COMPUTATIONAL CAD/FEM TECHNOLOGIES APPLICATION TO DENTAL MEDICAL FIELD

Carlo Maggior

Enabling Digital Dentistry s.r.l., Milano, Italy
email: enabling@enablingdigitaldentistry.it

Giuseppe Miccoli

IMAMOTER Institute – National Research Council of Italy, Ferrara, Italy

The aim of the study here presented would be that of showing the capabilities of a convincing medical and technical methodology aimed at developing a dental prosthesis forecast model and a simulation tool with state-of-the-art acceptable validation. A FEM model of the dental prosthesis was developed taking into account shape and material characteristics of the different parts of the prosthesis. Starting from patient specific raw data and CAD representations, stress and displacements were computed on specific contact tooth areas and connecting parts by means of a FEM linear static analysis. Boundary conditions and loads were defined and applied following an at present normally used patient specific simulation in personal medicine. More realistic load values in the contact bite areas could be experimentally measured and acquired by means of on purpose designed load cells. The main advantage of the proposed method consists in identifying and pointing out the dental prosthesis area with the highest and hazardous stress by means of a fast, reliable and automated procedure. So that the technical analyst can intervene immediately to reduce or eliminate the problem by varying dental prosthesis morphology and thickness, moreover improving material characteristics and/or testing new bio-compatible components. This way an optimal prosthesis can be realized without iatrogenic damage, in particular in complex implant projects. On this purpose, a new application is presented here dealing with structural behavior and bone modeling around implant by comparing zirconia and titanium FEM analyses results. Some valuable conclusions can be utilized for the implementation of zirconia implants, more bio-compatible than the usual titanium ones.

Keywords: dental prosthesis, prevention methodology, FEM, zirconia implants

1. Introduction

The term FEA (Finite Element Analysis) refers to the computation method by means of finite elements that allows the execution of engineering simulations in the solid mechanics area. Dental FEA consists in an application of such a technique in the dental field which allows the simulation of the structural behaviour of the CAD prosthodontic project in the binary STL format (acronym for STereoLitography or Standard Tesselation Language, native format of CAD stereolithography). This way indications of primary importance are provided to the analyst in terms of process, optimization and simulation to achieve a good result and move forward to the CAM phase. At present, the prosthesis – for instance, coping, crowns, bridges with connectors, Toronto bridges with cantilever
– modelled by a dental CAD code presents a morphology varying in terms of thickness and forms by virtue of the bite conditions of the application considered and the material type chosen for the prosthesis rehabilitation. The modelled sample is developed referring to the own expertise or relying to the average parameters pre-set by the software houses. These parameters do not guarantee, always and for all the conditions, the achievement of a suitable prosthesis to be placed inside the oral hollow. The prosthesis indeed has to avoid breakages and has not to generate stresses on the dental and bone structures subjected to an immediate and fatigue load, this way causing failures. In particular, the situation is critical for complex rehabilitations and mainly for plants where the connections are rigid as lacking in the natural periodontal damping apparatus.

The dental FEA simulation allows the analyst to reduce or eliminate many of these problems by locating the most stressed areas visually in the results post-processing phase with the capability of changing the prosthesis morphology and thicknesses and by optimizing the chosen materials.

The main aim of the present paper consists in making understandable to the most part of the readers novelty and effectiveness of the method in the dental medical field. For this reason the choice has been that of illustrating computation and simulation foundations only by focussing on some interesting applications of Dental FEA, strictly correlated to real cases of intervention and rehabilitation in the dental medical area.

At present the majority of dentists use titanium of 4° (pure titanium material) or 5°, i.e. an alloy more resistant to break and traction and made of the following elements: V 3.5 %, Al 6%, O 0.2%, Fe 0.4 %, H 0.01 %, C 0.1 %, N 0.05 %, for dental implant and abutment. The aim of the application consists in probing the possibility of introducing zirconia in the usually workflow, a new dental material more bio-compatible than titanium. So that one of the objective of the present paper would be a better understanding of the possibility of using implants with straight and angle abutment by comparing zirconia vs titanium performance. On this purpose FEM models have been developed with two congruent 3D solid meshes referring to custom patient specific mandible and realistic implant abutment model.

We considered a first model with realistic dental implant with a straight and 20° angle abutment. A static structural analysis was carried out taking into account different choices of zirconia e titanium 4° or 5° (the two titanium kinds common used) and various applied custom patient bite loads and constrains in the root of the implant. Von Mises stress has been computed as comparison parameter for the different cases analysed. In particular, our main interest has to be referred to the structural response by the implant alone and that of the same implant located inside a specific site bone of the patient mandible. This second case simulates the integration of the implant in the bone and we found a totally different stress referring to the previous case.

This study turns out to be remarkable as it shows the behaviour of an implant located in a specific bone mandible position in a confident way and realistically. That allows us to understand the possibility of using zirconia as implant material, with a better bio-compatibility than titanium for health solution and user application care.

2. Computational methods and results

The dental implant abutment model is a 3D object polydata then saved in a binary STL mesh file modelled by the common dental CAD software (Fig. 1), i.e. a mesh that describes the surface of the tri-dimensional object only with no other attribute by means of a series of triangles structured in a vector way. The mesh has to be analysed by suitable tools in order to locate possible defects and adjust them by means of an STL geometric model CAD optimization procedure.

The STL mesh so obtained is converted in formats compatible with the FEA code, making use of many kinds of formats, Step and Nastran for this application, by other codes and imported in FEA. The real shape of the prosthesis is defined in the pre-processing phase by mathematical information. The development of a 3D solid mesh is carried out by adding the necessary parameters to represent the boundary conditions at the nodes that define the model geometry. The mesh quality aspect ratio
is checked. Lastly the isotropic mechanical properties of the prosthesis materials are defined, as Young modulus, Poisson ratio and density, as for zirconia, Ti 4° or 5° considered in the present application (Table 1) and constraints and loads are applied (Figs. 2 and 3).

![Figure 1: STL binary](image1)

Table 1: Mechanical properties of the simulated materials

<table>
<thead>
<tr>
<th>Materials</th>
<th>Young modulus (E = N/mm²)</th>
<th>Poisson ratio</th>
<th>Density (Kg/m³)</th>
</tr>
</thead>
<tbody>
<tr>
<td>ZIRCONIA</td>
<td>201000</td>
<td>0.278</td>
<td>5600</td>
</tr>
<tr>
<td>TITANIUM PURE</td>
<td>155000</td>
<td>0.192308</td>
<td>4222.71</td>
</tr>
<tr>
<td>Ti 4° IMPLANT BODY</td>
<td>105200</td>
<td>0.19</td>
<td>4600</td>
</tr>
<tr>
<td>Cortical bone</td>
<td>17000</td>
<td>0.34</td>
<td>1280</td>
</tr>
<tr>
<td>Trabecular bone</td>
<td>350</td>
<td>0.25</td>
<td>1280</td>
</tr>
</tbody>
</table>

![Figure 2: Multiple constraints selected in implant body osseous integration](image2)
2.1 Implant abutment straight, zirconia vs titanium

The mandible bone patient specific model is developed starting from a TCCB (tac cone beam), i.e. a specific type of tomography used in dental field. The main object 3D optimized polydata is developed by an algorithm that consider the pixel data sensor density and then converted in a mesh STL binary file. The file, converted in Nastran and imported in Midas FEA, allows to obtain a 3D meshed and CAE optimized model of the mandible and dental implant (Fig. 4).

Figure 3: Multiple loads, 100 N

Figure 4: Mandible and dental implant 3D congruent mesh

Many structural static linear analyses were carried out (Midas code) by planning different working conditions, as for instance different loads, constraints and materials properties. The software can solve each case independently or simultaneously. The FEM model unknowns consist in the model nodes displacements and starting from them the solver (MSC Nastran) computes finite elements deformations and stresses taking into account materials properties. We considered 20 iterations for each load case and a convergence tolerance 0.001.

The results of such a kind of analyses are represented usually in terms of node displacements and finite elements von Mises stress colour plots. Iconographic representations of different stress locations and nominal values can also be obtained with various effects in the post-processing phase by means of pictures and films also in Augmentation Reality. In particular, the von Mises stress representation is one of the most used.

A CAD format STL binary file was created for an implant abutment straight and a tetrahedral elements mesh developed in Midas FEA. Zirconia material properties were assigned (Table 1) and multiple surface constrains selected and multiple custom patient specific loads applied. Some results of the analysis are shown in Fig. 5 as von Mises stress.
2.2 Implant abutment angles, zirconia vs titanium

A second dental applications was considered making use of the Dental FEA method, taking into account an implant abutment angles of 20° and comparing zirconia and titanium type 4° materials. Fig. 6 represents an STL refinement of the implant abutment 20° and Fig. 7 shows the von Mises stress results comparison between the two simulation models considered.

![Diagram showing implant abutment 20° STL refinement](image)

Figure 6: Implant abutment 20° STL refinement

![Diagram showing implant abutment angle 20° von Mises stress results, zirconia vs titanium type 4°](image)

Figure 7: Implant abutment angle 20° von Mises stress results, zirconia vs titanium type 4°

2.3 Forecast model dental implant abutment straight – mandible bone

This represents the most complex application we have considered. A remodelling was carried out of the bone around the implant by starting directly from the provisional mesh model of the implant already fixed inside the bone (Fig. 8). The Fig. 9 shows the von Mises stress results in a small area when the straight implant abutment is integrated into the mandible bone.
2.4 Dental FEA for cement bridge

Another Dental FEA application we considered is the following dealing with a cement bridge (Fig. 10), where the inner size of the crowns and constraints locations are shown. The loads are applied on the bite surface with great accuracy thanks to the available data taking into consideration contact surface areas and force directions. (Fig. 11). The forces are considered as pressures with average values in the range of 50 N to 400 N depending on the bite conditions, more precisely 50 N to 100 N based on the dental typology considered and/or optimized by oral data transducer acquisitions [1], [2].

In particular, this example considered a cement bridge coping in zirconia. A tetrahedral mesh file was developed by the Nastran code and imported in Midas FEA and zirconia material properties were assigned (Table 1). Multiple surface constrains were selected and multiple custom patient specific loads applied. A result of the analysis is shown in Fig. 12 as von Mises stress.
3. Conclusions and next steps

The capabilities of a convincing medical and technical methodology are here illustrated. Dental FEA aims at developing a dental prosthesis forecast model based on a FEM simulation procedure with state-of-the-art acceptable validation. In particular, a new application is here presented dealing with structural behavior and bone modeling around implant by comparing FEM analysis results making use of different prosthesis constitutive materials, i.e. zirconia and titanium. So that some valuable conclusions could be utilized for the implementation of zirconia implants, more biocompatible than the usual titanium ones.

The analysis takes into consideration an implant abutment straight made by zirconia or titanium, in the double configuration, i.e. alone and mandible osseous integrated. A static linear FEM analysis is carried out starting from different boundary conditions in terms of simulation model loads and constraints. The main aim of the analysis consisted in comparing values and locations of von Mises stress computed in implant positions and contact points of particular interest when the two different materials are utilized. Computation results seem to emphasize that in both cases, i.e. implant abutment straight alone and mandible osseous integrated, von Mises stress shows lower and allowable values when making use of zirconia, in particular in the second case. This way a zirconia implant abutment straight turns out to be interesting for a prosthesis application, moreover being more biocompatible than the titanium one of common use nowadays.

Different conclusions for the other case considered, i.e. implant abutment angle 20°. Really a serious problem could be the excessive von Mises stress value computed at the implant neck making use mainly of zirconia. This way the implant safety could be threatened, at least for these severe abutment angles.

Further analyses have to be carried out but right now it is possible to forecast the implementation of zirconia for dental prosthesis implants as some international brands already are doing, even if this kind of marketing is really at the beginning.
Some remarks follow about Dental FEA and future implementations.

The dental prosthesis areas with the highest stress can be localised visually by means of Dental FEA, this way allowing the analyst to reduce or eliminate the stress by changing morphology and thickness and optimizing the plant chosen materials. The post-processing phase results give a precious indication for an optimized prosthesis more suitable for the specific individual conditions of the patient by simulating his physiologic, pathologic and functional state changing in time (fatigue analysis). In particular, this analysis results are essential for prostheses rigidly connected to plants where the stress strikes again on the bone around implant by influencing the bony homeostasis and for prostheses with complex and extended design. An aim of the Dental FEA method could be a prosthesis model automatically improved starting from a binary STL format file to be corrected and optimized taking into consideration the stress FEM analysis results. This way the Dental FEA method would propose by itself a prosthesis model more suitable to support the stress once cemented in the oral hollow.

A note: referring to the methodology above described and the different applications illustrated, the Dental FEA method takes into consideration only the averaged load values acting on the teeth at the contact areas. Certainly that represents a limit of the method. Anyway a project is already planned in the near future aiming at measuring the real load values by means of MEMs, this way achieving an optimization of the prosthesis model for an even more suitable and reliable plant.

REFERENCES