 3b.



(a)



(b)

Figure 4 : Two-dimensional meshes: (a) plane stress elliptical cross section of the jaw; (b) plane stress description of a frontal portion of the jaw with spring elements.

Finally, the two plane meshes of Figure 4 are proposed as an extreme simplification, to be used either in the case of lack of computing resources, or in the need of performing a large number of comparative analyses in the nonlinear range. The price to pay in terms of loss of touch with reality is high, but the models of Figure 4, while greatly reducing the computational burden, avoid some of the problems connected with the use of plane geometries, summarized in the Introduction.

The plane stress section of Figure 4a is quite similar to many plane meshes reported in the literature. The main difference is given by the loading and boundary conditions. In order to avoid the necessity of fixing the lower part of the section, as done usually, here we have fixed in the direction of its axis the top side of the implant, and have simulated the loading actions by means of body forces distributed inside the elements, in such a way as to roughly represent the shear actions transmitted by one cross section of the jaw to another. The difficulty here is to make a correct guess about the shear stress distribution over a complex geometry in the presence of variable stiffness.

The plane stress mesh of Figure 4b is in principle identical with that proposed in Meijer et al. (1993b), and suffers from the main problem of defining thicknesses and material parameters in such a way as to somehow take into account the existence of cortical bone around the trabecular one in the inner portion of the jaw (this could be done, in a very approximate way, either defining an equivalent thickness or using homogenization techniques for the elastic moduli). However, here the boundary conditions are more accurate than in Meijer et al. (1993b): a suitable set of linear springs is placed at the sides of the considered portion of the jaw, whose stiffness is computed in such a way as to approximately match that of the excluded portions of the jaw itself.

The subsequent Figures 5 to 8 show the results of the analyses performed on the described models, in terms of the von Mises equivalent stress (a scalar measure of the stress intensity, defined as $\sigma_M = \sqrt{\frac{3}{2}s_{ij}s_{ij}}$, where s_{ij} is the stress deviator). Recall that only the meshes of Figures 1, 2 and 4 describe the presence of the trabecular bone; therefore, we focus our attention, for comparative purposes, only on the stresses in the cortical bone around the implant. We will add later a brief comment about the importance of the simulation of the trabecular bone.

The main difference between the reference results, depicted in Figure 5, and all the three-dimensional others is in the stress diffusion in the jaw, far from the implant. This was predictable, and was not the main objective of the analysis. On the contrary, the stress peaks around the

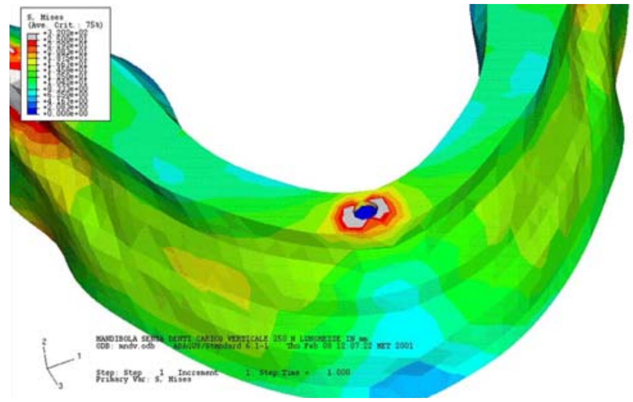


Figure 5 : Three-dimensional mesh of the complete edentulous jaw with a single implant: von Mises stress contours around the implant.

implant are of the same order of magnitude for all the analyses, except for the mesh of Figure 3a. The best approximation of the reference result is that given by the model of Figure 3b, illustrated in Figure 7b; here, both the peak value and the distribution around the fixture hole are quite close to the reference ones.

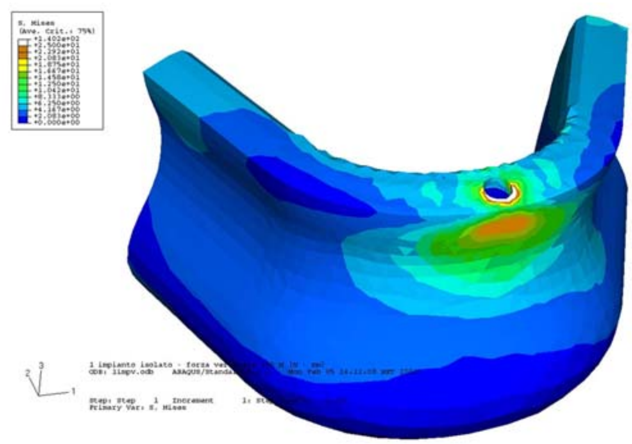
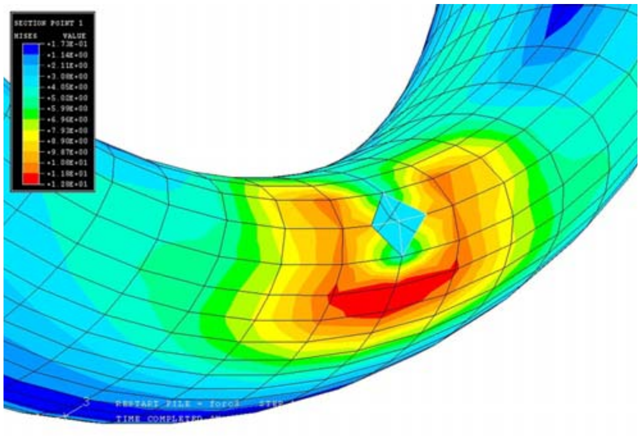
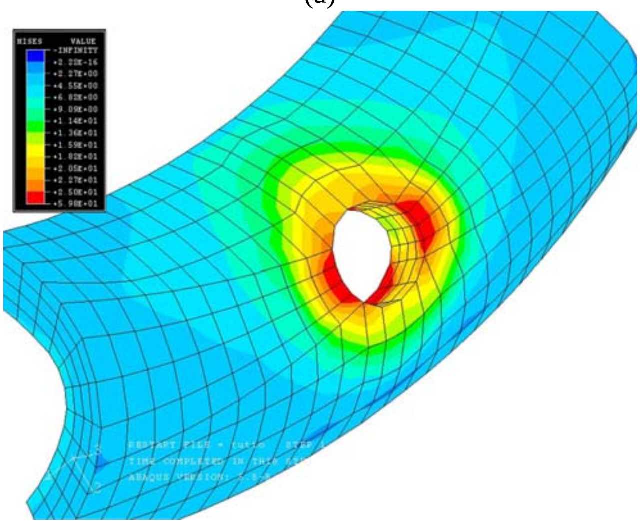


Figure 6 : Partial three-dimensional mesh: von Mises stress contours.

The other two three-dimensional models appear less accurate. The shell-element mesh of Figure 3a (results of Figure 7a) underestimates significantly the peak values, and the solid-element mesh of Figure 2 (results in Figure 6) clearly suffers the vicinity of incorrect boundary conditions. This last mesh would almost certainly have furnished accurate results if, for instance, it had been coupled with beam elements simulating the omitted portion of the jaw, such as done for the meshes of Figure 3.

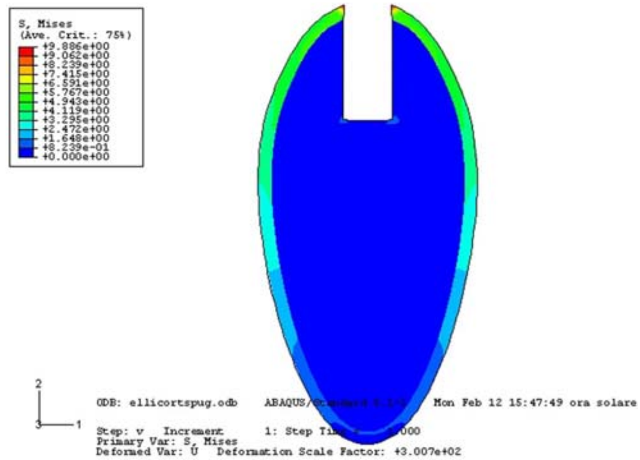


(a)

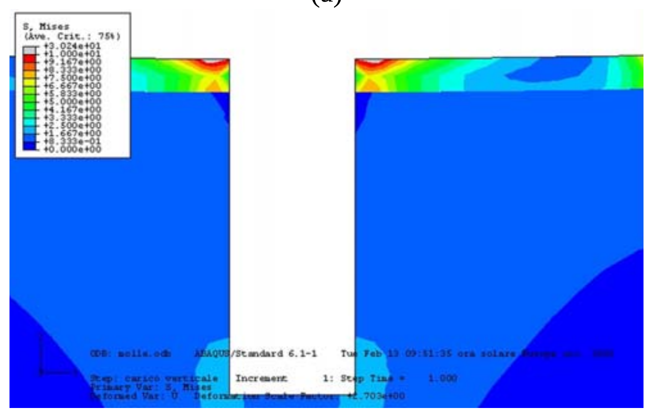


(b)

Figure 7 : Simplified three-dimensional meshes: (a) von Mises stress contours for the shell/beam model of Figure 3a; (b) von Mises stress contours for the solid/shell/beam model of Figure 3b.



(a)



(b)

Figure 8 : Two-dimensional meshes: (a) von Mises stress contours for the elliptical cross section model of Figure 4a; (b) von Mises stress contours for the frontal mesh with spring elements of Figure 4b.

The results of the plane models (Figure 8) are clearly inadequate but interesting. For the plane stress cross-section of the jaw it must be said that some numerical experiments have shown a substantial insensitivity of the stress in the cortical bone to the actual load definition (we have tried applying body forces in the cortical bone only, in the trabecular bone only and in the whole mesh), whereas some differences arise in the trabecular bone. In any event the peak stress values are here badly underestimated. The model of Figure 4b seems better suited to catch the peak stress values in the spongious bone, giving results reasonably close to the three-dimensional ones, even if it still underestimates the peak stresses in the cortical bone and can obviously predict stress diffusion in one direction only.

To conclude, it seems that the best compromise between accuracy and cost is provided by the model of Figure 3b. This permits the assessment of the following points: (i) around the implant, a mesh of solid, continuum elements is mandatory; (ii) an accurate simulation of the geometric details is not equally important; (iii) when considering only a limited portion of the jaw around the implant, the definition of correct boundary conditions is crucial. The conclusion above are in agreement with those reached by Meijer et al. (1993b).

A comment must be added about the importance of including the trabecular bone into the model. This bone really has almost no stiffness, if compared to the cortical one, and the extremely good match between the results of Figures 5 and 7b seems to confirm that its description is not essential for a fair estimate of the stresses *in the cortical bone in the proximity of the implant*. Nevertheless, a simulation in which the trabecular bone part was removed from the full model of Figure 1 has shown quite different results from those of Figure 5, with higher peak stresses and a much larger zone of stress diffusion. Obviously the trabecular bone acts as a sort of soft "elastic" bed, supporting in an effective way the cortical one. Thus, it appears that a good model, even if approximate, should take into account also the trabecular bone. This conclusion becomes trivial if one is interested in details of the stress in the trabecular bone itself, which, then, should be better treated in a discrete way, such as done, for instance, by Lavernia et al. (1982) or Patra et al. (1999).

Finally, it must be pointed out that the peak stress values, of about 45 *MPa*, in the cortical bone close to the implant should be taken as indicative values only, owing to the rather crude boundary conditions prescribed on the reference mesh of Figure 1, and to the absence of the adjacent teeth in these simulations. The next section addresses this last issue.

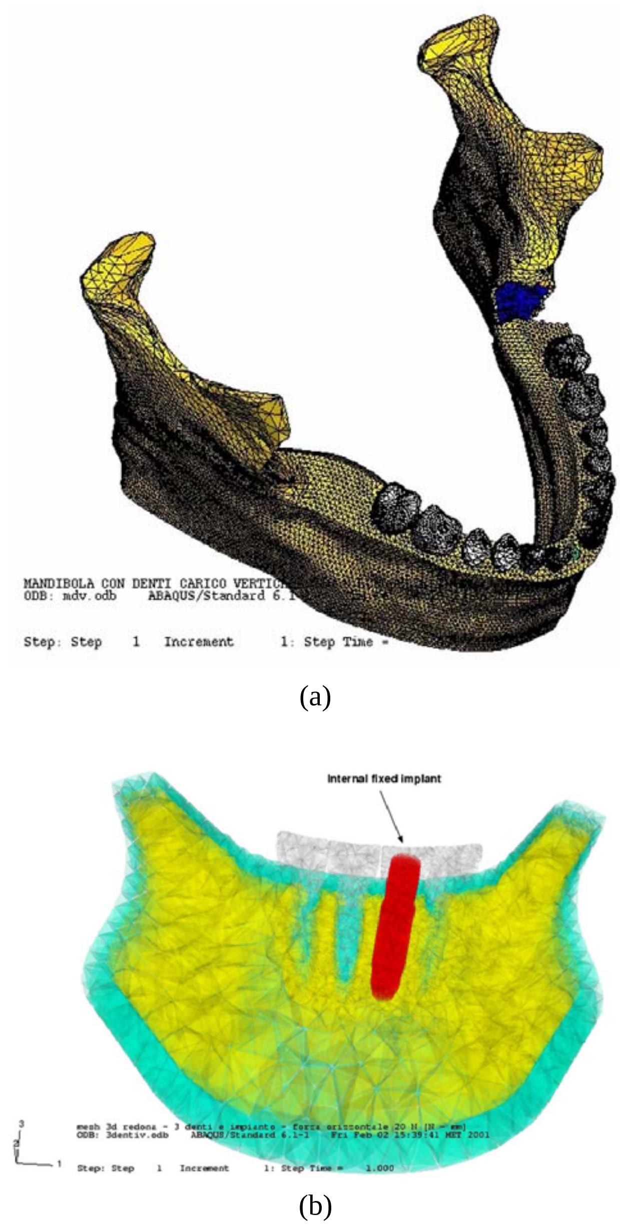


Figure 9 : Three-dimensional meshes of the jaw with teeth and a single implant: (a) mesh of the complete jaw; (b) mesh of the frontal portion of the jaw with implant plus three teeth only.

3 Importance of the Presence of Adjacent teeth

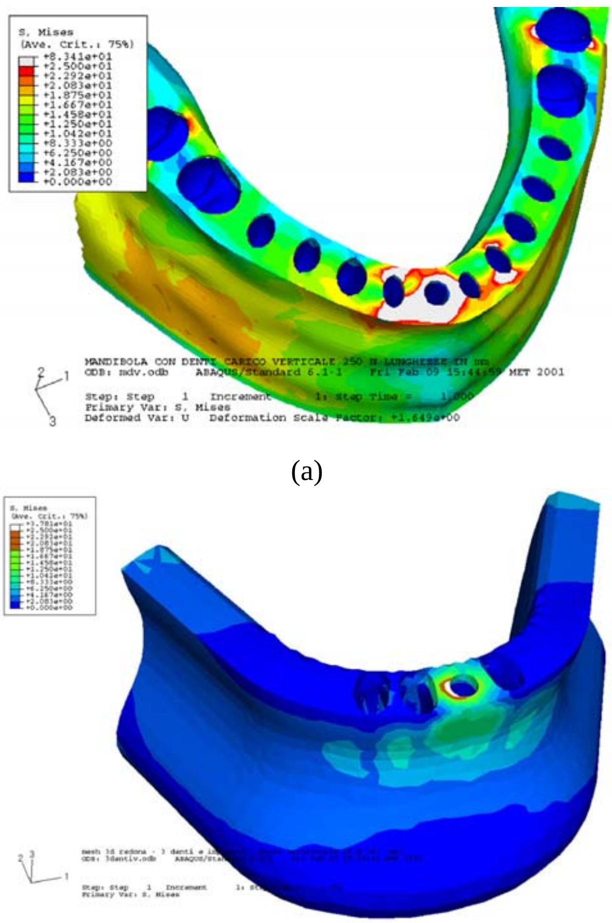
Here we consider a question apparently not much studied in the literature, i.e., the effect of the interaction between an implant and the surrounding teeth in terms of stresses and strains around the fixture of the implant itself. To this purpose, we make reference to the meshes of the jaw of Figures 1 and 2 only, in which either a full set of teeth is added, such as shown in Figure 9a, or only three teeth adjacent to the implant are added, as shown in Figure 9b.

Now we analyze the results obtained for two different loading conditions, under the two different modeling situations, for each of the two meshes of Figures 9. The examined combinations are:

1. vertical load of 250 N, implant with no surrounding teeth;
2. vertical load of 250 N, implant with surrounding teeth;
3. horizontal load of 20 N, implant with no surrounding teeth;
4. horizontal load of 20 N, implant with surrounding teeth.

The horizontal load acts in the sagittal plane of the jaw, in a direction orthogonal to the implant axis, towards the exterior of the jaw. The comparison between the results given by the mesh of Figure 9a and that of Figure 9b allows one to understand if, in order to correctly catch the effect of the interaction between implant and surrounding teeth, it is necessary to model the full jaw or it is sufficient to describe only the teeth adjacent to the implant, such as done in the mesh of Figure 9b.

The results are collected in Figures from 10 to 12. The effect of the simulation of the influence of adjacent teeth is seen by comparing Figure 10a with 5, 10b with 6 (vertical loading), 11a with 12a and 11b with 12b (horizontal loading) for the two different meshes. The comparison of Figures 10a with 10b and 11a with 11b allows to understand the importance of the description of the teeth far from the implant.



(a)

(b)

Figure 10 : Three-dimensional meshes of the jaw with teeth and implant, von Mises stress contours, vertical loading condition: (a) full mesh; (b) mesh of the frontal portion.

Let's start with comments on the vertical loading case. A comparison between Figures 5 and 10a clearly indicates that an analysis done considering a single tooth is in substantial error. In the presence of a full set of teeth, the jaw is much weaker than if considered as a solid entity with a single implant inserted into it. The zone interested by the peak stresses is much wider in Figure 10a, engulfing a good portion of the two adjacent teeth on each side of the implant. The peak von Mises stress is almost twice, in the presence of the full denture, than with a single implant. A similar conclusion could be drawn from the simpler model of Figures 6 and 10b, but now the presence of incorrect boundary conditions tends to hide this effect. Thus, it is difficult, on the basis of these results only, to obtain a clear idea about the importance of simulating the whole set of teeth, rather than few adjacent ones; the result shown in Figure 10a (which represents, among all those reported here, the "best" one) would indicate, however, that at least two teeth for each side of the studied implant should be included into the model.

Similar indications are provided by the analyses run for horizontal loading. Figures 11a and 12a again prove that the study of an isolated implant leads to incorrect results, underestimating the more accurate ones (note that here the scale of stresses is 1/5 than that for vertical loading). Again, at least two adjacent teeth, on both sides of the implant, seem to be severely involved by this loading condition; their presence influences appreciably the stress state around the studied implant. The simpler model of Figures 11b and 12b is not capable of giving the same information with the same strength, no doubt, again, owing to the effect of the boundary conditions. However, the displayed tendency is the same as before.

To conclude, Table 2 shows, for ease of comparison, the peak von Mises stress values in all the models analyzed in this and the previous Section under vertical loading conditions, together with some information about the size of the numerical models. The first column reports the peak values in the cortical bone; note that in the meshes including the full denture, unlike what happens in the edentulous case, the maximum stress values are not found in the proximity of the loaded implant, but around the adjacent tooth, on the same side of the jaw. The second column reports the peak stresses in the spongious bone at its interface with the cortical one, and the third column concerns the peak stress values in the spongious bone directly below the implant. It is apparent that neither the shell nor the plane models can provide "realistic" results. The analysis of the results shows also that in the absence

Table 2 : Peak von Mises stresses

Model	**Max. Mises in cortical [MPa]**	**Max in spong. below cort. [MPa]**	**Max in spong. below impl. [MPa]**	**Number of elements**	**Number of nodes**
3D all teeth (Fig. 5)	83	6.6	1.9	221,986	52,709
3D edent. (Fig. 1)	48.9	1.4	1.6	33,768	9,075
3D edent. no spong.	125.6			135,801	42,386
Partial 3D (Fig. 2)	43.5	6.2	7.4	39,490	9,191
Shell (Fig. 3a)	20.1			1,084	1,120
Shell + 3D (Fig. 3b)	40.8			2,352	3,162
Pl. stress (Fig. 4a)	11.1	0.5	0.8	2,520	1,334
Pl. stress (Fig. 4b)	17.8	0.7	3.3	2,420	1,284

of the spongious bone in the numerical model the peak stress values are found in the interior part of the cortical bone, whereas, in the presence of the spongious, the peak stress values in the cortical bone arise in the upper part of the bone.

These results point towards the conclusion that, in order to predict with accuracy the stress state around an implant, one needs a solid, three-dimensional mesh, including at least four teeth (two for each side of the implant), possibly with a simplified geometry but with great attention paid to the boundary conditions. The stiffness of the non-meshed portion of the jaw is in fact quite important to the intensity and distribution of the stresses around the implant under analysis. Simpler models can be used only with the full consciousness that they introduce significant sources of error, usually with a tendency to underestimating the real stresses.

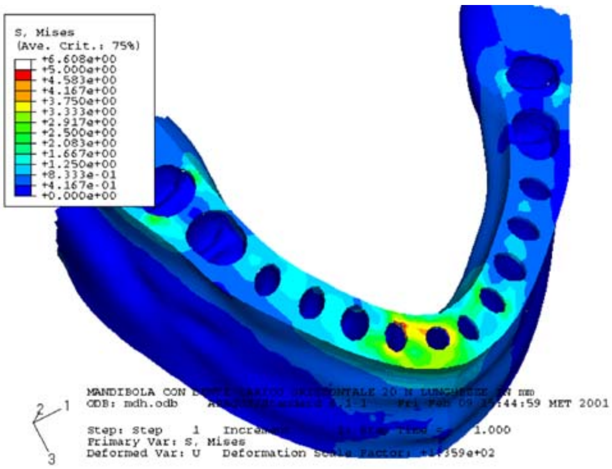
4 Material Modeling Issues, With Special Reference to the Presence of the Periodontal Ligament

When the subject of the investigation is the detailed stress analysis in the surroundings of a tooth root, it is essential to account for the presence of the periodontal ligament (hereafter shortened as PDL). While all the other materials can be treated, as a first approximation and for short-term loading, as linear elastic, the PDL cannot. The situation may be different if one is interested in the global biomechanics of the jaw, in which case the presence of the PDL can be neglected, or if one is interested to long-term loading. In this last case the correct material modeling of the jaw becomes a really tough problem, since viscosity-like and remodeling effects need to be taken into account; here we do not comment on this topic, outside the purposes of our present work.

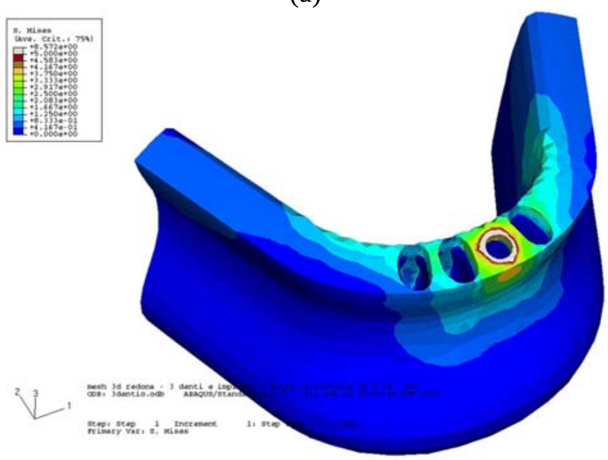
Even if one restricts his interest to the analysis of short-term loading on a single tooth, his task is still a difficult one; beside the modeling issues addressed in the previous Sections, there is the next one related to the inclusion of the PDL into his mesh. The presence of the periodontal ligament has been considered, so far (among others, in Andersen et al., 1991a, 1991b, Korioth et al., 1994a, 1994b, McGuinness et al., 1991, Middleton et al., 1996, Rees and Jacobsen, 1997), essentially in terms of a linear elastic or viscoelastic behavior. Such an assumption is clearly poor for soft tissues, invariably characterized by a strong nonlinearity even in the elastic range, with tendency to locking (Fung, 1993); this difficulty reflects in the fact that the proposed Young modulus for the periodontal ligament varies from 0.07 MPa to 1750 MPa (Rees and Jacobsen, 1997).

From the mechanical point of view, and confining our attention to short-term loading only (thus ruling out orthodontic loading and remodeling phenomena), the tooth-PDL-bone system can be modeled as a composite structure, characterized by a small tooth mobility, almost completely due to the large deformation of the PDL, so that tooth and bone work in a small-strain regime. The work done by Pini (1999) provides both experimental results (for bovine teeth) and a nonlinear elastic constitutive model proposed to predict some features of the observed behavior.

This constitutive model is thought for use in a three-dimensional description of the PDL, discretized by means of solid finite elements. Yet this way of modeling presents a series of difficulties, mainly in the numerical analysis of the problem. In essence, these may be

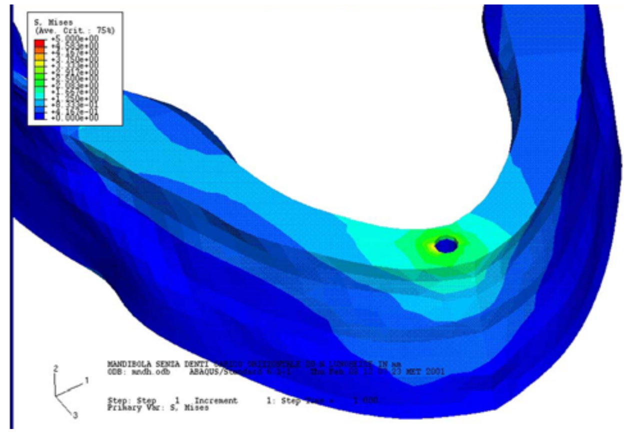


(a)

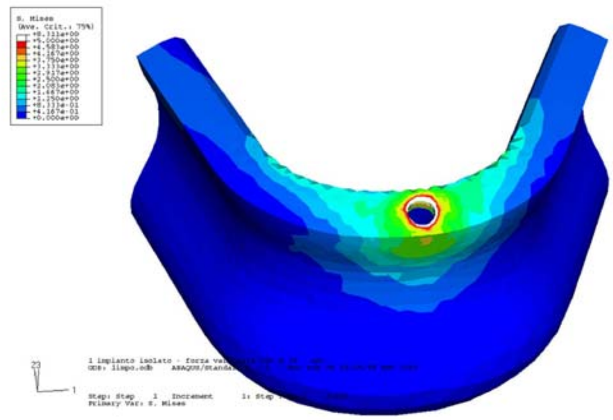


(b)

Figure 11 : Three-dimensional meshes of the jaw with teeth and implant, von Mises stress contours, horizontal loading condition: (a) full mesh; (b) mesh of the frontal portion.



(a)



(b)

Figure 12 : Three-dimensional meshes of the edentulous jaw with implant, von Mises stress contours, horizontal loading condition: (a) full mesh; (b) mesh of the frontal portion.

summarized as follows:

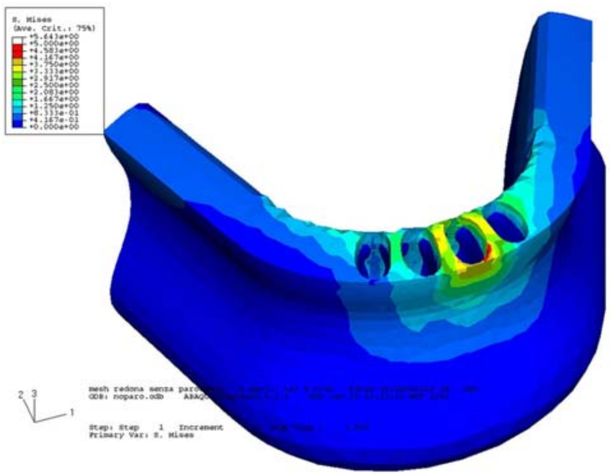
1. the global behavior of the tooth-PDL-bone system depends on the specific conditions of the surfaces connected by the ligament, not only on the ligament itself. In other words, from the methodological point of view, the introduction of the PDL in the mathematical model creates two additional interfaces: the bone-PDL and PDL-tooth interfaces;

2. the use of a hyperelastic constitutive law implies recurring to a large strain formulation for the whole problem; this is superfluous for the largest part of the model, which always remains within a small strain regime;

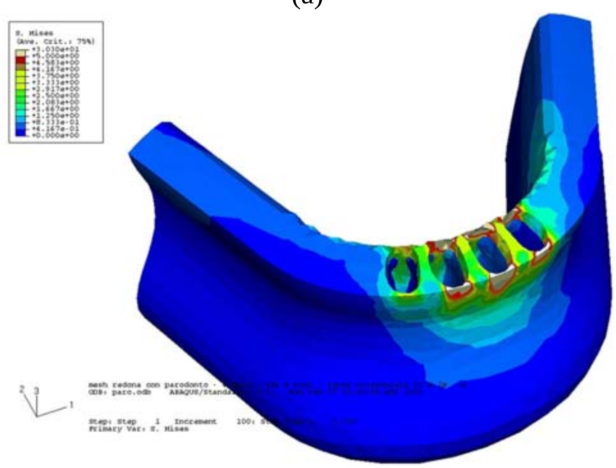
3. the use of large strain hyperelastic continuum constitutive laws introduces the need of identifying several parameters, often lacking a direct physical meaning; moreover, such models often exhibit strong sensitivity to parameters difficult to characterize experimentally, as well as instabilities at unrealistically small values of deformation, thus making the numerical analysis quite complicated;

4. another difficulty is specifically related to the use of a Finite Element technique. A correct discretization of the PDL as a continuum requires indeed a fine, three-dimensional mesh, owing to the small thickness of the PDL itself. On the other hand, a "coarse"

mesh is sufficient for describing bone and teeth. The necessary smooth transition between the characteristic lengths of the two meshes implies the use of a large number of finite elements, with the consequent increase of the computational burden.



(a)



(b)

Figure 13 : Three-dimensional mesh of the frontal portion of the jaw with four teeth, horizontal loading condition: (a) von Mises stress contours in the absence of the periodontal ligament; (b) von Mises stress contours in the presence of the periodontal ligament (from Gei et al., 2002).

To overcome all these difficulties it appears necessary to introduce simplifications both on the geometry and on the material description.

A reasonable solution is to model the PDL as a zero-thickness *interface*, governed by a simplified, ad hoc constructed, phenomenological constitutive law. In this way the PDL does not represent a third material subjected to large strains, as inevitable in a continuum description, and the analysis falls within the small-strain regime.

Such an idea has been pursued in Gei et al. (2002), starting from the experimental evidence given in Pini (1999) and in Pini et al. (2000). These tests are unfortunately done on bovine teeth only, and under rather unrealistic laboratory conditions. However, the relevant results are detailed enough to allow the definition of a simple interface model which, in turn, enables one to construct Finite Element models of the jaw-PDL-tooth system taking into account the essential effects associated to the presence of the PDL. Some results are summarized in Figure 13, taken from Gei et al. (2002).

These refer to the analysis of the model of Figure 9b in which 4 teeth (and no implant) are now present: Figure 13a illustrates the von Mises equivalent stress under the application of a transversal load of 20 N on one of the two central incisors, in the absence of PDL (i.e., for a perfect bond between the tooth root and the surrounding bone), whereas Figure 13b gives the same information taking into account the presence of the PDL, simulated as a zero-thickness interface. All the other parts of the mesh are defined as linear elastic and isotropic, using the same material data as shown in Table I above. The material behavior of the PDL, introduced in the interface model, is illustrated in Figure 14, where the considered normal stress-strain (a) and shear stress-strain curves (b) are shown, both as experimental points, taken from Pini (1999) and Pini et al. (2000), and as adopted analytical expressions.

The difference between the two situations is quite apparent, and does not need much comment; these differences are less evident under a purely axial loading condition (not shown here — more details can be found in Gei et al., 2002). It may also be argued that in the presence of a better representation of the boundary conditions these differences should decrease, since the global reduction of stiffness should somehow reduce the effect of the extremely low initial stiffness of the PDL.

In any case it is apparent that the presence of the PDL cannot be neglected in any analysis of the tooth/bone system. With reference to the results shown in Figure 13, it can be safely said that (i) no other simpler approach to this problem would provide analogous results, which, by the way, even in the case of the "simple" PDL model illustrated in Figure 14, correspond to a mobility curve for

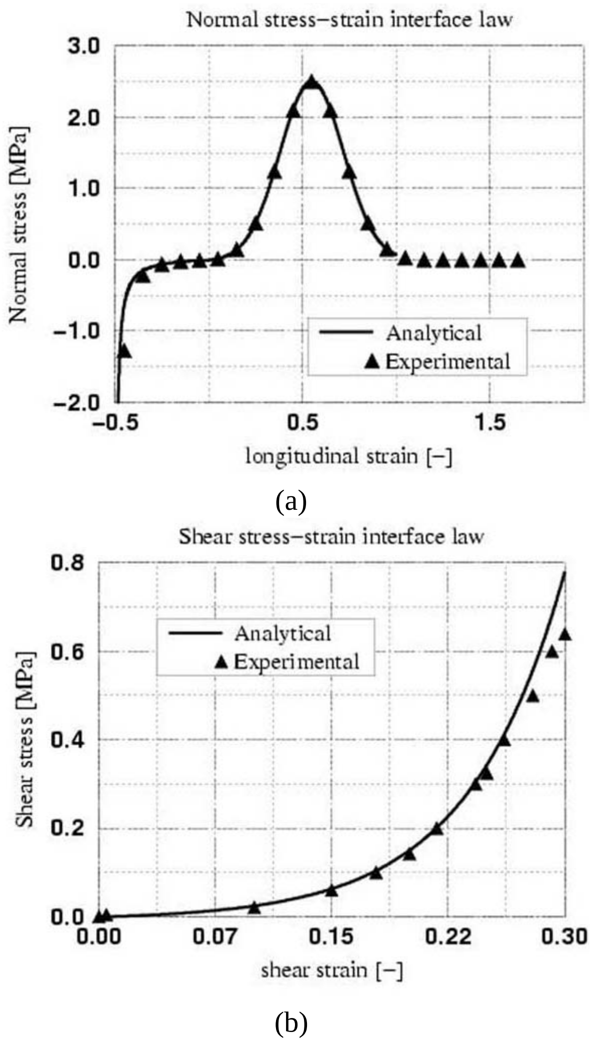


Figure 14 : Uniaxial nonlinear stress-strain curves for the periodontal ligament treated as an interface: (a) normal stress behavior; (b) shear stress behavior. Black triangles are experimental data taken from Pini (1999) (Figure 13a) and Pini et al. (2000) (Figure 13b); the solid lines correspond to the interface model implemented in Gei et al. (2002).

the tooth qualitatively similar to those reported, for instance, in Parfitt (1960); that (ii) the use of a continuum mesh for the PDL, coupled with a hyperelastic model (Pini, 1999; Pini et al., 2000; Natali et al., 2000), would have furnished similar results at a much higher price and with many more difficulties; and that (iii) better results could be obtained by using more refined models for the PDL, but without the need of further complicating the description, in terms of material properties, of the "hard" parts of the system (again, this is valid only for short-term loading).

5 Further Issues and Conclusions

The mathematical description of the stress state around a tooth or a fixed dental implant is a problem exhibiting a high degree of complexity, and the topics discussed in this work are just a part of the approach to the definition of a correct model. There are other issues to be taken into consideration, which cannot be treated in detail here for lack of space (and of experience on the authors' part). We can only list some of these issues, as possible research fields.

The possibility and convenience of approaching the numerical analysis of the jaw-teeth/implants system by means of the Boundary Element method, instead of the widely popular Finite Element one, is an example. The Boundary Element method appears indeed to be a good candidate as a solution tool for the problem under discussion here, in that it can treat different zones of linear elastic materials, separated by nonlinear interfaces, in a very effective way, without the need of discretizing the whole volume but requiring only the discretization of its boundary. Work is in progress towards the implementation of such a code in a three-dimensional version; preliminary comments are given in Salvadori (2000).

Another issue which should be addressed, in order to understand its relative importance, is the full description of unilateral contact with friction, quite significant at least in the analysis of implants, made by several independent parts connected to each other, and to the bone, by means of screwing devices of various types. The inclusion of contact into a finite element analysis is possible (and already done, for instance, in Sakaguchi and Borgersen, 1995), but extremely expensive, specially in a three-dimensional context. It would be nice to understand if such a feature needs a detailed description, or if its presence can be taken into account in a simplified manner.

The loading condition is of course another difficult problem. In a full three-dimensional context it is possible to simulate the forces exerted by the single muscles at their insertion points (as done, for instance, in Korioth et al., 1994a, 1994b); but the same becomes increasingly less feasible as the simplification of the model increases. In the extreme case of a small three-dimensional mesh, or of a two-dimensional mesh, it is necessary to define the loads directly on the studied tooth or implant, and, in doing so, care must be taken in distinguishing between the application of forces or of displacements. Also this topic would require a comparative analysis, in order to make clear what are the proper loads to apply to a tooth or implant in order to reasonably approximate the effect of the real masticatory load.

Another interesting aspect concerning the loading type is related to the inclusion of dynamics effects into the analysis. This would pose additional material modeling problems, since it is expected that damping would play an important role.

We do not add further comments on the material modeling subject, specially for the long-term loading condition, and related remodeling issues — too much a complex problem to be even briefly hinted at.

The comparative analyses described in this work allow to formulate some conclusions in terms of the best compromise between accuracy of results and computing price. Keeping in mind that we have not addressed issues related to long-term loading, and that we are specifically interested in the detailed analysis of the stress state around a tooth or a fixed osteointegrated implant, we have observed what follows:

1. two-dimensional models should be avoided. This conclusion corroborates what said in Meijer et al. (1993b); it can be slightly relaxed only when comparative analyses are done, and there is no way to run three-dimensional analyses. In such extreme cases care should be taken in properly defining suitable boundary and loading conditions, in such a way as to avoid both the use of incompatible meshes and the introduction of unrealistic stress concentrations. Furthermore, if the loading is defined as directly applied on the examined tooth/implant, care should be taken in deciding whether such a load corresponds to a force or a prescribed displacement, in order to avoid a misinterpretation of the results;

2. a full three-dimensional model is not necessary for the purposes stated above. The best compromise, which also does not require either sophisticated geometry acquisition tools or heavy mesh generation steps, appears one in which the geometry of the jaw is reduced to a small part surrounding the analyzed object, without the need of introducing too many geometry details. However, such a part should be discretized by means of solid, continuum elements (i.e., shell elements should be avoided). The boundary conditions can be either defined in terms of self-equilibrated loads/compatible displacements derived from previous three-dimensional analyses, to be applied on the studied portion of the jaw, or — in a simpler but still effective way — defined by replacing a large portion of the jaw, far from the studied detail, with "equivalent" spatial elements (beams or shells);

3. the effect of adjacent teeth should not be neglected; however, in the same spirit as in the above item 2., it seems that the inclusion of just some surrounding teeth would be enough;

4. in the case of the analysis of a normally functioning tooth it is essential to try and well describe the existence of the periodontal ligament (as an interface), whereas, at least for short-term loading, an extremely accurate description of teeth and bone seems to be less important;

5. the analysis via Boundary Element method appears to be a promising way to deal with this problem, if the only important nonlinearity is confined into the periodontal ligament. "Meshless" methods might turn out to be competitive as well.

Even if a truly realistic description of such a complex system would require a three-dimensional, time dependent, inelastic, non-isotropic model, possibly taking into account fluid-solid interaction (in the periodontal ligament), with the further complication of the description (specially in the case of the analysis of implants) of several surfaces of unilateral contact with friction, we feel that, in order to individuate the stress state around a healthy tooth, and thus to define the design target of an optimum implant, simplified analyses like those above described should constitute a reasonably valid starting point.

Acknowledgement: Work done within the research project "Criteri di progetto per impianti dentali ottimizzati rispetto alla stabilità biomeccanica dell'interfaccia osso-impianto", financed by the Italian Ministry of University and Scientific and Technologic Research (MURST).

Thanks are due to Professors Corrado Paganelli, Stefano Salgarello, and Pier Luigi Sapelli, of the Odontoiatric Clinic of the University of Brescia, for many helpful discussions.

We acknowledge the help of Drs. Giorgio Ceruti, Andrea Leali, Federico Mennuti, and Adriano Redona in the generation of three-dimensional finite element meshes.

The finite element code ABAQUS has been run at the Department of Civil Engineering of the University of Brescia and at the Department of Structural Engineering of the Politecnico of Milano under academic licenses.

References

Andersen K. L., Pedersen E. H., Melsen B. (1991a), Material parameters and stress profiles within the periodontal ligament, *Am. J. Orthod. Dentofac. Orthop.*, **99**, 427–440.

Andersen K. L., Mortensen H. T., Pedersen E. H., Melsen B. (1991b), Determination of stress levels and profiles in the periodontal ligament by means of an improved three-dimensional finite element model for various types of orthodontic and natural force systems, *Journal of Biomed. Eng.*, **13**, 293–303.

Atluri S. N., Shen S. (2002a), *The meshless local Petrov-Galerkin (MLPG) method*, Tech. Science Press, CA, USA.

Atluri S. N., Shen S. (2002b), The meshless local Petrov-Galerkin (MLPG) method: a simple and less-costly alternative to the Finite Element and Boundary Element methods, *CMES: Comp. Modeling in Engineering & Sciences*, **3**(1), 11–52.

Brunski J. B. (1992), Biomechanical Factors Affecting the Bone-dental Implant Interface, *Clinical Materials*, **10**, 153–201.

Capsoni A., Ceruti G., Corradi L. (1999), A Finite Element model for the analysis of local stress peaks induced by dental plants (in Italian) Proc. XIV National Congress of the Italian Association of Theoretical and Applied Mechanics (AIMETA), Como (I), 6–9 october, 1999.

Carini A., Salvadori A., Sansoni G. (2000), Numerical analysis of the jaw-teeth system, Proc. XIII Convegno Italiano di Meccanica Computazionale, Università di Brescia, 13–15 november, 2000.

del Valle V., Faulkner G., Wolfaardt J. (1998), Distribuzione delle deformazioni prodotte da impianti craniofacciali osteointegrati: uno studio numerico (in italian), *Quintessence International*, **4**, 119–130.

Fogazzi P., Manfredini P., Genna F. (2000), A simple interface element for the numerical modelling of the periodontal ligament, Proc. XIII Convegno Italiano di Meccanica Computazionale, Università di Brescia, 13–15 november, 2000.

Fung Y. C. (1993) *Biomechanics: Mechanical Properties of Living Tissues*, Springer-Verlag.

Gei M., Genna F., Bigoni D. (2002), An interface model for the periodontal ligament, *ASME J. Biomechanical Engineering*, **124**(5), 538–546.

Hart R. T., Hennebel V. V., Thongpreda N., Van Buskirk W. C., Anderson R. C. (1992), Modeling of the biomechanics of the mandible: a three-dimensional finite element study, *Journal of Biomechanics*, **25** (3), 261–286.

Hibbitt, Karlsson & Sorensen (2000), ABAQUS User's Manuals, Release 6.1, Pawtucket, RI, USA.

Korioth T. W. P., Hannam A. G. (1994a), Deformation of the Human Mandible During Simulated Tooth Clenching, *Journal of Dental Research*, **73** (1), 56–66.

Korioth T. W. P., Dent C., Hannam A. G. (1994b), Mandibular Forces During Simulated Tooth Clenching, *Journal of Orofacial Pain*, **8**, 178–189.

Lavernia C. J., Cook S. D., Weinstein A. M., Klawitter J. J. (1982), The influence of the bone-implant interface stiffness on stress profiles surrounding Al_2O_3 and carbon dental implants, *Annals of Biomedical Engineering*, **10**, 129–138.

Meijer H. J. A., Starmans F. J. M., Steen W. H. A., Bosman F. (1993a), A three-dimensional, finite-element analysis of bone around dental implants in an edentulous human mandible, *Archives of Oral Biology*, **18** (6), 491–496.

Meijer H. J. A., Starmans F. J. M., Bosman F., Steen W. H. A. (1993b), A comparison of three finite element models of an edentulous mandible provided with implants, *Journal of Oral Rehabilitation*, **20**, 147–157.

McGuinness N. J. P., Wilson A. N., Jones M. L., Middleton J. (1991), A stress analysis of the periodontal ligament under various orthodontic loadings, *European Journal of Orthodontics*, **13**, 231–242.

Middleton J., Jones M., Wilson A. (1996), The role of the periodontal ligament in bone modeling: The initial development of a time-dependent finite element model, *American Journal of Orthodontics and Dentofacial Orthopedics*, **109** (2), 155–162.

Natali A., Pavan P., Pini M., Ronchi R. (2000), Numerical analysis of short time response of periodontal ligament, Proc. 12th Conference of the European Society of Biomechanics, Dublin, Ireland.

Parfitt G. J. (1960), Measurement of Physiological Mobility of Individual Teeth in an Axial Direction, *J. Dent. Res.*, **39**(3), 608–618.

Patra A. K., D'Souza K. S., Meenaghan M., deTolla D. (1999), Simulation of Dental Implants Under Cyclic Loading Using a Voronoi Tessellation Based Frame Model of Trabecular Bone and *p* version finite elements for Compact Bone, Proc. X Conference on The mathematics of finite elements and applications (MAFELAP 1999), Brunel University, 22–25 June 1999.

Pini M. (1999), Mechanical Characterization and Modeling of the Periodontal Ligament, PhD Thesis, University of Trento, Italy.

Pini M., Vena P., Contro R. (2000), Parameter identification of a non-linear constitutive law for the periodontal ligament allowing for tensile and shear laboratory tests, Proc. XIII Convegno Italiano di Meccanica Computazionale, Università di Brescia, 13–15 november, 2000.

Prandini M., Campi M., Fogazzi P. (2000), Identification of a model for the stress-strain behavior of the periodontal ligament, Proc. XIII Convegno Italiano di Meccanica Computazionale, Università di Brescia, 13–15 november, 2000.

Rees J. S., Jacobsen P. H. (1997), Elastic modulus of the periodontal ligament, *Biomaterials*, **10**, 995–999.

Richter E.-J., Orschall B., Jovanovic S. A. (1990), Dental implant abutment resembling the two-phase tooth mobility, *Journal of Biomechanics*, **23** (4), 297–306.

Rieger M. R., Adams W. K., Kinzel G. L., Brose M. O. (1989), Alternative materials for three endosseous implants, *The Journal of Prosthetic Dentistry*, **61** (6), 717–722.

Sakaguchi R. L., Borgersen S. E. (1995), Nonlinear Contact Analysis of Preload in Dental Implant Screws, *The International Journal of Oral and Maxillofacial Implants*, **10** (3), 295–302.

Salvadori A. (2000), Symmetric Galerkin BEM for domains connected by cohesive interfaces: formulation and implementation, Proc. XIII Convegno Italiano di Meccanica Computazionale, Università di Brescia, 13–15 november, 2000.

Tanne K., Sakuda M., Burstone C. J. (1987), Three-dimensional finite element analysis for stress in the periodontal tissue by orthodontic forces, *Am. J. Orthod. Dentofac. Orthop.*, **92**, 499–505.

Vaillancourt H., Johnson W. R., Pilliar R. M. (1990), A finite element model for porous implants, *Development and Design with Advanced Materials*, G. C. Sih, S. V. Hoa, J. T. Pindera (Editors), Elsevier, 207–217.

van Zyl P. P., Grundling N. L., Jooste C. H., Terblanche E. (1995), Three-Dimensional Finite Element Model of a Human Mandible Incorporating Six Osteointegrated Implants for Stress Analysis of Mandibular Cantilever Prostheses, *The International Journal of Oral & Maxillofacial Implants*, **10** (1), 51–57.