Effect of Stiffness of Single Implant Supported Crowns on the Resultant Stresses. A Finite Element Analysis

Sara Ahmed Sayed Ahmed [1], Mohamed A. A. Eldosoky[1], Mohamed Tarek El- Wakad[1], Emad MTM Agamy[2]

[1]Department of Biomedical Engineering, Faculty of Engineering, Helwan University, [2]Department of Prosthetic Dentistry, Faculty of Oral and Dental Medicine, Minia University, Egypt

ABSTRACT

Objective: In the present study, the 3D finite element method was used to investigate the effect of crown material on stress distribution in the bone surrounding immediately loaded single dental.

Materials & Methods: A 3D Finite Element model of mandibular first premolar was constructed to evaluate the performance of seven crown materials with different degree of stiffness (Porcelain, zirconium, Porcelain fused to gold, pure titanium, titanium alloy, Poly methyl methacrylate, and Polyether ether ketone PEEK). The model was constructed using Solid Works version 2010 software. The model simulated also a cement layer between the implant abutment and the crown (Virolink II, Vivadent). An axial static occlusal force of 200 N was applied to eight points in each functional cusp. The three-dimensional (3D) FE model was analyzed by ABAQUS/CAE version 6.10 software.

Results: The results of this study indicated that among all crown materials the maximum von Mises stress values was observed in porcelain crown design (345.390 MPa).The highest von Mises stresses were found in the abutments for all models. In implants, the greatest stress was concentrated on the cervical region. PMMA and PEEK crown designs transferred less stress to abutment and screw. In all models, von Mises stresses increased in the coronal third of cortical bone in which the maximum von Mises stresses observed in the implant – cortical interface.

Conclusions: Using more rigid material for the superstructure of an implant supports prosthesis did not have any effect on the stress values and stress distribution at the bone tissue surrounding implant. However, in the abutment, cement and crown structure, stress distributions and localizations were affected by the material's rigidity. More clinical studies are needed to evaluate the survival rate of these materials.

Keywords: Dental implant, finite element analysis, prosthetic materials, immediate loading.

INTRODUCTION

Dental implants have become a significant aspect of prosthodontic treatment. Despite of high success rates; complications and failures still occur. One factor that is increasingly being implicated with dental implant failure is occlusal overloading as the support of teeth and implants is inherently different. The natural teeth are visco- elastically supported in the bone by the periodontal ligament which acts as a shock absorber between a root and surrounding bone. In contrast to natural teeth; there is no periodontal ligament between dental implants and their surrounding bone. The occlusal loads are transmitted directly to surrounding bone which could cause micro fracture in the interface between bone and implant, fracture of implant.[1, 2] One of the factors that affect load transfer at the bone implant interface, influences stress distribution in dental implants and consequently

affects morphology of the surrounding bone is the type of prosthetic material.

In this regard, however, the results of various in vitro and in vivo studies appear to be somewhat controversial. Skalak proposed that the use of acrylic resin for construction of the prostheses would contribute to dissipate a significant portion of the impact forces during mastication, due to the low stiffness of this material.[3] However, the results of an *in vivo* study by Bassit *et al*. showed that the resilience of an acrylic resin veneer is insufficient to cause significant change in the force transmission through the prosthesis as compared to a ceramic veneer.[4] Also Eskitascıoglu et al investigated the influence of porcelain- and acrylic based material on stress distribution when dynamic forces were applied in vertical and lateral directions on metal-supported crowns over implants. Porcelains were found to absorb and distribute the stress in itself and

consequently cause less transfer of stress to implant and surrounding tissue compared to acrylic-based materials.[5] Ismail et al; analyzed the effect of the occlusal materials (porcelain, precious, and non-precious alloy, acrylic or composite resin) on the stress in bone and implant, and they reported similar results for all the investigated materials.[6]

The classic two-stage protocol of implant placement is associated with longer treatment time, multiple patient visits, and higher treatment expenses. On the other hand the elimination of the healing period offers advantages in terms of cost of treatment and convenience to patients. Thus, immediate loading protocol of dental implants has attracted ever-growing attention in the literature as well as in clinical practice. The main advantage of immediate implant loading is the significantly reduced time interval between implant surgery and prosthetic rehabilitation. The patients do not undergo the emotional and functional stress of being edentulous when they are treated under immediate implant loading protocol. [7-8]

Finite element method (FEM) is expected to be one of the most convincing of computational technique that can be used to evaluate the stress on the implant and its surrounding bone under real situations in the field of biomedical engineering. This technique is based on the premise that an approximate solution to any complex engineering problem can be reached by subdividing the structure/component into smaller more manageable (finite) elements. [9] Therefore three dimensional (3D) finite element analysis (FEA) was selected for use in this study to investigate the effects of prosthesis material type on stress distribution in the bone surrounding immediately loaded single implants.

MATERIALS AND METHODS

In the present study, a three dimensional (3D) finite element model was designed for the mandibular first premolar as shown in Figure1. A section of mandible bone model was constructed based on computed tomography (CT) scanned images of a human mandible in the premolar region, the model represented a15mm segment in mesio-distal direction consisting of a spongy center surrounded by an approximately 1.8mm cortical bone in bucco-lingual direction. A commercially available implant system; Semados implant system, (Ø 4.0 mm x13 mm; Bego,

Bremen, Germany) implant featuring a conical internal hexagonal abutment (Ø4.0 mm x 6 mm) with a connection depth of 2.5 mm and a screw (1.8mm x7mm) were used . [10]The prosthetic crown was developed from natural premolar by using laser camera to teeth then imported to CAD software to convert it to solid form. The thickness of the luting cement was considered to be 50µm. The implant was drawn, assembled, and positioned into the bone section by using Solidworks 2010 software.

For FEA calculations, ABAQUS CAE 6.10 commercial finite element package was used. The entire model was meshed using free meshing technique in which the C3D4 elements tetrahedral elements (4-node-tetrahedron) were used. The total number of nodes was 80024 and total number of elements was 393301. The number of elements and nodes in each part of the model are shown in (Table 1).

Material Properties

In this study, all materials used were assumed as homogeneous, isotropic and linearly elastic.[11-12]The implant was pure titanium while screw and abutment were titanium alloys. Seven different materials were used to simulate the crown as shown in Table 2. Material properties for bone and implant system were taken from the literature.[13-19]

Loading Conditions

A static vertical load (200 N) was applied over cusps of the crown. Loads were separately applied to the functional cusps of the crown in which each functional cusp was divided into 8 areas and each load was exerted on these areas. In other words, the loads were applied to eight points for each functional cusp. [20-21]

Interface Condition

To simulate the interface of an immediately loaded implant, a frictional coefficient (F.C) of 0.3 was applied at the bone–implant interface and at implant- abutment interface. [22-23]Table 3 lists the contact pairs and their respective contact behavior.

Boundary Conditions

Two sides of cortical and cancellous bone surfaces were constrained at the nodes in bucco-lingual direction and mesio-distal direction. At the same time, the nodes on the base of the cortical bone were constrained in all directions as shown in (Figure 2). [24]

RESULTS

The stress levels were calculated using the von Mises stress value, which is an appropriate criterion for stress evaluation of ductile materials.

Stress Distribution in the Main Model

The Von Mises stresses values obtained in main model in case of immediate loaded implant using various materials for artificial crown are shown in (Figure 3). In general the highest stresses were found in abutment and the lowest stresses were found in cortical bone. It was noticed that the stress distribution patterns of occlusal surface were similar in all materials. When the main model was investigated, it was found that the model with Porcelain crown received highest von Mises stress value (345.390MPa) while the model with PEEK crown received the lowest maximum von Mises stress (313.094MPa).The difference between the highest value of von Mises stress of PEEK, and the highest value of von Mises stress of PMMA was negligible (about 313 MPa).

Stress Distribution in the Crown

The Von Mises stresses values obtained in crown structures in case of immediate loaded implant using various materials for artificial crown are shown in (Figure 4, Figure 5).The maximum von Mises stresses obtained in the crown structure are shown in Table4. When the crown structures were investigated, the stress distribution pattern showed that the maximum von- Mises stresses were concentrated at the points of load application on the occlusal surfaces especially at the crown cusps. The highest von Mises stress was obtained in the porcelain crown (345.390 MPa) and the lowest Von Mises stress obtained in PMMA crown (208.355 MPa).

Stress Distribution in the Cement layer

The Von Mises stresses values obtained in cement layer in case of immediate loaded implant using various materials for crown are shown in (Figure6).Changing the superstructure materials affect the stresses values in the cement layer in which the maximum Von Mises stress value was observed in the mesio-occlusal region of the cement in all models and the minimum is observed in the disto- cervical part of the cement layer. The highest maximum Von Mises stress value (308.192MPa) was obtained in PMMA crown model and the lowest maximum Von Mises stress value (33.189 MPa) was obtained in zirconium crown model.

Stress Distribution in the Abutment

The Von Mises stresses values obtained in abutment in case of immediate loaded implant using various materials for artificial crown are shown in (Figure 7). For all crown materials, the maximum von Mises stress did not reach the yield strength of titanium alloy which equal to 800 MPa. The highest maximum Von Mises stress value (313.453MPa) was obtained at porcelain fused to gold model and the lowest maximum von Mises stress was obtained at PEEK model (313.094 MPa). The difference between the maximum value of von Mises stress of PEEK, and the maximum value of von Mises stress PMMA was negligible. Changing the superstructure materials affects the stress distribution in the abutment in which the maximum Von Mises stress value was observed in the distal surface at occlusal third of the abutment in cases of PMMA and PEEK crown. However it was observed at the cervical region of the abutment at implant- abutment junction in the remaining crown materials. While the minimum Von Mises stress values was observed in the mesial surface of the abutment in all crown materials.

Stress Distribution in the Abutment Screw

The Von Mises stresses values obtained in a screw in case of immediate loaded implant using various materials for artificial crown are shown in (Figure8).For all crown designs, the maximum von Mises stress did not reach the yield strength of titanium alloy (TI 4V AL) which equal to 800 MPa. The highest maximum Von Mises stress value (31.009MPa) was obtained at PMMA model and the lowest maximum von Mises stress was obtained at zirconium model (30.427 MPa). The high values of von Mises stresses were observed at the upper one third of the screw and the middle of screw neck. These regions represent the connection section where the screw inserts deep into the abutment. While the low values of von Mises stresses were located at the lower one-third of the screw.

Stress Distribution in the Implant

The Von Mises stresses values obtained in an implant in case of immediate loaded implant using various materials for artificial crown are shown in (Figure 9). Changing the crown material did not affect the stress distribution pattern on implant in which the stress in implants for all crown materials was localized in the distal surface of the implant's cervical third and then

distributed toward the remaining of the implant's body. For all models, an inconsiderable difference in the stresses values of implant was observed; in which the maximum Von Mises stress value (98.210MPa) was obtained in PMMA model and the lowest maximum von Mises stress (98.117MPa) was obtained in zirconium model. The maximum Von Mises Values were observed in the implant and abutment junction region and then the stress values decreased toward the apical region of the implant in which the minimum stress value was observed.

Stress Distribution in the supporting Bone

The Von Mises stresses values obtained in the Surrounding bone in case of immediate loaded implant using various materials for artificial crown are shown in (Figure10.) The maximum von Mises stress of cancellous bone was (4.553MPa) for all crown materials, with no significant difference observed maximum von Mises stress in cortical bone (about 20.9MPa) between all crown design. In all models, von Mises stresses increased in the coronal third of cortical bone in which the maximum Von Mises stresses observed in the implant – cortical interface. In addition; the maximum von Mises stresses in cancellous bone were observed in the implant – cancellous interface and in cortical – cancellous interface.

DISCUSSION

One of the basic problems in biomechanical engineering is formulation of implant mechanical characteristics in a way that ensures, both structure durability and an optimal load patterns in surrounding tissues. Since it is postulated that the biomechanics of the implants would be improved if a mobility similar to the one allowed by the periodontal ligament was incorporated. In implant supported fixed partial dentures the stresses occur as a result of occlusal forces transmitted to the supporting bone by restorative material, abutment, and the implant. The stresses must be at physiological levels, and extreme stress concentrations should be eliminated. For this reason the stresses in materials and supporting tissues must be analyzed.

Alternatives to reduce the forces transmitted to implants have been studied, including variations in implant positioning, implant design, prosthesis shape, occlusal requirements, prosthetic components and prosthetic materials. In the present study, a 3D finite-element stress analysis method was used to evaluate the stresses generated in the dental implant system components (implant, abutment, screw, cement, crown), and supporting bone in case of immediate loaded implant with various degrees of stiffness of crown material under functional forces.

In FEA, the mechanical performance of the interface between dental implant system components could be evaluated by von Mises stresses. Principal stresses are used to evaluate the stresses induced around the implants in the bone—a typical brittle material. However, in literature the von Mises stress and maximum principle stress [25-29] are used as a valuable measure of all stresses which are generating in the bone- implant interface. In the present comparative study the von Mises stress was used as an EQV (equivalent value). It has been extensively documented in the literature and well accepted to use von Mises as an EQV. Moreover, von Mises is more conservative and not mainly care about the direction of the equivalent values as in the maximum principle stress. In the comparative studies of dental implants and especially in the elastic response without considering the failure, it is more accurate and suitable for using von Mises stress as an EQV of all normal and shear stresses generated in the bone. [24]

The model used in this study implied several assumptions regarding the simulated structures. The structures in the model were all assumed to be homogeneous and isotropic and to possess linear elasticity. The properties of the materials modeled in this study, particularly the living tissues, however, are different. For instance, it is well described that the actual cortical bone of the mandible is transversely isotropic and inhomogeneous.[30] Additionally, perfect bond interface was established between some contact surfaces in the model; which does not necessarily simulate clinical situations. Also, it is important to point out that the stress distribution patterns would have been different, depending on the materials and properties assigned to each layer of the model used in the experiments. Thus, the inherent limitations in this study should be considered.

Hojjatie and Anusavice also accepted all materials as linear elastic, homogeneous, and

isotropic, and ignored cement thickness in their finite-element stress analysis study. In the current study cement thickness was considered to maintain the reality of the FEA and investigate its effect on the stress distribution. [31]

The same occlusal morphology was used to evaluate the effect of various materials on stresses transferred to supporting bone, implant, and abutment for all models. In the current study, the locations for the force applications were specifically described as cusps tips in which distributed vertical loading of 200 N was used. However, the geometric form of the tooth surface can produce a pattern of stress distribution that is specific for the modeled form. The pattern could be different with even moderate changes to the occlusal surface of the crown. The occlusal form chosen for this model does not mean that the same form would represent all premolar teeth.

There are several studies that investigated the effects of different occlusal materials on implants. Papavasiliou *et al.* observed no differences between acrylic-resin–veneered gold and porcelain fused to metal as occlusal materials. [32] Bassit et al demonstrated that using different occlusal surface materials does not produce different stresses in implants. [33]Cibrika *et al.* did not observe a significant statistical difference when they used resin, gold, and ceramic as occlusal surfaces. [34]However, in the current study different occlusal materials generated approximately similar stresses in implants but differences in stress related to crown material. The reason for these discrepancies may be the result of differences between materials used in the current study and the other studies.

Gomes *et al.* evaluated the effect of different material combinations ((IPS Empress 2, In-Ceram, PFBM, PFNM) on stress distribution within metal-ceramic and all-ceramic single implant-supported prostheses by three-dimensional finite element analysis, they concluded that the use of different materials to fabricate a superstructure for a single implant-supported prosthesis did not affect the stress distribution in the supporting bone. [35] Moreover, Sevimay *et al.* investigated the effect of different occlusal surface materials on stress generation under functional forces. When using vertical loading at two locations, they concluded that using more rigid or resilient material for the superstructure of an implant-supported prosthesis

did not have any effect on stress distribution and stress values at the bone tissue surrounding implant.[36] However, in the abutment and crown structure, stress distributions and localizations were affected by the material's rigidity. These results are in agreement with the findings of the current study in which no differences were found among various materials; when the stress distribution in supporting bone was investigated,.

It is important to highlight that highest stress value was obtained in abutment and the lowest was obtained in the cortical bone. Also the buccal curvature of the cortical bone showed a more uniform stress distribution than the lingual one.

Papavasiliou *et al.* investigated the effect of the osseointegration degree to stress distribution and found higher crestal stresses than apical stresses under all conditions. [37]In the current study, the stresses were concentrated in the neck of the implant due to the rigid connection between the implant and the bone. The elastic modulus of cortical bone is higher than spongy bone; for this reason, cortical bone is stronger and more resistant to deformation.

When the stress distribution in artificial crown was investigated, porcelain crown showed the highest stress concentration. The high stress values were obtained in ceramic and metal crown materials and the low stress values in polymer crown materials. The reason of these differences may be that the modulus of elasticity of porcelain which made it more resistant to deformation.

A consistent observation from all models was concentration of maximum stresses at the porcelain surface at the loading points. For this reason, interceptive occlusal contact in the crown should be eliminated and proper occlusal relationship should be provided. The materials selected for the occlusal surface of the implant-supported prosthesis may affect the transmission of forces and the maintenance of occlusal contacts

In the light of the results of the present study, selection should be customized for the individual case for optimum esthetics and performance. Finite element models have limitations because the mechanical properties and the nonlinear behavior of biological tissues cannot precisely be imitated. More clinical trials are necessary to confirm further the findings of the present study.

CONCLUSION

A three-dimensional finite-element analysis model was constructed to investigate the effect of different occlusal surface materials on stress generation under functional forces, within the limits of this study; the following conclusions can be drawn:

1- Porcelain crown induced higher value of Von Mises stress while PMMA crown induced lower value of Von Mises stress

2- Using more rigid material for the superstructure of implant supported prosthesis did not have any effect on the stress values and stress distribution at the bone tissue surrounding implant. However, in the abutment, cement and crown structure, stress distributions and localizations were affected by the material's rigidity.

3- The stress values recorded in the artificial crown in most of cases decreased with the diminution of the stiffness of its material.

4- Using resilient materials for superstructure increased the stresses within prosthetic screw.

REFERENCES

1- Lídia Carvalho , António Ramos , José A. Simões(2003): "Finite Element Analysis Of A Dental Implant System With An Elastomeric Stress Barrier", Summer Bioengineering Conference,Florida,0733-2

2- Roberto Adrian Markarian, Raul Gonzalez Lima(2005): "The Influence of the Prosthetic Materials Stiffness On Load Transfer To Dental Implants", Technology Meets Surgery International ,São Paulo.

3-SkalakR(1983): ", Biomechanical considerations in osseointegrated prostheses." J. Prosthet Dent., 49:843–849.

4- Bassit R, Lindsro¨m H, Rangerty B (2002): "In-vivo registration of force development with ceramic and acrylic resin occlusal materials on implant-supported prosthesis", Int J.Oral Maxillofac Implants,55:34-38.

5- Eskitascıoglu G, Baran I ́, Aykac Y, Oztas D (1996) : " Investigation of the effect of different esthetic materials in implant-crown design", Turkish J. Oral Imp ,35:23-25.

6-Ismail YH, Kukunas S, Pipko D, Ibiary W (1989): "Comparative study of various occlusal materials for implant prosthodontics [abstract] ". J. Dent Res.,1-2.

7-Misch CE(1998): "Non-functional immediate teeth in partially edentulous patients", A pilot study of 10 consecutive cases using the Maestro dental implant system. "Compend Contin Educ Dent.pp,15.

8-Shrikar R Desai, Rika Singh, I Karthikeyan, G Reetika, Jyothilaxmi (2012): "Three-dimensional finite element analysis of effect of prosthetic materials and short implant biomechanics on D4 bone under immediate loading", Journal of Dental Implants , 2 (1):2-12.

9- Arularasan R, Hemanandhan T,Thamizhselvan B,Arunkumar S,Senthilnathan S,Prathap S (2016): "MODELING AND SIMULATION OF ENGINE CYLINDER FINS BY USING FEA", International Journal of Advanced Research, 4(1): 1391- 1396.

10-Chun-Bo Tang, Si-Yu Liu1 Guo-Xing Zhou, Jin-Hua Yu Guang-Dong Zhang1, Yi-Dong Bao and Qiu-Ju Wang(2012): "Nonlinear finite element analysis of three implant–abutment interface designs", International Journal of Oral Science 4: 101–108.

11-Ahmet Kursad Culhaoglu, Serhat Emre Ozkir, Gozde Celik, Hakan Terzioglu (2013): " Comparison of two different restoration materials and two different implant designs of implant‑supported fixed cantilevered Prostheses: A 3D finite element analysis", European Journal of General Dentistry , 2(2):11-22.

12- Hojjatie B, Anusavice KS (1990): "Three dimensional finite elementanalyses of glass ceramic dental crowns", J Biomechanics,23:1157–1166.

13-Wang TM, Leu LJ, Wang J, Lin LD (2002):" Assessment of stress distribution around implant fixture with three different crown materials". Int J Oral Maxillofac Implants, 17:231‑7.

14-Ciftçi Y, Canay S (2001) "Stress distribution on the metal framework of the implant‑supported fixed prosthesis using different veneering materials", Int J .Prosthodont,14:406‑11.

15-Himmlová L, Dostálová T, Kácovský A, Konvicková S (2004): "Influence of implant length and diameter on stress distribution: A finite element analysis",J Prosthet Dent.,91:20‑5.

16- Misch CE (1999): Contemporary Implant Dentistry. Missouri: Mosby p. 151‑61.

17- Akça K, Iplikçioğlu H (2001): " Finite element stress analysis of the influenceof staggered versus straight placement of dental implants",Int J .Oral Maxillofac Implants, 16:722‑30.

18- Barbier L, Vander Sloten J, Krzesinski G, Schepers E, Van der Perre G(1998): "Finite element analysis of non‑axial versus axial loading of oral implants in the mandible of the dog". J. Oral Rehabil., 25:847‑58.

19-Mericske‑Stern R, Assal P, Mericske E, Bürgin W(1995): "Occlusal forceand oral tactile sensibility measured in partially edentulous patientswith ITI implant". Int J. Oral Maxillofac Implants, 10:345‑53.

20- Ghasemi L, Khazaei S, Bajgholi F, Nasri N, Iranmanesh P (2014):"Stress Distribution in Luting Cement Layer in Implant Supported Fixed Partial Dentures Using Finite Element Analysis", Journal of Islamic Dental Association of IRAN (JIDAI) , 26(2):23-25.

21-Pedram Iranmanesh , Alireza Abedian, Naeimeh Nasri1, Ehsan Ghasemi1, Saber Khazaei (2014) : "Stress analysis of different prosthesis materials in implant-supported fixed dental prosthesis using 3D finite element method", Dental Hypotheses , 5(3):44-47.

22-Schwitalla A, Abou-Emara M, Spintig T, Lackmann M, Müller N (2015): "Finite element analysis of the biomechanical effects of PEEK dental implants on the peri-implant bone'', Journal of Biomechanics, 55:481–7.

23- Syafiqah Saidin Mohammed Rafiq Abdul Kadir „ Eshamsul Sulaiman ,Noor Hayaty Abu Kasim (2012):"Effects of different implant–abutment connections on micromotion and stress distribution: Prediction of microgap formation", journal of densitry , 2 0 1 ; 4 6 7 – 4 7 4.

24- Mohammed Moustafa Mahmoud Mohammed Hassan(2015) :" Optimization Of The Stresses Developed In Bone Surrounding A Dental Implant Using Finite Element Method'', Thesis Submitted in Partial Fulfillment of The Requirements for the Degree of PHD of Science In Production Engineering OPTIMIZATION, Helwan university.

25- Li T, Kong L, Wang Y, Hu K, Song L, Liu B, Li D, Shao J, Ding Y (2009): " Selection of optimal dental implant diameter and length in type IV bone: a three-dimensional finite element analysis", Int. J. Oral Maxillofac ,33:23-26.

26-Merdji A., B. B. (2010):"Stress analysis in dental prosthesis", Computational Materials Science 49, 126 - 133.

27-Xi Ding X (2009): "Implant–Bone Interface Stress Distribution in Immediately Loaded Implants of Different Diameters: A Three-Dimensional Finite Element Analysis", Journal of Prosthodontics ,18:393 - 402.

28- Hong Guan R (2011):"Dynamic modelling and simulation of dental implant insertion process - A finite element study" Finite Elements in Analysis and Design, 47: 886–897.

29-Oguz Kayabas E (2006). Static, dynamic and fatigue behaviors of dental implant using finite element method", Advances in Engineering Software, 37:649–658

30- Ashman RB, Van Buskirk WC (1987): "The elastic properties of a human mandible", Adv Dent Res., 1:64–67.

31-Hojjatie B, Anusavice KS (1990): " Three dimensional finite element analyses of glass ceramic dental crowns", J. Biomechanics ,23:1157–1166.

32- Papavasiliou G, Kamposiora P, Bayne SC, Felton DA(1996): " Three dimensional finite element analysis of stress distribution around single tooth implants as a function of bony support, prosthesis type and loading during function", J. Prosthet Dent .,76:633– 640.

33- Bassit R, Lindsro¨m H, Rangerty B (2002):" In-vivo registration of force development with ceramic and acrylic resin occlusal materials on implant-supported prosthesis", Int J Oral Maxillofac Implants, 17:17–23.

34-Cibrika RM, Razzoog ME, Lang BR, Stohler CS(1992): "Determining the force absorption quotient for restorative materials used inimplant occlusal surfaces", J. Prosthet Dent .,67:361–364.

35- Gomes ÉA, Barão VA, Rocha EP, de Almeida ÉO, Assunção WG(2011): "Effect of metal-ceramic or all-ceramic superstructure materials on stress distribution in a single implant supported prosthesis: three- dimensional finite element analysis", Int J Oral Maxillofac Implants,26:1202-9.

36- Sevimay, Usumez, and Eskitascioglu (2005): "The Influence of Various Occlusal Materials on Stresses Transferred to Implant-Supported Prostheses and Supporting Bone: A Three-Dimensional Finite-Element Study", J. Biomed Mater Res. Part B: Appl Biomater, 73B:140–7.

37-Papavasiliou G, Kamposiora P, Bayne SC, Felton DA(1997): "3D FEA of osseointegration percentages and patterns on implant bone interfacial stresses", J. Dent .,25:485– 491.

List of Tables

Table1:Number of the nodes and elements for all parts of the model		
Part	**Nodes**	**Elements**
Cortical	10001	46002
Cancellous	12749	64128
Implant	15576	72793
Abutment	12139	61593
Screw	1347	5488
Cement	2720	8054
Crown	25492	135243

Table 2: Material properties adopted in the study		
Material	**Elastic modulus (GPa)**	**Poisson 's ratio**
cortical bone	13.7	0.3
cancellous bone	1.37	0.3
Pure Titanium	117	0.3
Titanium alloy (Ti 4V AL)	110	0.3
Cement ; virolink II	8.3	0.35
PMMA	2.38	0.45
PEEK	4.1	0.4
Zirconium	200	0.3
Porcelain	70	0.19
Porcelain fused to gold	86.2	0.33

Table 3:Contact pair definitions		
Contact Pairs		**Type of contact**
Cortical	Cancellous	Perfect bond
Implant	Bone	F .C=0.3
Implant	Screw	Perfect bonded
Abutment	Screw	Perfect bonded
Implant	Abutment	F.C =0.3
Abutment	Cement	Perfect bonded
Crown	Abutment	Perfect bonded
Cement	Crown	Perfect bonded

List of Figures
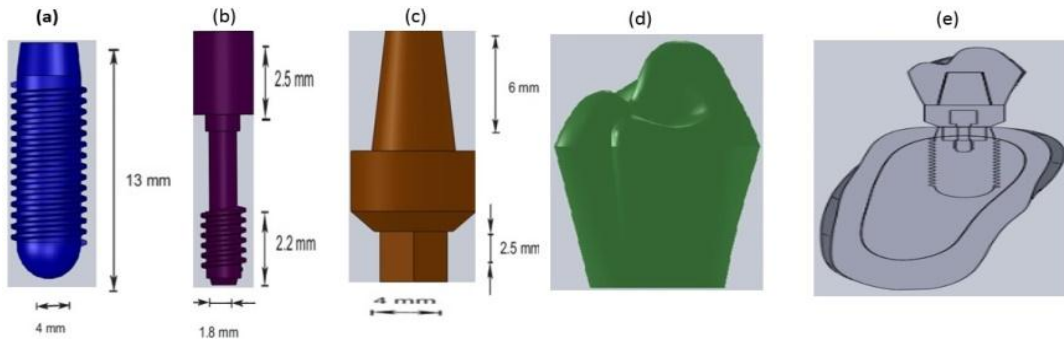
Fig.1 3D solid model of: (a) implant, (b) screw, (c) abutment, (d) crown, (e) section view of main model.
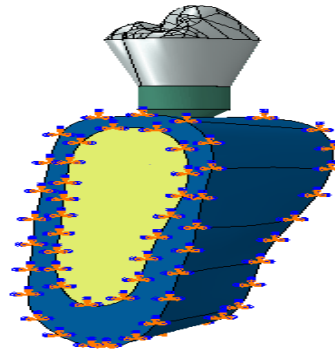
Fig.2 Boundary condition of the finite element model
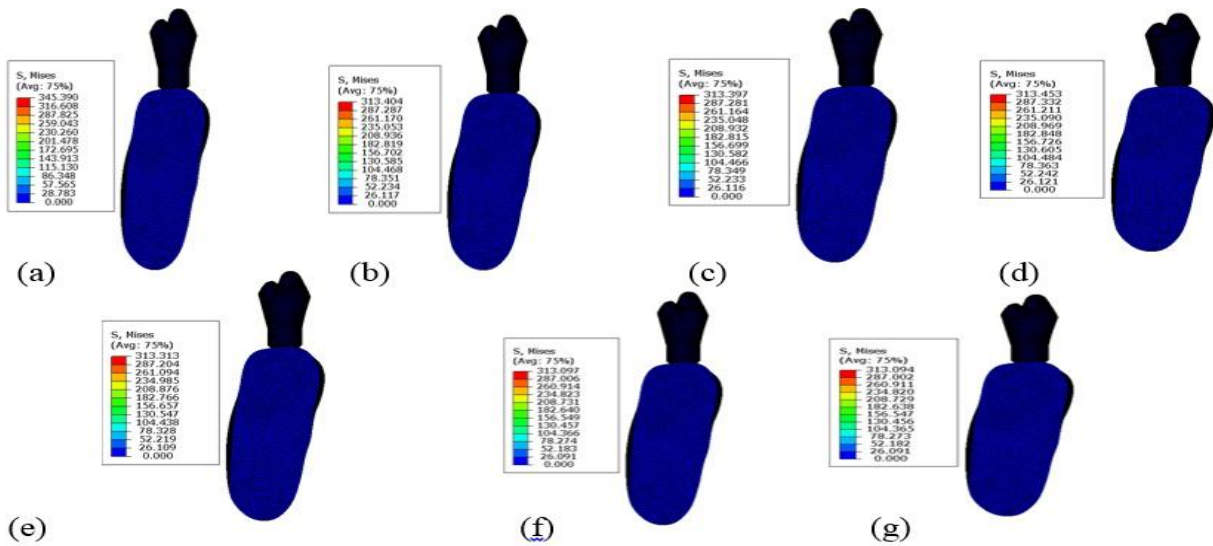
Fig .3 Von Mises stresses in main model (a) Porcelain, (b) Porcelain fused to gold, (c) Ti alloy,
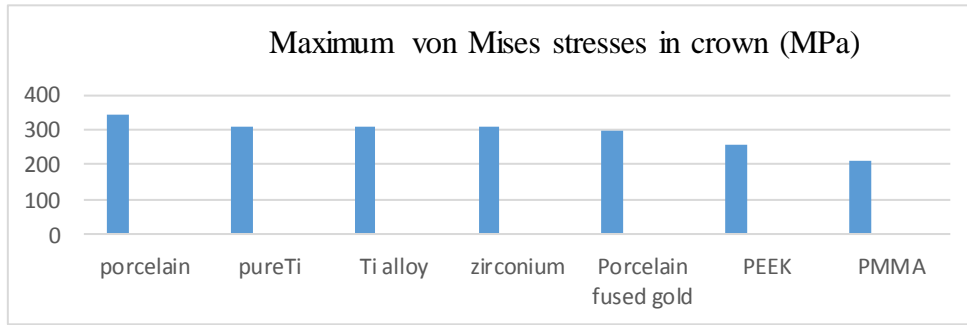
(d) Pure Ti, (e) Zirconium, (f) PMMA, (g) PEEK

Fig.4 Graph of maximum von Mises stresses in crown
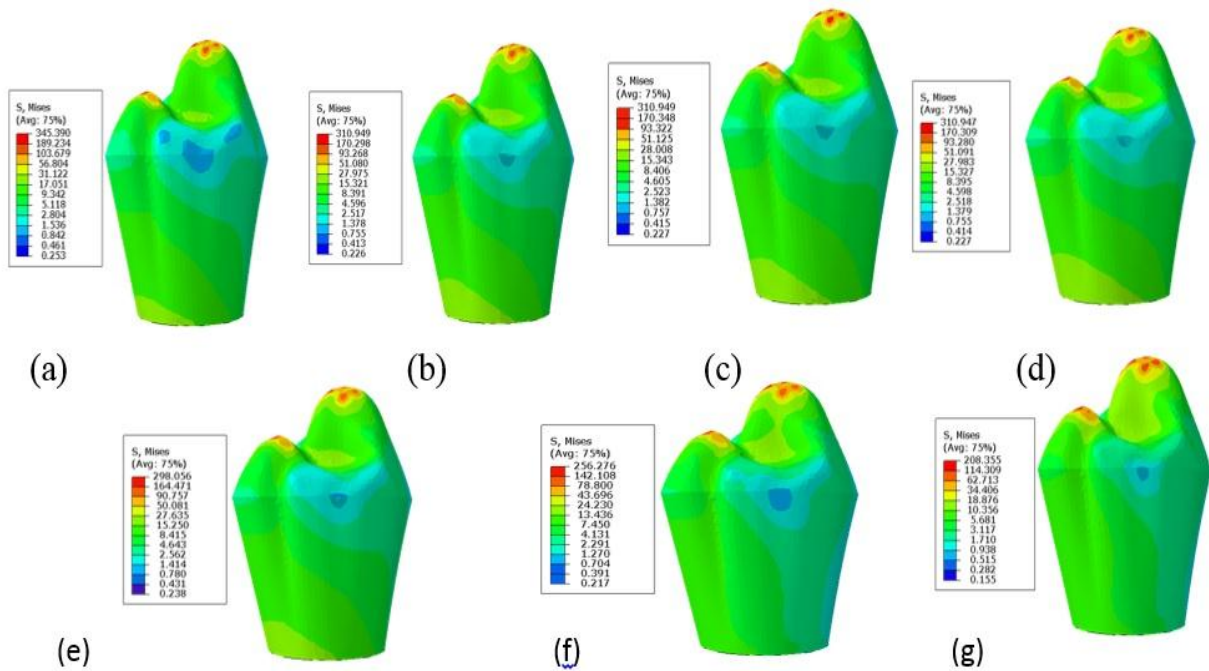
Fig.5 Von Mises stresses in crown (a) Porcelain, (b) Pure Ti, (c) Ti alloy, (d) Zirconium, (e) Porcelain fused to gold, (f) PEEK, (g) PMMA
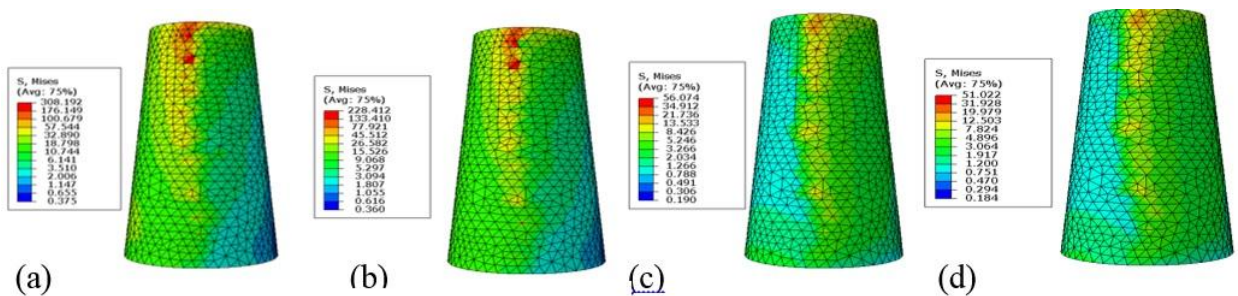
Fig.6 Von Mises stresses in cement, (a) PMMA, (b) PEEK, (c) Porcelain, (d) Porcelain fused to gold, (e) Ti alloy ,(f)Pure Ti ,(g) Zirconium

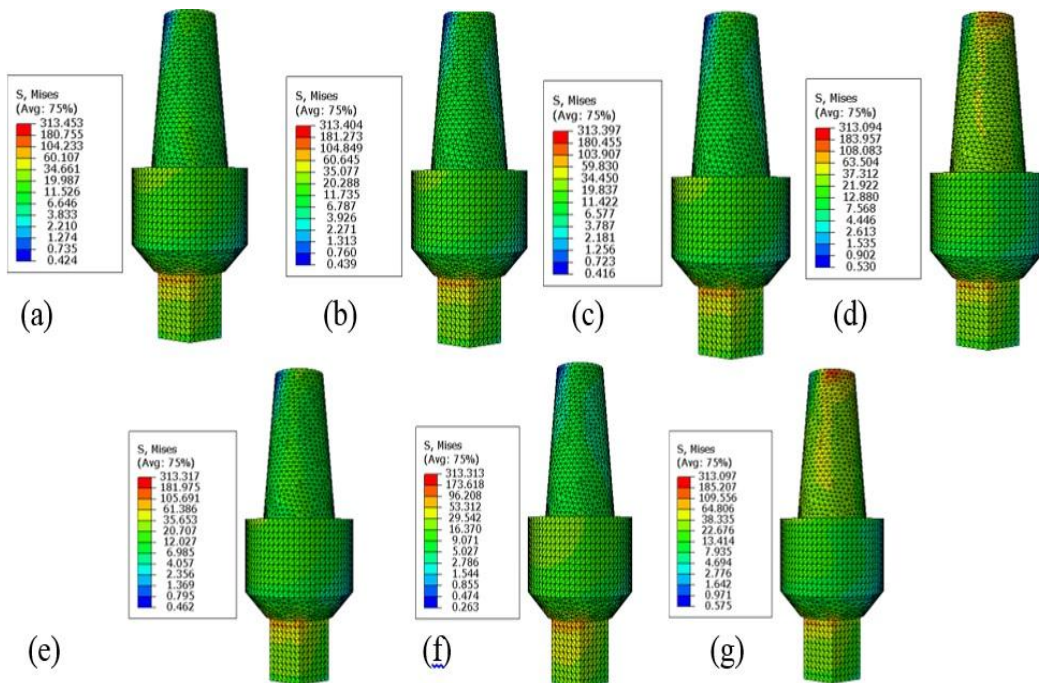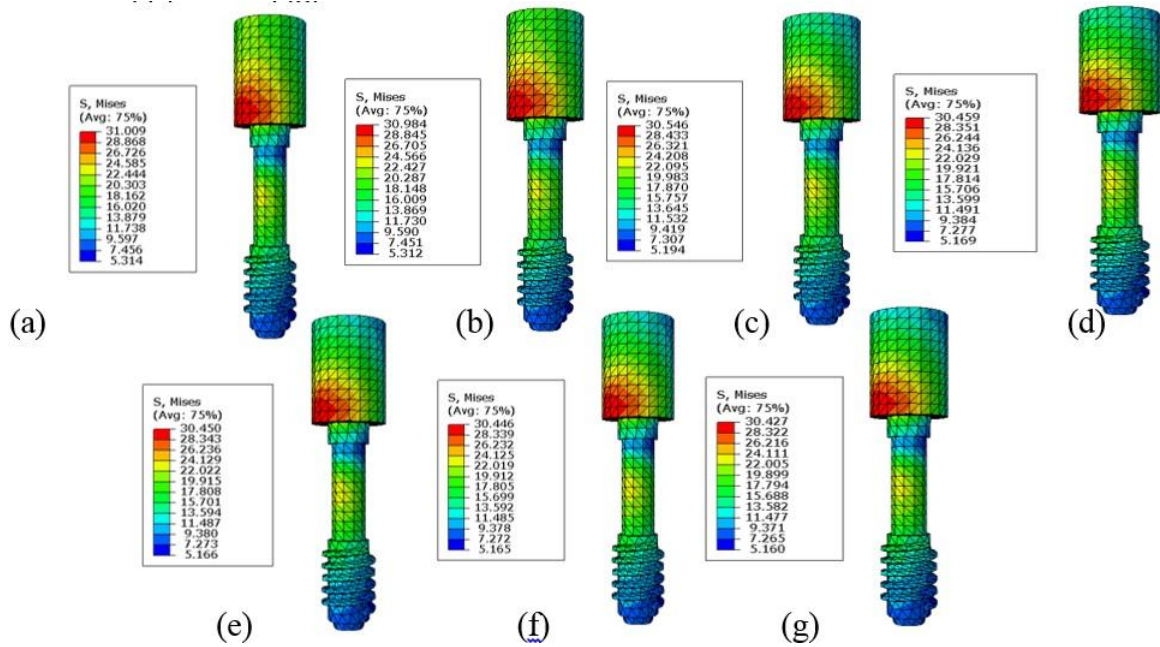Fig.7 Von Mises stresses in abutment, (a) Porcelain fused gold, (b) Ti alloy, (c) Pure Ti, (d) Porcelain, (e) Zirconium, (f) PMMA, (g) PEEK

Fig.8
Von

Mises stresses in screw, a) PMMA, (b) PEEK, (c) Porcelain, (d) Porcelain fused to gold,(e) Ti alloy , (f)Pure Ti , (g) Zirconium
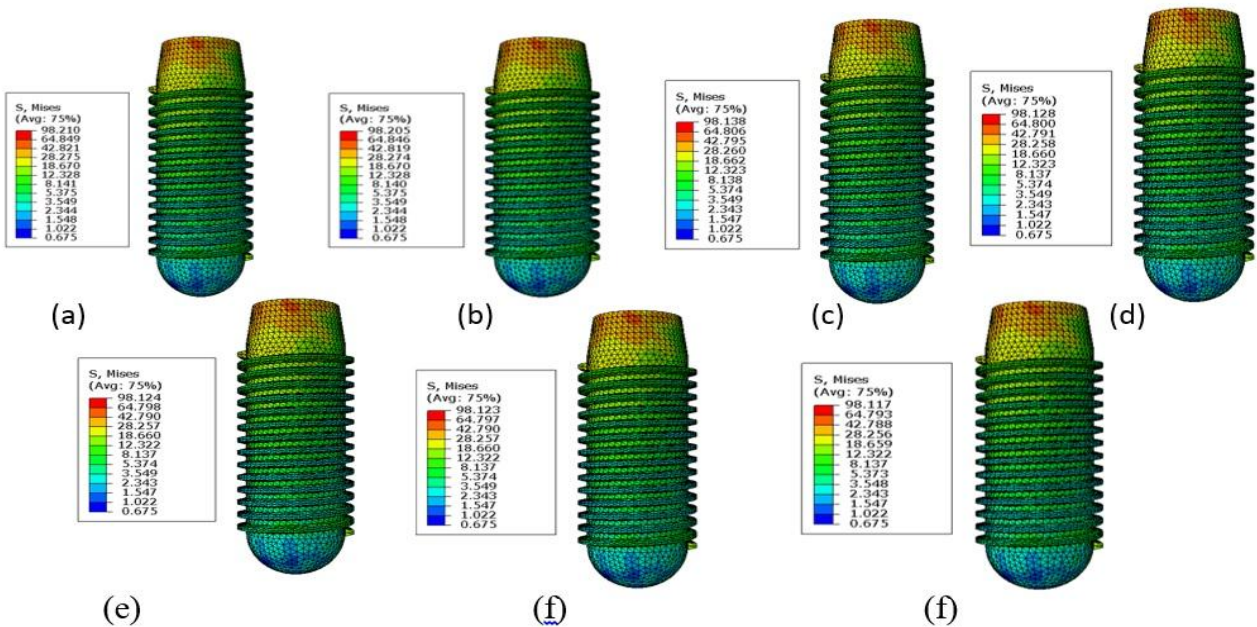
Fig.9 Von Mises stresses in Implant, (a) PMMA, (b) PEEK, (c) Porcelain, (d) Porcelain fused to gold,(e) Ti alloy ,(f)Pure Ti , (g) Zirconium
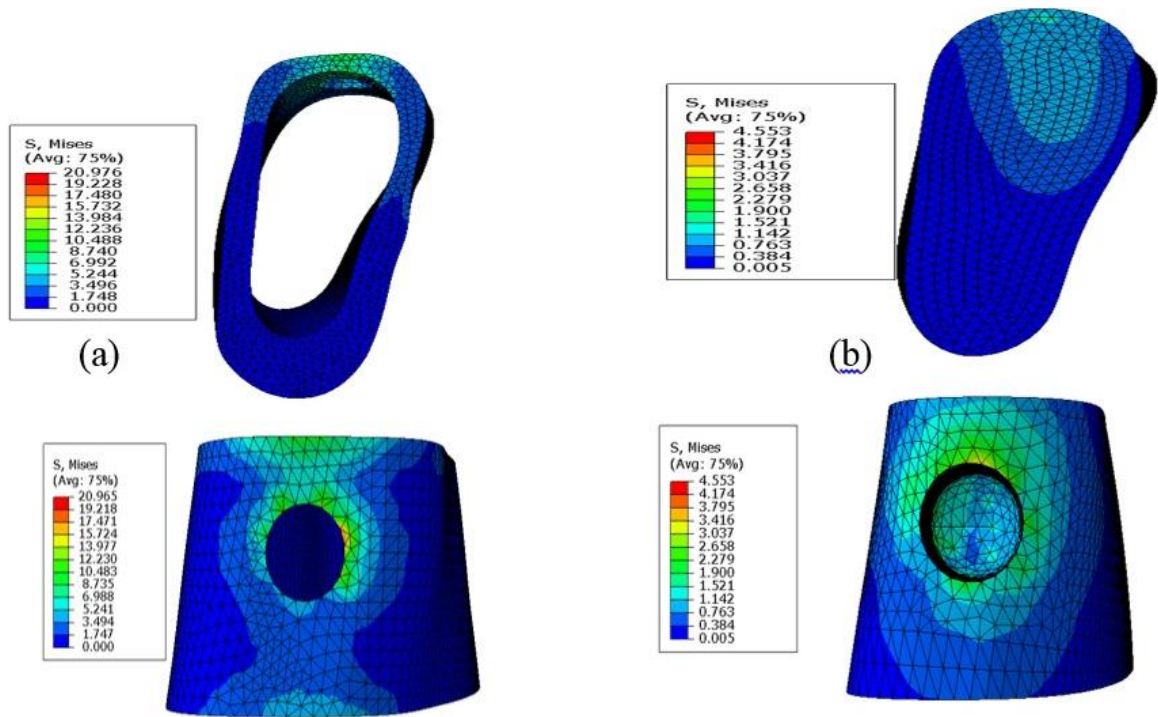
Fig. 10 The von Mises stresses in surrounding bone, (a) Front view of cortical bone, (b) Front view of cancellous bone, (c) Occlusal view of cortical bone, (d) Occlusal view of cancellous bone