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
1 Associated Professor of fixed prosthodontic Department, Faculty of Dentistry, Mansoura University, Egypt
1 Professor and Head of fixed prosthodontic Department, Faculty of Dentistry, Mansoura University, Egypt
1 Professor of fixed prosthodontic Department, Faculty of Dentistry, Mansoura University, Egypt

Abstract:

Aim: 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 prosthesis.[1] The longevity of the implant prosthetic system depends on many factors one of them is the strains 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.[2,3]

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 prosthesis.[6] 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 resin (Exit 50, Egyptian Swiss For Manufacturing And Trading 6 October Egyp) 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.) and modeled using commercial general purpose CAD/CAM software; “Nx 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 theroot 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 × 10 mm screw-type, commercially pure titanium dental implant system Neo Biotic was selected for this study. The simulated fixed partial denture constriction 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-misses stress values of the three models during load application 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 (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 (minimal 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, distoalveolar crest of premolar; section 7, mesioalveolar 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, distoalveolar 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.

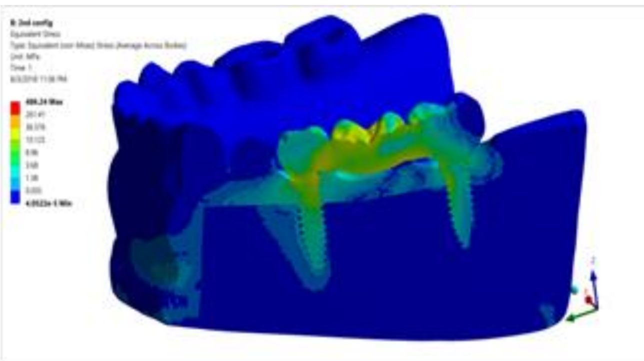

Figure 1: The stress concentration in FE Model 1

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 (distoalveolar crest 26.78 Mpa and mesioalveolar crest 24.89 Mpa) in line 1&2 and highest stress concentration were located at the neck of implant (mesioalveolar crest 24.76 Mpa and distoalveolar 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 mesioalveolar crest 6.57 Mpa and distoalveolar crest 4.01 Mpa in line 1&2 and at mesioalveolar crest 26.13 Mpa and distoalveolar 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 mesioalveolar crest 11.63 Mpa(in line 1&2) and at mesioalveolar 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.

Table 1.: Maximum equivalent von Mises stress in the three models.

Stress / Model	Maximum equivalent von Mises stress values on the whole model
Model I	486 MPa
Model II	184 MPa
Model III	140 MPa

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 wide spread 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 al**[14] 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 al**[11], **Sato et al**[16] and **Koosha and Mirhashemi.**[15]

The results of the current study was also in consist with **Shamami et al**[17] who found higher interfacial stresses distribution patterns in the implant especially at oblique loading direction. And in agreement with **Guven et al** [18]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 al**[19] 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 al** [20] 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 al**[21] 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 **Svensson et al**[22] and **Koyano et al**[23] 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 previoos studies of Yamanishi et al[24], Himmlova et al[25], Guven et al[18]and Kumar et al[14].

Regarding the model iii , greater stress concentration was observed in the mesial and distal sides (line I 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

1 Lang NP, Pjetursson BE, Tan K, Bragger U, Egger M, Zwahlen M. A systematic review of the survival and complication rates of fixed partial dentures (FPDs) after an observation period of at least 5 years. II. Combined tooth-implant-supported FPDs. Clin Oral Implants Res 2004; 15: 643–653.

2 Rubo JH, Capello Souza EA. Finite-Element Analysis of Stress on Dental Implant Prosthesis. Clin Implant Dent Relat Res 2009; 12: 105–113.

3 Grando AF, Rezende CEE, Sousa EAC, Rubo JH. Effect of veneering material on the deformation suffered by implant-supported fixed prosthesis framework. J Appl Oral Sci 2014; 22: 209–17.

4 Sones AD. Complications with osseointegrated implants. J Prosthet Dent 1989; 62: 581–585.

5 Williams KR, Watson CJ, Murphy WM, Scott J, Gregory M, Sinobad D. Finite element analysis of fixed prostheses attached to osseointegrated implants. Quintessence Int 1990; 21: 563–570.

6 van Steenberghe D. A retrospective multicenter evaluation of the survival rate of osseointegrated fixtures supporting fixed partial prostheses in the treatment of partial edentulism. J Prosthet Dent 1989; 61: 217–223.

7 Sivamurthy G, Sundari S. Stress distribution patterns at mini-implant site during retraction and intrusion—a three-dimensional finite element study. Prog Orthod 2016; 17: 1–11.

8 Park J-M, Kim H-J, Park E-J, Kim M-R, Kim S-J. Three dimensional finite element analysis of the stress distribution around the mandibular posterior implant during non-working movement according to the amount of cantilever. J Adv Prosthodont 2014; 6: 361–371.

9 Satoh T, Maeda Y, Komiyama Y. Biomechanical rationale for intentionally inclined implants in the posterior mandible using 3D finite element analysis. Int J Oral Maxillofac Implants 2005; 20: 533–539.

10 Cağlar A, Aydin C, Ozen J, Yilmaz C, Korkmaz T. Effects of mesiodistal inclination of implants on stress distribution in implant-supported fixed prostheses. Int J Oral Maxillofac Implants 2006; 21: 36–44.

11 Özçelik TB, Ersoy AE. An investigation of tooth/implant-supported fixed prosthesis designs with two different stress analysis methods: An in vitro study. J Prosthodont 2007; 16: 107–116.

12 Borcic J, Braut A. Finite Element Analysis in Dental Medicine. http://dx.doi.org/105772/50038 2012; : 1–20.

13 Dundar S, Topkaya T, Solmaz MY et al. Finite element analysis of the stress distributions in peri-implant bone in modified and standard-threaded dental implants. Biotechnol Biotechnol Equip 2016; 30: 127–133.

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.

15 Koosha S, Mirhashemi FS. An Investigation of Three types of Tooth Implant Supported Fixed Prosthesis Designs with 3D Finite Element Analysis. J Dent (Tehran) 2013; 10: 51–63.

16 Sato Y, Teixeira ER, Tsuga K, Shindoi N. The effectiveness of a new algorithm on a three-dimensional finite element model construction of bone trabeculae in implant biomechanics. J Oral Rehabil 1999; 26: 640–3.

17 Shamami DZ, Karimi A, Beigzadeh B, Derakhshan S, Navidbakhsh M. A Three-Dimensional Finite Element Study to Characterize the Influence of Load Direction on Stress Distribution in Bone Around Dental Implant. J Biomater Tissue Eng 2014; 4: 693–699.

18 Guven S, Beydemir K, Dundar S, Eratilla V. Evaluation of stress distributions in peri-implant and periodontal bone tissues in 3- and 5-unit tooth and implant-supported fixed zirconia restorations by finite elements analysis. Eur J Dent 2015; 9: 329–339.

19 Dundar S, Topkaya T, Solmaz MY et al. Finite element analysis of the stress distributions in peri-implant bone in modified and standard-threaded dental implants. Biotechnol Biotechnol Equip 2016; 30: 127–133.

20 Lúcia S, Moraes D De, Verri FR, Santiago JF. Three-Dimensional Finite Element A n a l y s i s o f Va r y i n g D i a m e t e r and Connection Type in Implants with High Crown-Implant Ratio. 2018; 29: 36–42.

21 Wang C, Fu G, Deng F. Difference of natural teeth and implant-supported restoration: A comparison of bone remodeling simulations. J Dent Sci 2015; 10: 190–200.

22 Svensson KG, Trulsson M. Impaired force control during food holding and biting in subjects with tooth- or implant-supported fixed prostheses. J Clin Periodontol 2011; 38: 1137–1146.

23 Koyano K, Esaki D. Occlusion on oral implants: Current clinical guidelines. J Oral Rehabil 2015; 42: 153–161.

24 Yamanishi Y, Yamaguchi S, Imazato S, Nakano T, Yatani H. Influences of implant neck design and implant–abutment joint type on peri-implant bone stress and abutment micromovement: Three-dimensional finite element analysis. Dent Mater 2012; 28: 1126–1133.

25 Himmlová L, Dostálová T, Kácovský A KS. Influence of implant length, diameter, and geometry on stress distribution: A Finite Element Analysis. J Prosthet Dent 2004; 91: 20–25.

1.