Design and Analysis of a New Dental Implant using Finite Element Method

S. Rahmati
Department of Mechanical Engineering,
Islamic Azad University, Majlesi Branch, Isfahan, Iran
E-mail: rahmati@rapidtoolpart.com

H. Kheirollahi*
Department of Mechanicaland Manufacturing Engineering,
University of Manitoba, Winnipeg, Canada
E-mail: kheirolh@myumanitoba.ca
*Corresponding author

A. Azari
Department of Prosthodontics, Faculty of Dentistry,
Tehran University of Medical Sciences, Tehran, Iran
E-mail: azari@sina.tums.ac.ir

Abstract: An overall treatment with dental implant consist of several essential parts, including a superstructure containing fixture (part of the implant embedding in the bone), abutment (a machined tapered prepared superstructure attached to the fixture), a retainer (which hold the aesthetic portion of the restoration) and the crown (aesthetic part of the restoration). In this common kind of restoration, crown is cemented to metal framework. However, in this research a novel method of design of dental implant is presented which includes only two components: Integrated implant, and Crown. The new design is based on an integrated implant which enables dentist to fit the crown by a snap-fit ringinstantly. In order to carry out stress analysis, the finite element method (FEM) is applied on implant and bones to verify different loading conditions. The results of FEM analysis indicate thatthe proposed design undergoes different loading conditions successfully.

Biographical notes: S. Rahmati is Associate Professor of Additive Manufacturing Technology at mechanical engineering department of Islamic Azad University, Majlesi Branch in Iran. He received his PhD from Nottingham University (UK) on rapid tooling (1999), his MSc from Loughborough University (UK) on CIM (1994), and BSc from Ottawa University (Canada) in 1985. His research interest includes additive manufacturing, rapid tooling, medical additive manufacturing, CAD/CAM, CIM, and advanced manufacturing processes. **H. Kheirollahi** is PhD student at department of mechanical and manufacturing engineering, University of Manitoba, Winnipeg, Canada. His research interests include Finite Element Method and Computational Biomechanics. **A. Azari** is Professor at Department of Prosthodontics, Faculty of Dentistry, Tehran University of Medical Sciences, Tehran, Iran. His research interests include Computer Assisted Implantology, Computer Guided Implantology and Computer Assisted Surgery.

1 INTRODUCTION

Since dental implants were introduced for rehabilitation of the completely edentulous patients in the late 1960s, an awareness and subsequent demand for this form of therapy has increased significantly [1].The use of implants have revolutionized dental treatment modalities and provided excellent long term results [2]. In evaluation of the long-term success of a dental implant, the reliability and the stability of the implant and bone play a great role. In general, the success of the treatment depends on many factors affecting the bone and implant [3]. Prosthetic components are subjected to a complex pattern of horizontal and vertical force combinations [4]. Analyzing force distribution at the bone–implant interface is an essential step in the overall analysis of loading, which determines the success or failure of an implant. The finite element analysis (FEA) allows researchers to predict stress distribution in the contact area of implants with compact and spongy bones.

With the increasing demand and clinical applications of dental implants, more and more implant–abutment interface related designs and performance issues have been investigated and reported [5], [6]. Currently, there are over 20 different geometric variations of implant–abutment interface commercially available [7]. However, long-term follow-up studies on implants indicate that many complications occur after the prosthetic phase [5]. These complications include loss of osseointegration [8], abutment screw loosening [9], abutment screw fracture [10], and similar issues [11], [12]. Simon et al. determined the success rate of implants restored as single molar and premolar crowns, and reported that dental implant failure rate to be 4.6%, of which 7% was related to abutment screw loosening [8]. Binonet al. observed a direct correlation between implant/abutment hexagonal rotational misfit and screw loosening [13].

An overall treatment with dental implant consist of several essential parts, including a superstructure containing fixture (part of the implant embedding in the bone), abutment (a machined tapered prepared superstructure attached to the fixture), a retainer (which hold the aesthetic portion of the restoration) and the crown (aesthetic part of the restoration) as shown in Fig. 1. In this common kind of restoration, retainer is cemented to abutment, where in this mechanism, the components that have been cemented to each other may be disengaged and also the fabrication process is time consuming due to the sum of components which must be assembled [9]. Hence due to the abovementioned shortcomings, a new mechanism in dental implant restorations is proposed in order to resolve these draw backs. The proposed dental implant mechanism consists of two major components: implant and crown, where the crown easily fits to the implant by a snap-fit ring mechanism. In this new dental implant mechanism abutment and metal framework are avoided hence loosening of cemented components are not expected. Hence, in this mechanism due to reduction of parts assembled to each other, time of remedy reduces as after embedding implant to the patient bone, crown fits to implant by a snap-fit ring instantly. So the novel mechanism of dental implant helps in reduction of remedy time, increasing implant embedding quality and accuracy.
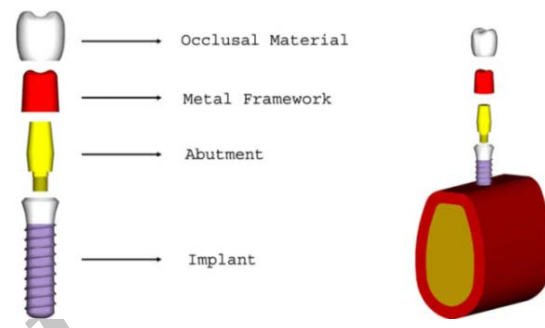

Fig. 1 3D solid model of implant, abutment, metal framework, occlusal material, bone and gingiva, respectively [16]

In this new dental implant mechanism, abutment and metal framework are excluded and hence loose-fitting of cemented parts are not expected to occur. Hence, due to the reduction of parts that are being assembled, operation time is reduced significantly, leading to quality and accuracy improvement of embedded implant. In this research, several scenarios are being investigated in order to succeed the final novel design of the new dental implant mechanism. Upon achievement of the proposed design, FEA method is applied in order to analyses it under different loading conditions. Hence in this research, three-dimensional (3-D) finite element analysis (FEA) is utilized extensively for quantitative evaluation of stress on the implant and its surrounding bone [5], [7], [14-16]. In other words, FEA is selected in this study to examine the effect of the static and dynamic combination of loading on the stress distribution for an implant-supported fixed partial denture and supporting bone tissue of the new dental implant complex.

2 DESIGN PROCESS OF NEW DENTAL IMPLANT MECHANISM

Basically in design of a new mechanism, all possible challenges that may influence its performance must be taken into consideration. In this research, for designing

of new dental implant mechanism, several scenarios have been investigated and finally an optimum design has been proposed.
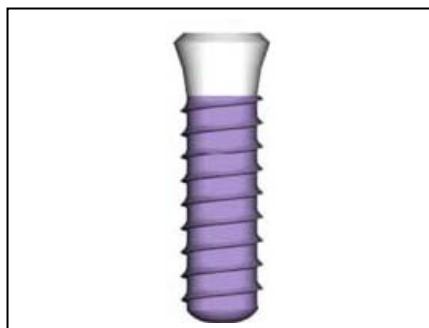

Fig. 2   Standard dental implant

For the proposed implant design, different changes have been made in the standard implant in such a way that the task of abutment and metal framework are integrated at the new dental implant mechanism (Fig. 2).

Fig. 3   New dental implant

At the top section of new implant a circular groove is generated to hold the snap-fit ring firmly (Fig. 3). The snap-fit ring that is designed for this mechanism is shown In Fig. 4. In the inner section of crown, a similar groove has been made with sharp corner in order to make a perfect match with the integrated implant body (Figs. 5& 6). Hence, by the proposed mechanism in this research, crown may be assembled to the implant body by a snap-fit ring mechanism effortlessly.

During the implant operation, while embedding the new dental implant into the patient bone takes place, first, spongy and compact bone of patient are drilled by surgeon and then the new implant is embedded in. Next, the patient has to wait for several weeks before completion where the solid integration between the implant and bone takes place. Thereafter, snap-fit ring

is placed in the implant groove where the sharp corner of snap-fit ring supports the crown securely and firmly. The embedding process of this new dental implant mechanism is shown in Fig. 6, and the assembly process of implant, snap-fit ring and crown is shown from different views in Fig. 7.
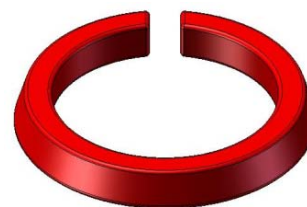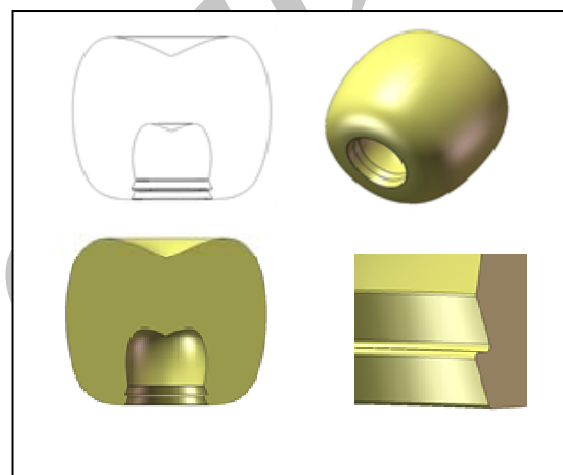

Fig. 4   Snap-fit ring

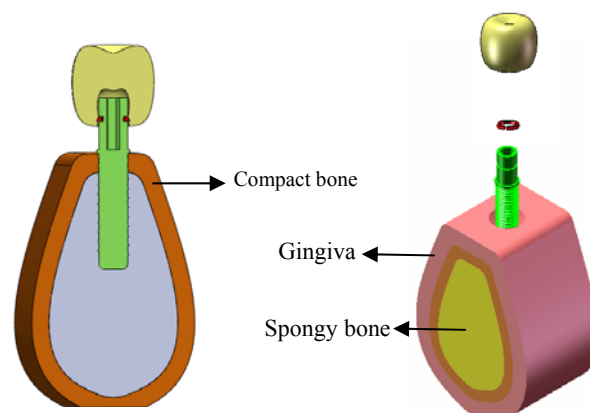**Fig. 5**   Crown and its inner groove

Fig. 6   Embedding process of new dental implant mechanism in patient bone

3   FINITE ELEMENT MODELING

3.1. CAD modeling

A 3-D model of a mandibular section of bone with missing second premolar and its superstructures were

used in this study. A mandibular bone model was selected, simulating A-2 type bone, according to the classification system of Lekholm and Zarb [17]. A bone block, 24.2 mm high and 16.3 mm wide, representing the section of the mandible in the second premolar region was modeled. It consisted of a spongy bone and compact bone by 2 mm of gingiva where the implant was positioned in the modeled spongy and compact bone block. Next, the crown fits to implant by a snap-fit ring. The implant and its superstructure were modeled by using Solid Works CAD software as shown in Figures 2-7.
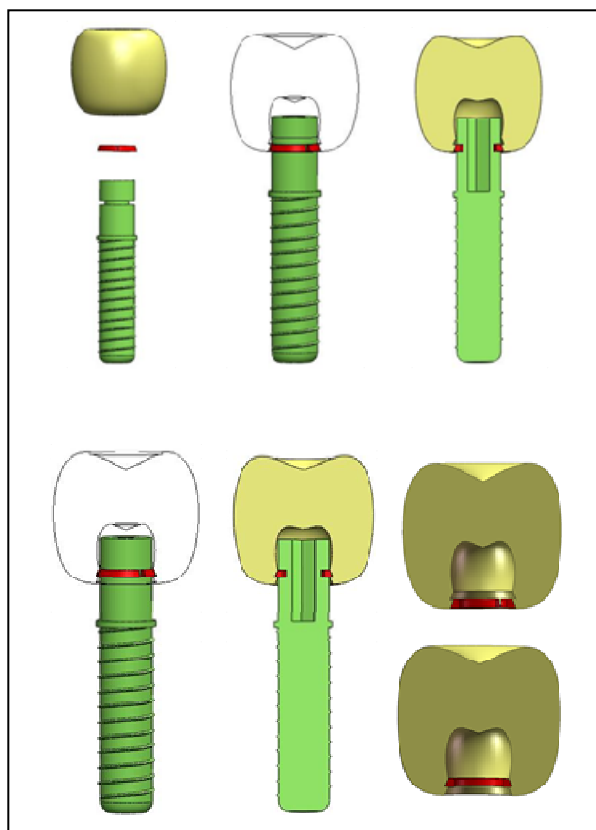

Fig. 7   Assembly process of implant, snap-fit ring and crown in new dental implant mechanism

3.2. Finite element analysis

3.2.1. Finite element modeling

Finite element model required in FE analysis is created by discrediting the CAD geometric model as shown in Fig.8 into smaller and simpler elements. The finite element models are shown in Fig.8.

The FEM model consists of total178017nodes and total of 126544elements. The physical interactions at implant–bone, implant–snap fit ring, implant–crown, and snap fit ring–crown during loading are taken into account through bonded surface-to-surface contact features of Cosmos Works software. The finite element analysis has been performed by using FEM commercial code of Cosmos Works software.
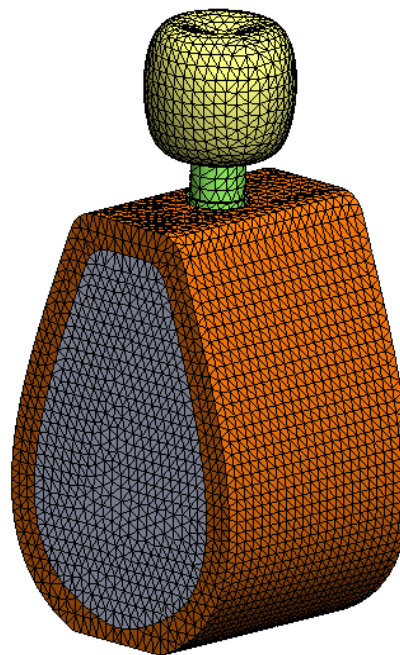

Fig. 8   Finite element model of spongy and compact bone, implant, snap-fit ring and crown

In this work, Ti–6Al–4V for implant fixture, cobalt–chromium alloy for snap-fit ring, feldsphatic porcelain for crown are used in the finite element analysis. The structures in the model were all assumed to be homogeneous and isotropic and to possess linear elasticity. Mechanical properties of materials used in this study are shown in Table 1.

Table 1 Mechanical properties of materials used in the study

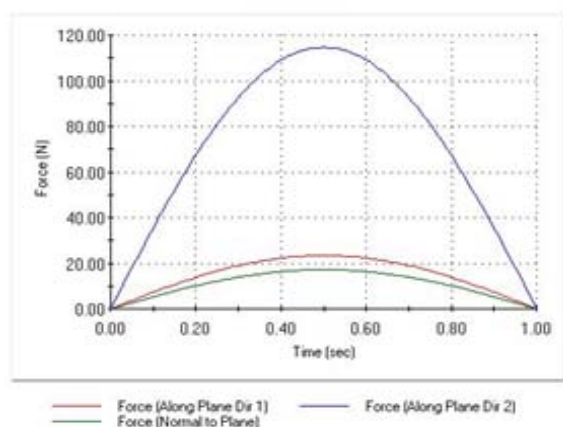| Material	Young's Modulus (GPa)	Poisson ratio (ν)	Yield strength (MPa)
Ti-6Al-4V	110	0.32	800
Cobalt chromium alloy	220	0.30	720
Feldsphatic porcelain	61.2	0.19	500
Compact bone	14.7	0.30	130
Spongy bone	1.4	0.30	130
Gingiva	0.0196	0.30	-

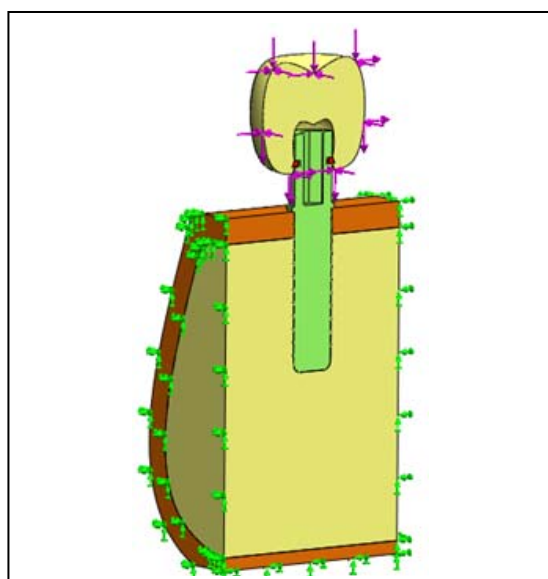**Fig. 9** Applied loads and boundary conditions of FEM model

Fig. 10 Dynamic loading variations during masticatory in 1sec

3.2.2. Loading and boundary conditions

Static and dynamic analyses of the implant should be conducted to ensure the design safety. In the literature, implants are often worked according to the results of static analysis. Static finite element analyses are mostly conducted under masticatory forces. However, dynamic effects may add up at least to 10–20% more loading on implant which must be taken into account in order to avoid fracture or fatigue failure of the implant. To investigate how static and dynamic analysis results may differ from each other, implant is analyzed under static masticatory and dynamic loading.

Three dimensional loading of an implant during masticatory is respectively 17.1 N, 114.6 N, and 23.4 N in a lingual, axial, and mesiodistal direction, simulated average masticatory force in a natural, and oblique direction (Fig. 9).

These components represented masticatory force of 118.2 N in an angle of approximately 75º to the occlusal plane [16]. This 3-D loading acted on the lingual inclination of buccal cusp of the crown. The force magnitudes, as well as the acting point, were chosen based on the work of Mericske-Stern [16]. The FEM model was fixed at the bottom and two side surfaces of mandibular as shown in Fig. 9.

Time history of the dynamic loading components for one second is demonstrated in Fig. 10. These estimations were based on the assumption that an individual has three episodes of chewing daily, each 15 minutes in duration at a chewing rate of 60 cycles per minute (1 Hz). This is equivalent to 2700 chewing cycles per day or roughly 10^6 cycles per year [18], [19].
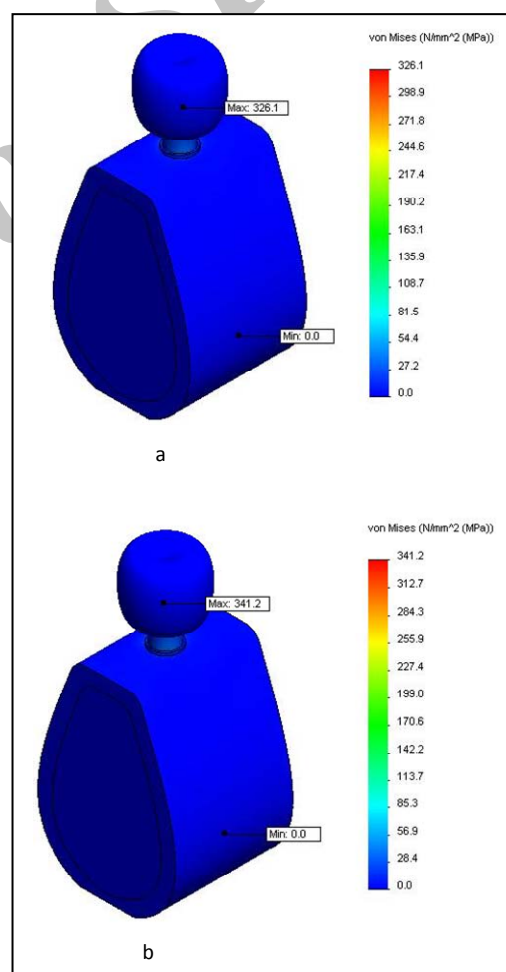

Fig. 11 Stress distribution in assembly model of compact and spongy bones, implant, snap-fit ring and crown during a) static and b) dynamic loading

4   FINITE ELEMENT ANALYSIS RESULTS

Finite element analysis results reveals that the maximum Von-Mises stresses in dynamic loading are more than those of static loading. In Fig. 11, stress distribution in assembly model of compact and spongy bone, implant, snap-fit ring and crown is shown for both loading conditions.Likewise, the maximum Von-Mises stresses that are developed at the implant, snap-fit ring, crown, spongy bone and compact bone for all loading conditions are presented in Table 2.

Table 2   Maximum Von-Misses stresses developed during static and dynamic loading

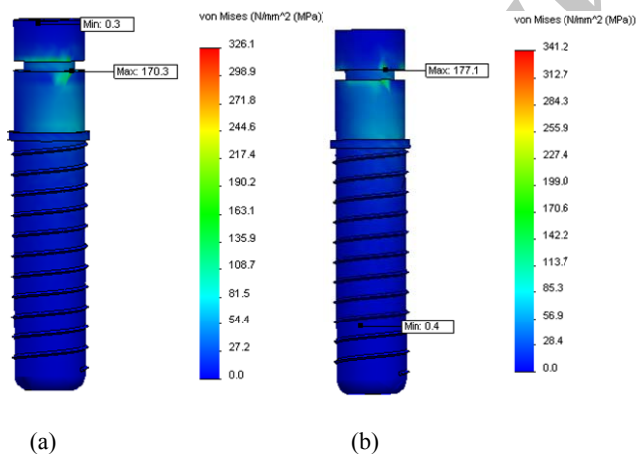|	Equivalent (Von-Misses) stress	
	Static loading (MPa)	Dynamic loading (MPa)
Implant	170.3	177.1
Snap-fit ring	311.5	341.2
Crown	326.1	273.9
Compact bone	31.9	37.4
Spongy bone	21.2	20.2

(a)   (b)

Fig. 12   Stress distribution of the implant fixture during a) staticand b) dynamicloading

4.1. Implant
Figure 12 represents the stress distribution in the implant fixture during static and dynamic loading, where the maximum stresses were located on the neck of the implant while it is connected to snap-fit ring for both loading conditions.The maximum Von-Misses stresses for the implant in static and dynamic loadings were 170.3MPa and 177.1MPa, respectively. For the static and dynamic loading, the maximum stress values within the implant body were 21.3% and 22.1% of the yield strength, respectively. The maximum stress values at the implant body of both loading conditions were well below the yield strength (yield strength for Ti-6Al-4V, is 800 MPa).
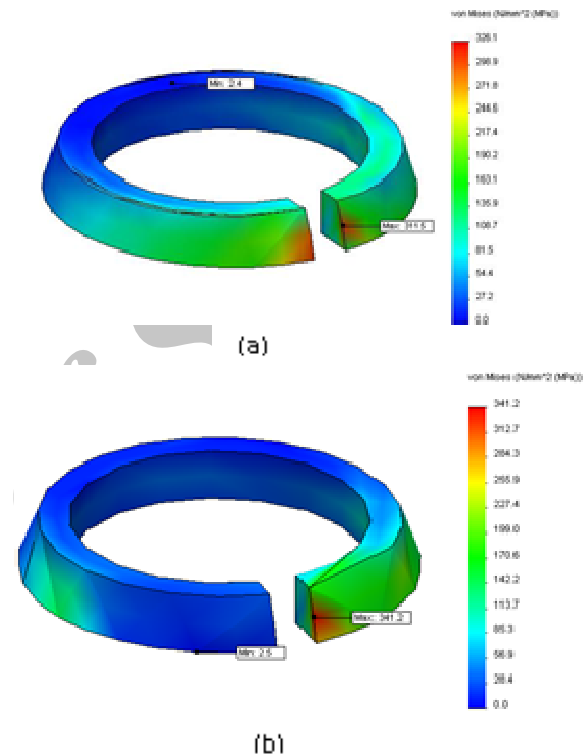

(a)

(b)

Fig. 13   Stress distribution of the Snap-fit ring during a) static and b) dynamic loading

4.2. Snap-fit ring
Figure 13 represents the stress distribution in the snap-fit ring during static and dynamic loading. The highest Von-Misses stress value was found for the snap-fit ring during dynamic loading. However, the lowest stress value was observed on snap-fit ring during static loading. The maximum Von-Misses stresses for the snap-fit ring in static and dynamic loading were 311.5MPa and 341.2MPa, respectively. For static loading, the maximum stress value within the snap-fit ring was 43.2% of the yield strength. The maximum stresses within the snap-fit ring during dynamic loading reached 47.4% of the yield strength. For both loading conditions, the maximum Von-Misses stress values in the snap-fit ring were investigated to be well below the yield strength (yield strength for Cobalt-chromium, is 720 MPa).

4.3. Crown

The maximum stress values on the crown, after applying two different loading conditions, are demonstrated in Fig. 14. The maximum stresses were located on the connection area of crown to snap-fit ring for both loading conditions. The maximum Von-Misses stresses for the crown during static and dynamic loading were 326.1MPa and 273.9MPa, respectively. For static loading, the maximum stress value within the crown was 65.2% of the yield strength. The maximum stresses within the crown of dynamic loading reached 54.8% of the yield strength. For both loading conditions, the maximum Von-Misses stress values in the crown were investigated to be well below the yield strength (yield strength for Feldsphatic porcelain, is 500 MPa).
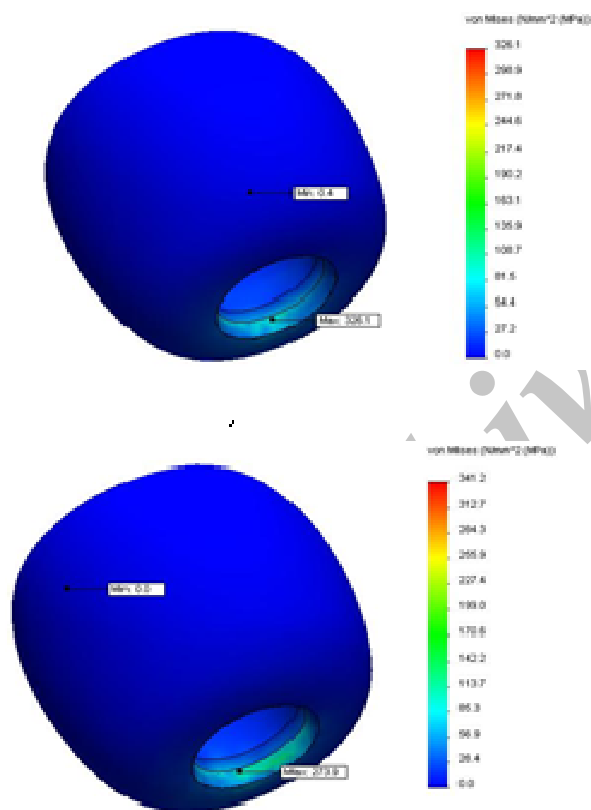

Fig. 14   Stress distribution of the crown during a) static and b) dynamic loading

4.4. Compact bone

Figure 15 represents the stress distribution of the compact bone during static and dynamic loading. The maximum stresses were located on the connection area of compact bone to implant for both loading conditions. The maximum Von-Misses stresses for the compact bone during static and dynamic loading were 31.9MPa and 37.4MPa, respectively. For the static and dynamic loading, the maximum stress values within the compact bone were 24.5% and 28.7% of the yield strength, respectively. The maximum stress values at the compact bone of two loading conditions were well below the yield strength (yield strength for compact bone, is 130MPa).
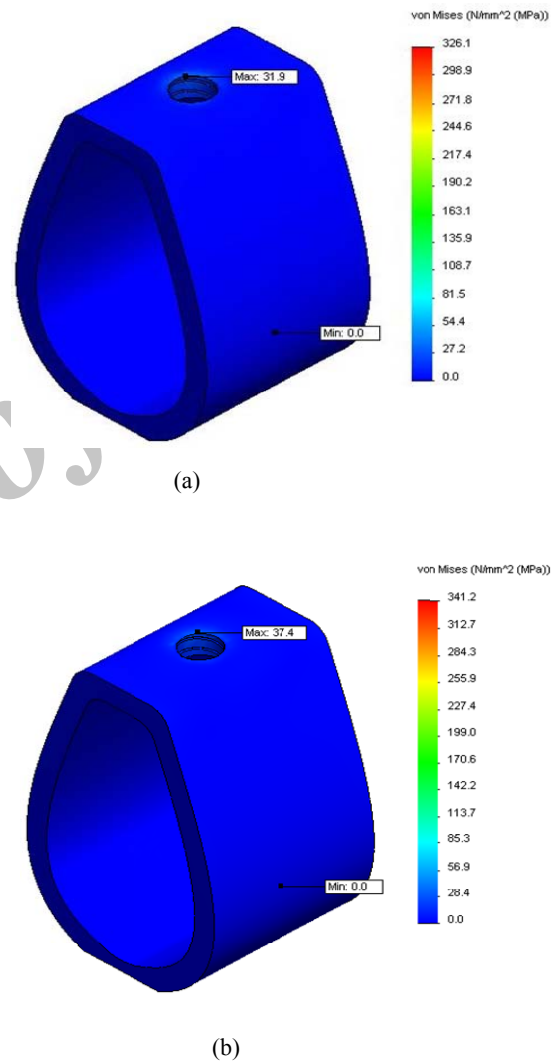

(a)

(b)

Fig. 15   Stress distribution of the compact bone during a) static and b) dynamic loading

4.5. Spongy bone

Figure 16 represents the stress distribution of the spongy bone during static and dynamic loading. The maximum stresses were located on the connection area of spongy bone to implant for both loading conditions. The maximum Von-Misses stresses for the spongy

bone during static and dynamic loading were 21.2 MPa and 20.2 MPa, respectively. For the static and dynamic loading, the maximum stress values within the spongy bone were 16.3% and 15.5% of the yield strength, respectively. The maximum stress values at the spongy bone of two loading conditions were well below the yield strength (yield strength for spongy bone, is 130 MPa).
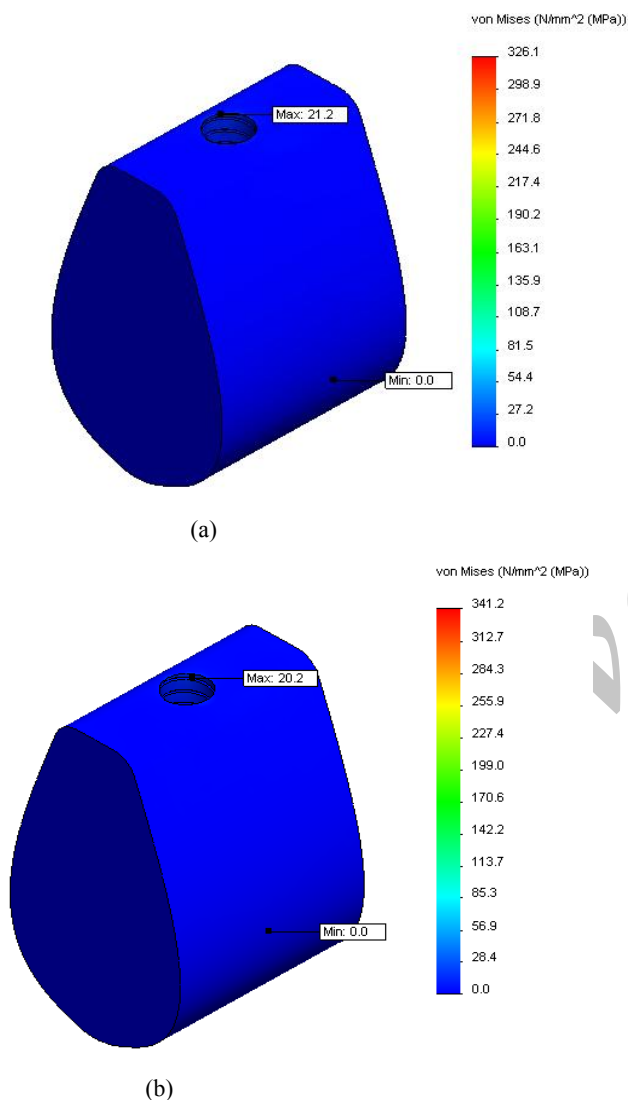

(a)

(b)

Fig. 16   Stress distribution of the spongy bone during a) static and b) dynamic loading

5   DISCUSSION

The finite element method is one of the most frequently methods used in stress analysis in industry and science. It is commonly utilized for analyzing hip joints, knee prostheses, and dental implants [16]. The results of the FEA computation depend on many individual factors,

including material properties, boundary conditions, interface definition, and also on the overall approach to the model [20]. It is apparent that the presented model was only an approximation of the clinical situation. The basic purpose of the bioengineering in dentistry which analyzed biomechanical principles in in-vitro studies was to extrapolate the findings relevant to the risk factors instead of experiencing them empirically in clinical applications. However, the stress levels that actually cause biological response, such as resorption and remodeling of the bone, are not comprehensively known. Therefore, the data of stress provided from finite element analysis require substantiation by clinical research [21].

Modeling the exact geometry of the implant complex, including the thread helix of screw and screw bore, was essential for finite element analysis [21]. Several assumptions were made in the development of the model in the present study. The structures in the model were all assumed to be homogeneous and isotropic and to possess linear elasticity. When applying FEA to dental implants, it is important to consider not only axial loads and horizontal forces but also the combination loadings (oblique occlusal force) must be considered [22]. So in this study, 118.2 N force was applied to 75˚ angular to crown that simulates chewing force at oblique direction.

All interfaces between the components were assumed to be bonded [6] (and the references cited therein). Bones loss and early implant failure after loading results most often from excess stress at the implant–bone interface [23]. This phenomenon is explained by the evaluation of finite element analysis of stress contours in the bone. The mechanical distribution of stress occurs primarily where bone is in contact with the implant [1]. To investigate how static and dynamic analysis results differ from each other, implant is analyzed under static load and dynamic chewing load. The other works that supported this study are Zhang and Chen's work and Kaybasiand et al. Zhang and Chen compared dynamic loading with static loading in three dimensional FEA models with a range of different elastic moduli for the implant. Their results showed that, comparing the static loading model, the dynamic loading model resulted in higher maximum stress in bone-implant interface as well as a greater effect on stress levels when elastic modulus was varied [24]. Moreover, Kayabasiand et al. investigated static, dynamic and fatigue behaviors of the dental implant. Their results showed that dynamic effects may add up to about 10-20% or addtional loading to the implant, which must be taken in to account in order not to cause fracture or fatigue failure on the prosthesis [16]. A finite element study demonstrated that the maximum stresses in bone concentrated at the connection between the implant and bone. Maintenance of bone levels may

be achieved by proper implant and prosthesis design. This aspect may be better understood by the use of computer aided analyses and relevant studies [25].

6 CONCLUSION

In this research a new dental implant was presented which constitutes of only two components: integrated implant, and the crown fitted to the implant by a snap-fit ring mechanism. This new design is based on simplifying the mechanism of attachment of retainer to the abutment enabling dentist to install the restoration easily. The results of FEM were presented and proved to satisfy different loading conditions successfully. One of the most important factors in the implant design is the investigation of static and dynamic behaviors of dental implant. In this study, static and dynamic behaviors of this new dental implant are investigated. For the loading conditions tested, the maximum stress values did not reach the yield strength of implant, snap fit ring, crown and spongy and compact bone materials. It seems that the implant resists all static and dynamic loading conditions with no trouble during loading.

The maximum Von-Misses stresses for the implant in static and dynamic loading were 170.3 MPa and 177.1 MPa, respectively (yield strength for Ti-6Al-4V, is 800 MPa). The maximum Von-Misses stresses for the snap-fit ring in static and dynamic loading were 311.5 MPa and 341.2 MPa, respectively (yield strength for Cobalt-Chromium, is 720 MPa).

The maximum Von-Misses stresses for the crown during static and dynamic loading were 326.1 MPa and 273.9 MPa, respectively (yield strength for Feldsphatic porcelain, is 500 MPa). The maximum Von-Misses stresses for the compact bone during static and dynamic loading were 31.9 MPa and 37.4 MPa, respectively (yield strength for compact bone, is 130 MPa). The maximum Von-Misses stresses for the spongy bone during static and dynamic loading were 21.2 MPa and 20.2 MPa, respectively (yield strength for spongy bone, is 130 MPa). In both static and dynamic loading, the maximum stress values for all components of the two loading conditions were well below the yield strength of the materials. Therefore, the dental implantsmay be designed and studied in computer environment before theyare implemented on the patients. This will save time for the design and prevents any possibledamage caused by miss-implementation of implants.

REFERENCES

[1] Schroeder, A., "Oral Implantology: Basic, ITI Hollow Cylinder System", New York: Thieme Medical Publishers, 1996, pp. 60-65.

[2] Adell, R., Eriksson, B., Lekholm, U., Branemark, P. I., and Jempt, T., "LongTerm Follow-up Study of Osseointegrated Implants in the Treatment of Totally Edentulous Jaws", Int. JOral Maxillofacial Implants, Vol. 5, 1990, pp. 347-359.

[3] Geng, J., Tan, K. B. C., and Liu, G., "Application of Finite Element Analysis in Implant Dentistry: AReview of the Literature", J. Prosthet Dent, Vol. 85, 2001, pp: 585-598.

[4] Graf, H., and Geering, A. H., "Rationale for Clinical Application of Different Occlusal Philosophies", Oral Sci Rev, Vol. 10, 1977, pp. 1-10.

[5] Kitagawa, T., Tanimoto, Y., and Odaki, M., "Influence of Implant/Abutment Joint Designs on Abutment Screw Loosening in a Dental Implant System", J. Biomed Mater Res B Appl Biomater, Vol. 75, No. 2, 2005, pp. 457-463.

[6] Theoharidou, A., Petridis, H. P., and Tzannas, K., "Abutment Screw Loosening in Single Implant Restorations: a Systematic Review", Int. J. Oral Maxillofacial Implants, Vol. 23, No. 4, 2008, pp.681-690.

[7] Tang, C. B., Liu, S. Y., Zhou, G. X., Yu, J. H., Zhang, G. D. Bao, Y. D., and Wang, Q. J., "Nonlinear Finite Element Analysis of Three Implant–Abutment Interface Designs", International Journal of Oral Science, Vol. 4, 2012, pp.101-108.

[8] Simon, R. L., "Single Implant-Supported Molar and Premolar Crowns: A Ten-Year Retrospective Clinical Report", J. Prosthet Dent, Vol. 90, 2003, pp. 517-521.

[9] Carlson, B., Jonsson, G., Sandahl, L., Nordin, T., Hising, P., Nelvig, P., and Branting, C. A., "1-Year Clinical Report of a One-Piece Implant Abutment", Int. J. Prosthodont, Vol. 14, 2001, pp. 159-163.

[10] Nergiz, I., Schmage, P., and Shahin, R., "Removal of a Fractured Implant Abutment Screw: A Clinical Report", J. Prosthet Dent, Vol. 91, 2004, pp. 513-517.

[11] Meijer, H. J. A., Geertman, M. E., Raghoebar, G. M., and Kwakman, J. M., "Implant-Retained Mandibular Overdentures: 6-Year Results of a Multicenter Clinical Trial on 3 Different Implant Systems," J. Oral Maxillofacial Surg, Vol. 59, 2001, pp. 1260-1268.

[12] Goodacre, C. J., Bernal, G., Rungcharassaeng, K., and Kan, Y. K., "Clinical Complications with Implants and Implant Prostheses", J. Prosthet Dent, Vol. 90, 2003, pp. 121-132.

[13] Binon, P. P., "The Effect of Implant/Abutment Hexagonal Misfit on Screw Joint Stability", Int. J. Prosthodont, Vol. 9, 1996, pp. 149-160.

[14] Sato, Y., Wadamoto, M., Tsuga, K., Teixeira, E. R., "The Effectiveness of Element Downsizing on a Three-Dimensional Finite Element Model of Bone Trabeculae in Implant Biomechanics", J. Oral Rehabil, Vol. 261999, pp. 288-291.

[15] Sahin, S., Cehreli, M. C., and Yalc, M. E., "The Influence of Functional Forces on the Biomechanics of Implant-Supported Prostheses-AReview", J. Dent, Vol. 20, 2002, pp. 271-282.

[16] Kayabasi, O., Yuzbasioglu, E., and Erzincinli, F., "Static, Dynamic and Fatigue Behaviors of Dental Implants using Finite Element Method", Advances in Engineering Software J. Vol. 37, 2006, pp. 649-658.

[17] Lekholm, U., and Zarb, G., Patient Selection and Preparation. In: Bra˚nemark, P. I., Zarb, G., Albrektsson, T., editors. Tissue-Integrated Protheses.

Osseointegration in Clinical Dentistry. Chicago: Quintessence; 1985, pp. 199-209.

[18]   Binon, P. P., and McHugh, M. J., "The Effect of Eliminating Implant/Abutment Rotational Misfit on Screw Joint Stability", Int. J. Prosthodont, Vol. 9, 1996, pp. 511-519.

[19]   Wiscott, H. W., Nicholls, J. I., and Belser U. C., "Stress Fatigue: Basic Principles and Prosthodontic Implications", Int. J. Prosthodont, Vol. 8, 1995, pp. 105-116.

[20]   Bathe, K. J., Finite Element Procedures, Upper Saddle River (NJ): Prentice-Hall; 1996, pp. 148-377.

[21]   Holmgren, E. P., Seckinger, R. J., Kilgren, L. M., and Mante, F., "Evaluating Parameters of Osseo Integrated Dental Implants using Finite Element Analysis-ATwo Dimensional Comparative Study Examining the Effects of Implant Diameter, Implant Shape, and Load Direction", J. Oral Implantolology, Vol. 24, 1998, pp. 80-88.

[22]   Zhang, J. K., and Chen, Z. Q., "The Study of Effects of Changes of the Elastic Modulus of the Materials Substitute to Human Hard Tissues on the Mechanical State in the Implant–Bone Interface by Three-Dimensional Anisotropic Finite Element Analysis", West China J. Stomatol, Vol. 16, 1998, pp. 274-278.

[23]   Misch, C. E., Contemporary Implant Dentistry. 2nd ed. St. Louis: Mosby; 1998, pp. 109-134, 207-217, 329-343, and 595-608.

[24]   Sertgoz, A., "Finite Element Analysis Study of the Effect of Superstructure Material on Stress Distribution in an Implant-Supported Fixed Prosthesis", Int. J. Prosthodont, Vol. 10, 1997, pp. 19-27.

[25]   De Tolla, D. H., Andreanas, S., and Patra, A., "The Role of the Finite Element Model in Dental Implants", J. Oral Implantol, Vol. 26, 2000, pp. 77-81.