Three-dimensional finite element analysis of the stress distribution around implant- supported and tooth-implant supported versus teeth-supported monolithic zirconia fixed prosthesis: In-vitro study

Eshraq Rajeh 1, Lamia Dawood 1, Mohammed H. Ghazy 1, Manal Abo Madina 1

1 Lecturer of Fixed Prosthodontic, Faculty of Dentistry, Sanaa University, Yemen
2 Associate Professor of Fixed Prosthodontist, Faculty of Dentistry, Mansoura University, Egypt
3 Professor and Head of Fixed Prosthodontic Department, Faculty of Dentistry, Mansoura University, Egypt
4 Professor of Fixed Prosthodontic Department, Faculty of Dentistry, Mansoura University, Egypt

Abstract:

Aims: To compare the distribution of stress formed around an implant and a natural tooth on different zirconia FDP design (implant supported, tooth-implant supported and teeth supported) by 3D finite element analysis.

Materials and Methods: A distal extension situation was utilized in this study to evaluate stress distribution around a natural tooth and an implant on three different designs: implant supported, tooth-implant supported and teeth supported zirconia FDPs. The stress values of the three models loaded with oblique forces (300 N) were analyzed using (3D) finite element analysis.

Result: The results of this study revealed that the implant presence a highest influence on the stress concentration in the mandible, while the molar and premolar reflects a low stress concentration. It was noticed that Model 1 has the highest stress concentration more than the other two models this flowed by Model 2. On the other hand, the less stress concentration noticed around Model 3.

Conclusion: Highest stresses was noted around the implant neck on the cortical bone region than those of natural teeth.

Key words: implant, fixed prostheses, tooth implant, monolithic zirconia, finite element, stress.

Introduction

Implant-retained restorations have been commonly used by clinicians due to their high success rate that accompanied with this type of prostheses. 3 The longevity of the implant prosthetic system depends on many factors one of them is the strain in the components of the system and around the implant and the stress distribution in sequence is influenced by the design of both prosthesis and implant as well as the materials. Furthermore implant location, position; and quantity and quality of the bone. 4,5 Occlusion force results in stress that generated on the implant and close to the supporting tissues which has an important effect on successful osseointegration. 4,5 Once the occlusal force is transferred over the implant to the supporting bones during masticatory force, stress distribution of occlusal load as well as the biological response of the body (regeneration of the bone) can be essential factors afterward implant placement. Maximum of stress concentrates on the crest of alveolar bone as the osseointegrated implant contacts the bone precisely with no even minute movement. Consequently, such concentration of more stress will give rise to bone resorption and further may result in failure of implant prostheses. 6,7 Thus, to enhance the success rate of implant, surrounding bone resorption should be held in.

In the field of implant dentistry, the finite element (FE) method has become a progressively useful means for the prediction of stress impact on the implant and its surrounding bone with more accurate computer simulation and modeling technologies. 7 By definition, the finite element analysis is a type of a numerical analysis method for analyzing parts or assembly to firm up the performance of a product in the engineering arena. FEA procedure comprises a creation of solid model, calculating the response of the structure (deformation, stress) by generating finite element models in relative to the solid model and defining the use environment (condition of boundary as well as condition load), then finally display these by a diagram. As pre-process: a solid model is prepared, then followed by generation of a FEM model. As the solver process, finite element equations are established and solved, and in the post-process, the analysis result is processed and demonstrated in a way easy to understand. 8

This in-vitro study was performed to evaluate and compare the stress distribution in mandibular body and lower FDPs supported by three different substrates (three geometric models were prepared; one for implant supported FDPs, one for tooth-implant supported FDPs and one for teeth supported FDPs.

Materials & methods

The current study was a numerical and analytical study including two steps; the first step included the model construction, while the second step included a three-dimensional finite element program application and analysis of the given data.

Model construction:

Three epoxy casts (Exit 50, Egyptian Swiss For Manufacturing And Trading 6 October Egpy) mandibular casts were milled with the three different configurations as the following:

Model 1: Implant-implant supported 3-unit fixed prosthesis
Model 2: Tooth-implant supported 3-unit fixed prosthesis
Model 3: Tooth-tooth supported 3-unit fixed prosthesis

The epoxy casts of all groups were scanned using a 3D scanner (Cera Map400 AmannGirrbach, Germany) (Fig. 1) and modeled using commercial general purpose CAD/CAM software; “Nexx Seimenes” version 8.0 (Siemens, Parkway, TX, USA) for generating the geometrical models. After that it was exported to an analysis package. The finite element software, ANSYS16.2 was used to analyze the models. The models were processed in ANSYS to generate the meshed structure. Meshing divided the entire model into smaller elements. The elements are interconnected at specific joints called nodes. Once meshing and contacts are distinct, the next process is to define boundary conditions. After defining the boundary of the model, the loads to be applied were defined; a buccolingual load with 45 degree inclination of 300N was applied on the pontic of FDP by a 3D finite element ball model (5.8mm in diameter) to the occlusal surface of the lower first molar, and then the stress analysis was completed by the incorporation of material properties. The material properties were determined from values obtained from the literature.9,10,11

Bone block with a height of 29 mm, width of 12, and cortical bone thickness of 1.5 mm surrounding the cancellous bone was
modeled. The height of the premolar crown was 8.5 mm, mesiodistal length (M-D) was 8 mm, buccolingual width (B-L) 7.5 mm, and the height of the root 16 mm. The height of the pontic was 9 mm, B-L width 10 mm, and M-D length 13 mm. The height of the implant abutment crown was 9.5 mm, B-L width 10 mm, and M-D length 13 mm. The periodontal membrane width was accepted as 0.2 mm. A solid 4 x 10 mm screw-type, commercially pure titanium dental implant system Neo Biotic was selected for this study. The simulated partial denture construction from monolithic zirconia with 1 mm thickness on the axial wall and finish line and with 1.5 mm on the occlusal surface was used .

The materials used for the models were presumed to be homogenous, isotropic, and linear, and the osseointegration of the implants was accepted as 100%. In the mathematical model while the implants were directly in contact with the bone, the natural teeth had primary mobility within the borders of the periodontal membrane.

After the geometry amendment was performed, finite element types were selected and material properties' data were entered, running the ANSYS software program was done to solve the problem.

Analysis of the study models was performed to analysis Von-Mises stress values of the three models during load application $\text{SEqv: Von Mises Stress}$. The stresses were measured in the mandible and on the premolar, molar and implant hardware. The stress figure (Fig.1) was indicated as a colored bar drawn on the right side of each figure where the stress values are indicated in Mega Pascal (MPa). The spectrum of colors representing ($\text{SEqv}$) in a descending order was red, orange, yellow, light green, turquoise, light blue and dark blue. Thus, areas with red color represented the highest stress values (maximum tensile and shear stresses) while those with dark blue color represented the lowest stress values (mininal tensile and shear stresses). The mimicked bone that surrounding tooth and implant models were divided into 12 sections to aid in the analysis of the stress mode . The maximal equivalent von Mises stress values in each section were recorded for each model on four planes.

The sections were as follows:

- section 1, mesioalveolar crest of premolar; section 2, mesio middle third of premolar; section 3, mesioapical third of premolar; section 4, distoapical third of premolar; section 5, disto middle third of premolar; section 6, distalvolar crest of premolar; section 7, mesialvolar crest of implant; section 8, mesio middle third of implant; section 9, mesioapical third of implant; section 10, distoapical third of implant; section 11, disto middle third of implant; section 12, distalvolar crest of implant.

The maximum stress in each section along four lines was recorded, added, and evaluated:

- Line 1 – section 1 + section 2 + section 3
- Line 2 – section 4 + section 5 + section 6
- Line 3 – section 7 + section 8 + section 9
- Line 4 – section 10 + section 11 + section 12.

**Result:**

Stress distribution was represented numerically and was color coded. The maximum stress in each zone on the mesial and distal surface of the tooth/teeth and implant/implants in the three models (Table 1). The maximum stress along four lines of the three models represented the amount of Von Mises stresses induced around the tooth/teeth and implant/implants. The results of this study revealed that the implant presence have high influence on the stress concentration in the mandible, while the molar and premolar reflects a low stress concentration in all four lines.

It was noticed that Model I has the highest stress concentration (486 Mpa) as shown in (table 1) then model II (184 Mpa ) and the less stress was noticed in model III (140 Mpa) . In model I the highest stress concentration were located at the neck of implant (distalvolar crest 26.78 Mpa and mesialvolar crest 24.89 Mpa) in line 1&2 and highest stress concentration were located at the neck of implant (mesialvolar crest 24.76 Mpa and distalvolar crest 19.89 Mpa) in line 3&4. While the lowest value of stress concentration in this model was presented at the apical region (4.10 Mpa)&( 3.20 Mpa) in (line 1&2) and (3&4) respectively.

In model II the highest stress concentration were located at mesialvolar crest 6.57 Mpa and distalvolar crest 4.01 Mpa in line 1&2 and at mesialvolar crest 26.13 Mpa and distalvolar crest 25.67 Mpa in line 3&4 respectively .The lowest value of stress presented at apical third of second premolar (1.40 Mpa) in line 1&2 and (1.90 Mpa) in line 3&4.

In model III the highest stress concentration were located at mesialvolar crest 11.63 Mpa (in line 1&2) and at mesialvolar crest 11.63 Mpa (in line 3&4). The lowest value of stress presented at apical third of second premolar (1.02 Mpa) in line 1&2 and (0.07 Mpa) in line 3&4.
Discussion
Finite Element Analysis (FEA) has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material and allowing designers to see all of the theoretical stresses within it. FEA has slowly but steadily found widespread popularity in the fields of medicine and dentistry. Especially in dentistry; where this tool of research methodology has been used to understand the behavior of various materials.12

It has been employed in biomechanical studies and was found to be a reliable technique in simulating bone behavior. It can be used to predict bone fracture and/or failure under increased loads. Because bone is very sensitive to applied loads and responds by remodeling to adapt to the new distribution of strains inside it, this technique can serve as a useful diagnostic tool to provide insight into strain distribution under various loads and designs.13,14

In the field of implant dentistry, the finite element (FE) method has become a progressively useful means for the prediction of stress impact on the implant and its surrounding bone with more accurate computer simulation and modeling technologies.7

The results of this study revealed that, under a static load of 300N, 3D FE presented that implants presence have a huge influence on the stress concentration in the mandible, while the molar and premolar reflects a low stress concentration, this could be interpreted by the presence of periodontal ligaments fibers with their cushioning effect and viscoelastic properties around the natural teeth which were in agreement with Kumar et al14 who found that, the highest stresses was noted around the implant than those of natural tooth in the TIFDP models with the rigid connection.

And in accordance with Koosha and Mirhashemi 15 who observed that, maximum stress values were concentrated at the crestal bone of the implant than that of natural tooth. The high stress concentration around the implants than that of natural teeth may be also illustrated by that: the implant rotational center which exist at the level of alveolar bone is more higher than that of natural tooth, an thus the cortical bone is the stress accumulation area in the implant support as reported by Ozcelik et al11, Sato et al16 and Koosha and Mirhashemi.15 The results of the current study was also in consist with Shamami et al17 who found higher interfacial stresses distribution patterns in the implant especially at oblique loading direction. And in agreement with Guven et al18 who reported a high stresses accumulated in bone tissues in implant-retained models than that of the tooth-retained model. Stress accumulation was observed in the cervical portion of the implant in implant-supported models, and in the surrounding bone of roots in tooth-supported models.

In our results, the highest stress value were noticed in the cortical bone area of implant along the four lines which was in agreement with Dundar et al19 who examined the stress distributions of two implant models under three different static loadings and concluded that, in all models, maximal strains were noticed in the neck region of the implants. And also in consist with Moraes et al20 as they found that the highest stresses was noted around the implant cortical bone region, however more favorable stress distribution was found with wide implant diameter as well as with axial loading direction.

The results of those Wang et al21 were analogues with our study results as they declared that, an evident differences were exist in the high stress region in which strain value was elevated in cortical bone around the implant neck than those of natural teeth. Furthermore, bone density distribution around natural teeth was more uniform and homogeneous. Our results also in accordance with those results of Svennsson et al22 and Koyano et al23 who refer to an inevitable difference that present between the natural teeth and dental implants.

The stress values around the implant and natural tooth were found to be more in the compact bone region and decreased gradually toward the apical region. This could be explained by the differences in the elastic modulus of cortical and spongy bones, in which the cortical bone having a greater modulus of elasticity which is more resistant to deformation and thus will bear more load than those of cancellous bone. As reported by previous studies of Yamanishi et al24, Himmlova et al25, Guven et al18 and Kumar et al14.

Regarding the model iii, greater stress concentration was observed in the mesial and distal sides (line 1 and line 2) of the second premolar tooth than those lines (line 3 and line 4) of second molar tooth which may be interpreted by large surface area of the second molar (two roots) and therefore more PDLs fibers and greater cushioning outcome.

Conclusion
Highest stresses was noted around the implant neck on the cortical bone region than those of natural teeth. Model I has the highest stress concentration then model II and the less stress was noticed in model III.

References


14 Kumar GA, Kovoor L, Oommen V. Three-dimensional finite element analysis of the stress distribution around the implant and tooth in tooth implant-supported fixed prosthesis designs. J Dent Implant 2011; 1: 75–79.


1.