MATTER: International Journal of Science and Technology
ISSN 2454-5880

Belagh & Kashani, 2017

Volume 3 Issue 3, pp. 195-211

Date of Publication: 19th December, 2017

DOI-https://dx.doi.org/10.20319/mijst.2017.33.195211

This paper can be cited as: Belagh, A.B.F & Kashani, H.M. (2017). Studying the Effect of the Preload Induced by Screw Tightening on Stress Distribution of a Dental Implant. MATTER: International Journal of Science and Technology, 3(3), 195-211.

STUDYING THE EFFECT OF THE PRELOAD INDUCED BY SCREW TIGHTENING ON STRESS DISTRIBUTION OF A DENTAL IMPLANT

Alireza BidarFarrokh Belagh
Department of Mechanical Engineering, Islamic Azad University West Tehran Branch, Tehran, Iran
alireza.bidar.tj@gmail.com

Hamed Moayeri Kashani
Department of Mechanical Engineering, Islamic Azad University West Tehran Branch, Tehran, Iran
moayeri.hamed@wtiau.ac.ir

Abstract

The success of implants is largely dependent on initial stability and long-term osseointegration due to optimal stress distribution around the bone and implant. The purpose of this study is the numerical analysis of stress distribution in jaw bone and implant using finite element analysis considering the static forces caused by screw tightening and masticatory preloads. These forces and design limitations have been applied in accordance with implant science in dentistry to provide a proper stress distribution. First, all the parts were modelled with Solidworks software and then transferred to Abaqus software for analysis and applying the forces. For a better and more exact stress distribution analysis in the bone and implant, this analysis was conducted by two steps, that after determining the properties of each part, boundary conditions, loading and finally meshing the complex using hexahedral meshes and Match mesh technique, the abutment *was tightened inside the implant with different tightening torques through six tests to apply preload in first step which this force applying induced stress in jaw and implant. Then, the amount of jaw force was applied to the crown surface. The results showed that the preload is quite effective in bone and implant stress distribution. However, its value only affects the surface stresses of the implant and has little effect on the of jaw bone stress value. This study can be carried on to evaluate the implants life considering Preload.*

Keywords

Dental Implants, Preload, Stress, Fatigue, Finite Element Analysis

---

1. Introduction

Implants are thin pins made of pure biocompatible titanium that usually have the elastic modulus 5-10 times greater than cortical bone. They are surgically inserted into the jaw bone and dental prosthesis can be attached to them. The complete bone and implant attachment usually takes about 3-6 months. Implants attach to jaw bone through a phenomenon called Osseo integration that was first discovered by Dr. Branemark. Osseo integration is the creation of a direct and structural connection between living bone and the surface of the implant. When dental implants were introduced in the late sixties, they revolutionized the dental therapies and provided very good long-term results for patients (Adell et al., 1990; Adell et al., 1981). In the long-term success of a dental implant evaluation, the reliability and the stability of the implant–abutment and implant bone is especially important (Kayabasi et al., 2006). These long-term successful evaluations always require a balance or dynamic equilibrium between mechanical and biological factors. Implants are combined with complications such as screw loosening, screw fracture, framework fracture, and infrequently, implant fracture in some prosthetic applications which these should be avoided in order to achieve their great success (Perez, 2012). To avoid such problems and design a successful dental implant, the main objective should be to ensure that the implant can support biting or jaw forces and deliver them safely to interfacial tissues over the long-term (Perez, 2012). Prosthetic components are always subjected to a complex pattern of horizontal and vertical force combinations (Kayabasi et al., 2006). Therefore, dental implants have been subjected to many cyclic loadings throughout their lives which are mostly caused during masticatory. The fatigue caused by these cyclic loadings can make the implant fracture which leads to a large burden to the patient from clinical viewpoint. Thus, long-term stability requires to be assured against fatigue damage, and it causes the requirement for establishing the structural stability implant itself along with the stability of adjacent osseous tissues surrounding the fixture and jaw bone (Su Bae & Jeong, 2011). The ideal conditions of the jaw bone lead to a high stability of the implant against cyclic loads, while thinness of the cortical bone along with low rigidity and low density of the jaw bone leads to low stability of the implant. Also, studies have shown that stress concentration in the cortical bone is much greater than the stress concentration in the cancellous bone (Guan et al., 2009; Yang & Xiang, 2007).

The assembly of the implant complex itself should be fully understood prior to analyzing the external forces. The assembly process itself generates some of the applying forces to the complex (Guda et al., 2008). Abutment screw tightens by input tightening torque to the upper part of the abutment within the implant which this result in an axial force called Preload. Therefore, one of the most effective mechanical factors to success the implants is the preload caused by tightening the screw of the implant. The loss of preload during prosthesis applying loads effects on the implant-prosthesis contact performance which this can increase stress in implant and other components, and eventually the surrounding bone. Applying more preload also increases the resistance against screw loosening and the stability of implant-abutment adhesion (FilhoI et al., 2010).

Computational methods such as finite element method are mainly used in biomechanics. It is an important tool to analyze and design various dental implant models. There is no doubt that finite element method is the most general and widely accepted technique in biomechanics. This method has been applied for various fundamental techniques analysis, implant applications, prosthesis design and implants effects investigating, loading magnitude and direction and implant-bone interface conditions (Perez, 2012). The mentioned text indicates that how important and necessary considering the preload can be to analyze the stress and the following the fatigue and fracture analysis. Since an implant should have a direct adhesion between the fixture and the jaw bone, a failure in the fixture can have serious consequences for the patient. So, better designs can be presented by studying the stress distribution and fatigue analysis considering the preload caused by the abutment tightening. Also, because the finite element analysis has no laboratory limitations in studying different parameters and is a powerful tool in biomechanical problems analysis, it is a proper method in these types of problems.

Therefore, in this study, the stress distribution around the jaw bone and implant taking into account multiple preloads in a dental implant by the finite element analysis and using the mesh technique and hexahedral meshes has been investigated.

2. Related Works

In (Guda et al., 2008), the authors identified significant and effective variables affecting the preload and examined the inherent variability of material properties, surface interactions and the probability of obtaining an optimal preload. The generated Finite Element model was integrated with probabilistic analysis software. Probabilistic analyzes were performed in a well-lubricated and a dry environment. The results showed that the elastic modulus of the abutment screw material (gold alloy) was effective at high preload values. However, Elastic moduli and the Poisson's ratios of titanium and the gold alloy have little influence on the preload variations at medium preload values. Also, the overall results of the probabilistic analysis showed that the probability of obtaining an optimal preload was only 0.02%.

In (Bulaqi et al., 2014), the authors studied the effect of the coefficient of friction and tightening speed of the abutment screw on the implant using a method base on stress distribution. For this purpose, after the precise geometric modeling to achieve the target torque, an angular displacement was applied to the abutment screw head at different coefficients of friction and tightening speeds in order to obtain the values of torque, preload, energy distribution, elastic energy, and efficiency. In addition, the torque distribution ratio and preload values were compared to predicted values. The results indicated that increasing the tightening speed and reducing the coefficient of friction have the same results and tightening speed has more influence at a lower coefficient of friction to obtain a given target torque. Increasing the tightening speed of an abutment screw and reducing the coefficient of friction are two influential factors to increase the preload and ultimately enhancing the stability and preventing screw loosening.

3. CAD and Finite Elements Modelling

3.1 CAD Modelling

First, a 3D model of a mandible obtained from CT scan results from Mimics software was imported into Solid works software by changing the format and a new and simplified model was designed according to the dimensions and characteristics of the main model (figure 1). The designed jaw was imported into Abaqus software after removing the first molar tooth. Jaw bone includes two parts; cortical bone and cancellous bone which cortical bone with a thickness of approximately 2 mm has surrounded the cancellous bone (Su Bae & Jeong, 2011). In order to have a better and faster analysis, a part of the jaw bone which the first molar tooth is located was separated from the model for meshing which this part has an approximately 28.30 mm width and 31.5 mm length (figure 2).

A 3D implant model of 3.6 mm diameter and 10 mm length was designed using Solid works software based on a real implant model made in Korea. Its abutment was also designed with a height of 5 mm (figure 3).

Cobalt-Chrome alloy (wiron 99; Bego, Bremen, Germany) was chosen for the material of the framework and feldspathic porcelain for occlusal[1] surface. The framework was finally located in a height of 6.86 mm and the occlusal surface in a height of 8.36 above from the upper body section of the implant. After the completion of the design process, different components were assembled and the implant was placed in the jaw bone in the first molar tooth position (Figure 4). Figure 5 presents a better understanding of how the components are assembled and the overall assembly of a cross section of the overall model.

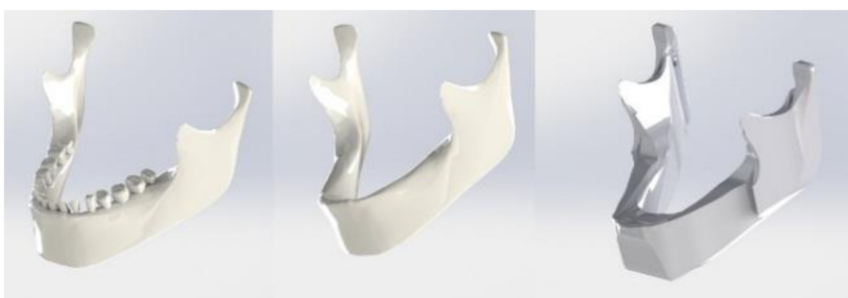

Figure 1: *3D Model of Mandibular*

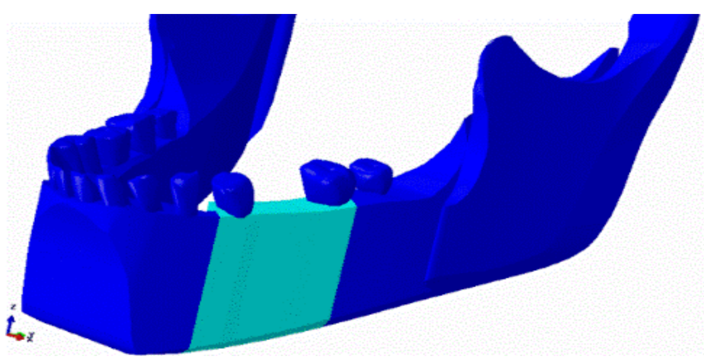

Figure 2: *Separated Part from Mandibular without First Molar Tooth*

[1] masticatory surface

Figure 3: *3D model of 1-implant 2-abutment 3-frame work 4- crown*

3.2 Finite Element Analysis

The finite element analysis requires geometric models to be in smaller and simpler elements (Figure 6). The finite element model is totally made of 112780 3D elements which separately include: 29988 elements for implant, 9864 elements for abutment, 16480 elements for occlusal surface, 2696 elements for the framework and 53752 elements for the jaw bone. It should be noted that we were inevitably compelled to use tetrahedral mesh for a part of the occlusal surface. 3D hexahedral elements in all the components are C3D8R-type and linear. The final meshing is illustrated in figure 7.

In this study, the implant and abutment were made of titanium grade 5 alloy (Ti–6Al–4V), frame work of cobalt-chrome alloy, occlusal surface of feldspathic porcelain which is a ceramic-metal combination for finite element analysis. The behaviour of the mentioned materials was also considered as isotropic linear. Mechanical characteristics of the applied materials are tabulated in Table 1.

Cortical and cancellous bones were modeled using two types of particular materials due to the importance of the bone behaviour to have a more exact and natural influence of the jaw bone on the implant. The outer layer or cortical bone was modelled by transversely isotropic characteristics. The inner layer or cancellous bone was also modelled by isotropic linear characteristics (Table 2).

In the present study, due to the bone repair ability of the hard tissue, the implant was completely attached to jaw bone and the osseointegration phenomenon was completely conducted.

Also, to screw the abutment into the implant, surface to surface contact was defined with a friction coefficient of 0.3. The contact type of the abutment to the framework and the framework to the occlusal surface was considered Tie supposing 100% adhesion (Kong et al., 2009).

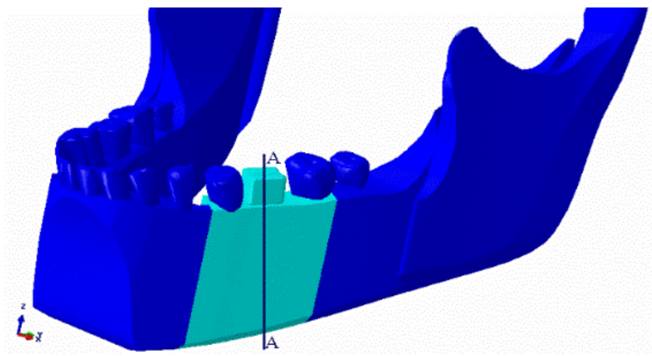

Figure 4: *Assembly of complex component*

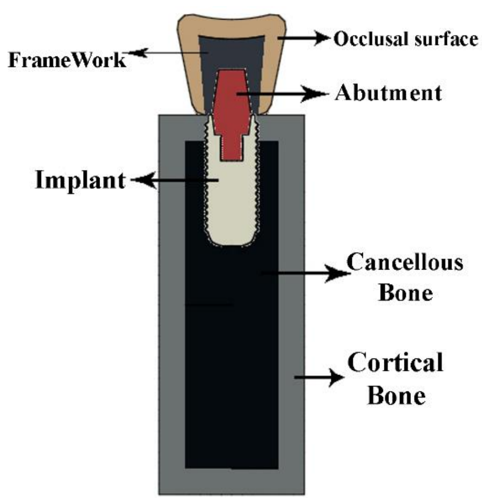

Figure 5: *Details of A-A Section*

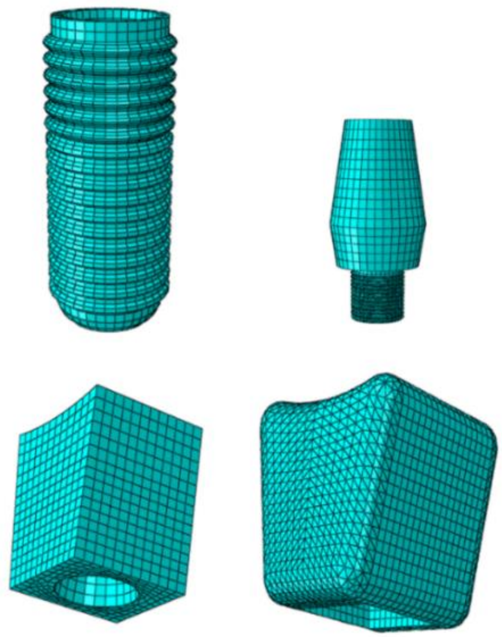

Figure 6: *Finite Element Model of Implant, Abutment, Frame Work, Crown*

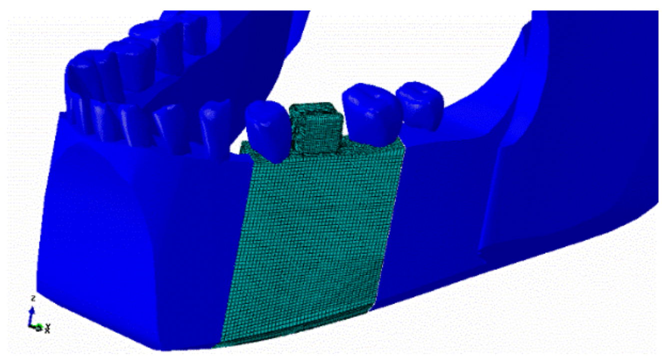

Figure 7: *Finite Element Model of Complex*

Table 1: *Mechanical Properties of Materials Used in this Study*

Material	Young's Modulus(GPa)	Poisson Ratio	Yield Strength(MPa)
Ti-6Al-4V	100	0.32	800
Cobalt-Chromium Alloy	220	0.3	720
Feldsphatic porcelain	61.2	0.19	500

Table 2: *Mechanical Properties of Bones*

Kind of bone	Young's Modulus(GPa)	Poisson Ratio	Shear modulus(GPa)
Cortical Bone	E_x=11.5	E_x=11.5	G_{xy}=3.6
	E_y=11.5	E_y=11.5	G_{xy}=3.3
	E_z=17	E_z=17	G_{xy}=3.3
Cancellous Bone	E=2.13	E=2.13	G=0.81

3.3 Loading and Boundary Conditions

Static and dynamic analyses of the implants are to be carried out to ensure about the design safety. Implants usually operate in accordance with static analyses. Static finite element analysis is often carried out under the influence of masticatory forces. The preload condition was achieved using contact analysis in the finite element models. For this purpose, the objective and contact surfaces between individual parts of the model were identified without merging the nodes between the components. Contact elements were determined between implant threads and bone, the mating surfaces of the implant and abutment, and at a distance of 0.005 mm between the contacting surfaces. The tightening torque of implant in jaw bone and between implant and abutment were both evaluated 350 N.mm. Therefore, in this study, the abutment was once tightened by the recommended and standard torque which was 350 N.mm and after that, was tested 5 times more with different torques in the abutment to investigate the influence of the preload on the stress distribution process.

The implants loading was applied with forces of 17.1 N, 114.6 N, and 23.4 N in a lingual, an axial, and a mesiodistal direction respectively. These values represented masticatory force of 118.2 N in the angle of approximately 75° to the occlusal surface. The force magnitudes, and also the acting point, were chosen based on an effort of Mericske-Stern (Kayabasi et al., 2006). To delimit boundary conditions, all the points and nodes of the lateral surfaces were fixed in three directions (Kayabasi et al., 2006; Guan et al., 2011). Boundary conditions and loading are shown in figure 8.

4. Results

Following the mentioned steps and investigating the meshing convergence at the end of the study, stress distribution has been studied in 6 separate analyses by applying different preloads. As expected, in all analyses, the maximum implant stress occurred in the first thread involved with the abutment after 2 loading stages, but as shown in Table 3 the maximum Von Mises stress did not reach the yield strength of titanium (The yield stress of titanium is 462 MPa) (Kayabasi et al., 2006). The torque applied changings also influenced the observed stress of the external threads of the implant. For further understanding of this subject, the stress values of the first involved thread of the implant and bone throughout the thread was surveyed for each 6 analyses in figure 9.

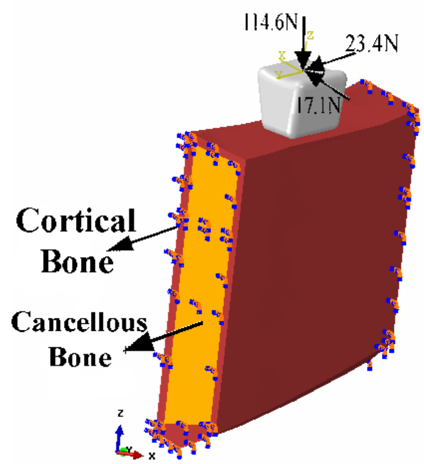

Figure 8: *Applied forces and boundary condition*

Table 3: *Maximum Values of max. Principal Stress of Implant in different Preload*

Preload(N.mm)	Max. principal stress(MPa)	Proportion of max principal stress to yield strength (%)
0	42.499	9.19
200	80.870	17.5
275	80.521	17.42
350	95.190	20.6
425	109.807	23.63
500	125.518	27.16

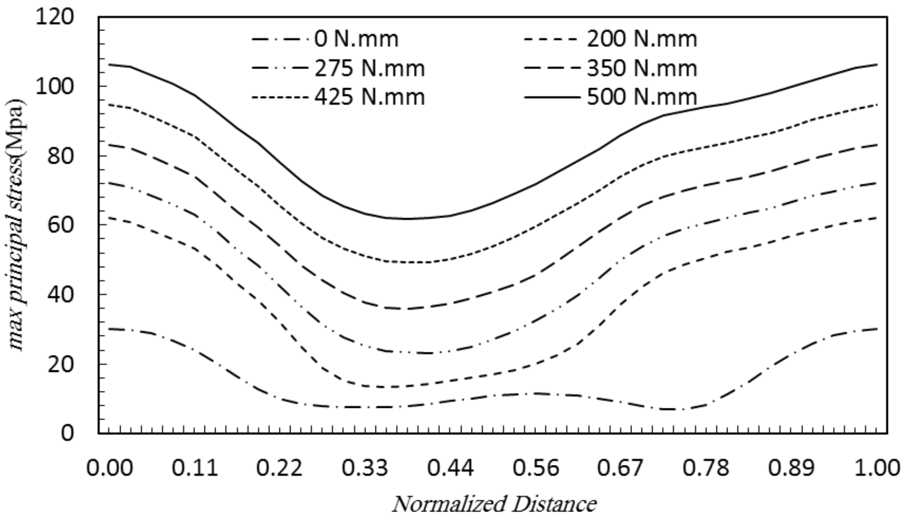

Figure 9: *Max Principal Stress Variations Curve*

Budynas and Nisbett offered equation (1) to determine the frictional resistance of conical torque (T_c) and equation (2) to determine the frictional resistance of thread torque (T_{th}). Wrench torque (T_w) is the sum of conical and thread torque (Bulaqi et al., 2014).

$$T_{\mathrm{c}} = \frac{\mu}{3\,sin\beta} \times \frac{D^3 - d^3}{D^2 - d^2} \times F = k_c \times F \tag{1}$$

$$T_{\mathrm{th}} = \frac{d_m}{2} \times \frac{L + (\mu \times \pi \times d_m \times sec\alpha)}{(\pi \times d_m) - (\mu \times L \times sec\alpha)} \times F = k_{th} \times F \tag{2}$$

$$T_{\mathrm{w}} = T_{\mathrm{c}} + T_{\mathrm{th}} = (k_c + k_{th}) \times F \tag{3}$$

$$T_{\mathrm{c-w}} = \frac{T_c}{T_w} = \frac{k_c}{k_c + k_{th}} \tag{4}$$

$$P = \frac{T_w}{k_c + k_{th}} \tag{5}$$

where T_{c-w} is the ratio of conical torque to the wrench torque, F is the preload generated in the screw, P is the preload at the desired torque, μ is the coefficient of friction, d is the inner head friction diameter, D is the outer head friction diameter, β is the cone angle, α is the half angle of the thread, L is the pitch, and d_m is the pitch diameter. These parameters are presented in figure 10. The relations between the tightening torques applied and the obtained preloads are illustrated in figure 11.

If the step of tightening the abutment screw is eliminated and in fact the preload applying is ignored, the maximum stress in the implant occurs in the first thread involved with the jaw bone with very little difference compared to the second thread. Since the created stress is trapped during tightening the abutment and remains constant, it can be predicted that the maximum stress will be transferred to the first thread involved in the implant and jaw bone by continuing the loading process at the stage of input applying force by the jaw bone and in fact with continuing the fatigue loading.

The applying torques values were 0, 200, 275, 350, 425 and 500 N.mm respectively which the torque of 350 N.mm was the standard torque for tightening the abutment screw. Von Mises stress distribution in internal and external threads of the implant for standard state and after applying the jaw force is illustrated in figures 12 and 13.

According to the stress distribution illustrated in figures 12 and 13 the first external involved thread with jaw bone has the maximum Von Misses stress after the internal implant threads which have the maximum stresses due to the presence of the preload. This demonstrates that fatigue loading has the it most significant impact on the first external thread in contact with the jaw bone which this point can be chosen as the critical point. For a better understanding the process was once carried out regardless of the preload and only by applying the force of the jaw bone to the crown. As shown in figure 14 the maximum stress occurs in the first involved thread of the jaw and implant which the value of stress, although its stress value has a negligible difference compared to the second involved thread.

After studying the stress distribution in the implant applied by different preloads, the value and pattern of stress distribution in bone should be addressed due the bone vulnerability against the applying forces.

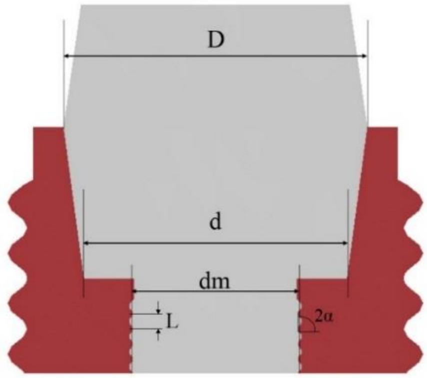

Figure 10: *Geometric Parameters*

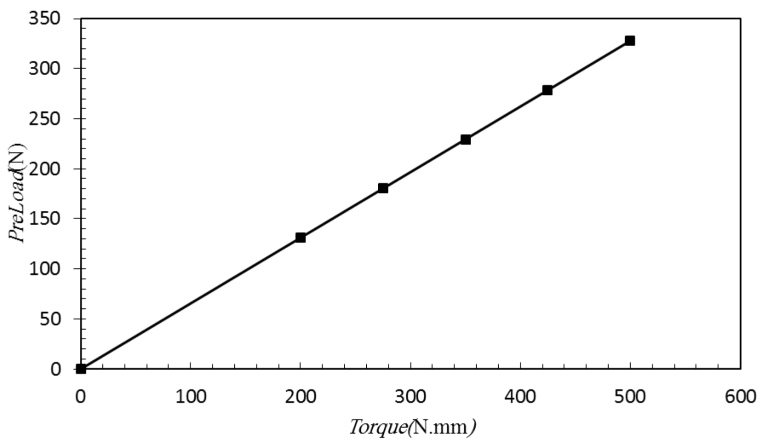

Figure 11: *Linear Relation between Applied Torque and Preload*

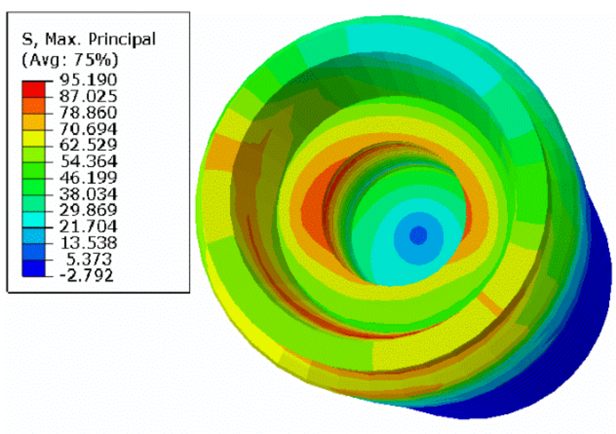

Figure 12: *Stress Distribution Contour of Max Principal Stress in Implant Inner Threads at Standard Torque Tightening*

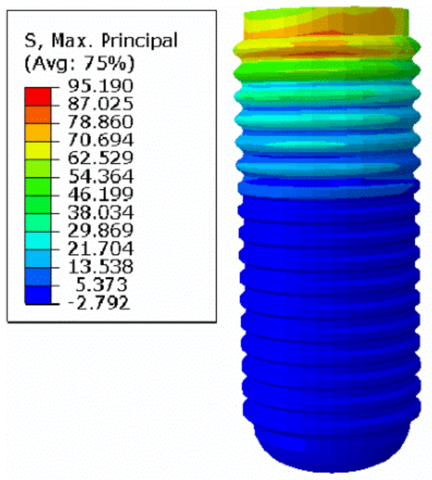

Figure 13: *Stress Distribution Contour of Max Principal Stress in Implant Outer Threads at Standard Torque Tightening*

The distribution and concentration of stress in bone by the standard preload is illustrated in figure 15. The comparison of Von Misses stress and yield stress of the bone is tabulated in table 4 for each test. Since the created stresses are concentrated in cortical bone, this comparison has been made (The yield stress for cortical bone is 69 Mpa).

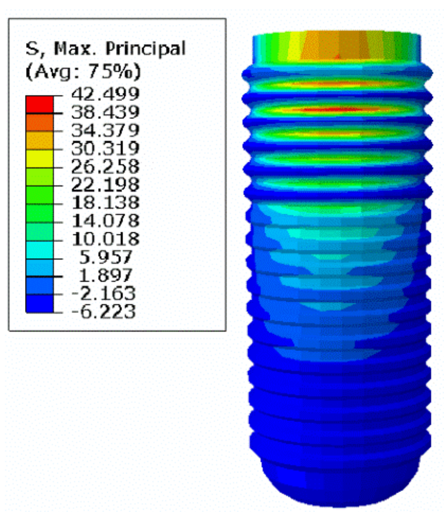

Figure 14: *Stress Distribution Contour of Max Principal Stress in Implant without Preload*

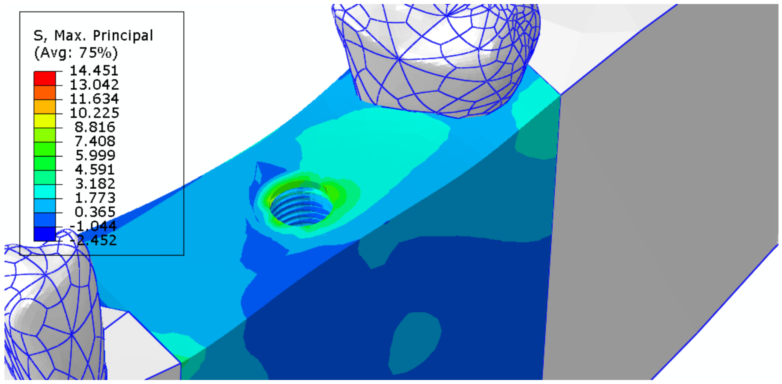

Figure 15: *Stress Distribution Contour of Max Principal Stress in Bone at Standard Preload*

Table 4: *Maximum Values of max. Principal Stress of Cortical Bone*

Preload (N.mm)	Max. principal stress (MPa)	Proportion of max principal stress to yield Strength (%)
0	7.941	11.5
200	14.208	20.59
275	14.257	20.66
350	14.451	20.94
425	14.773	21.41
500	14.727	21.34

5. Conclusion

The purpose of this study was to investigate the stress distribution in dental implant and jaw bone by applying different preloads and also without applying preload using finite element analysis inevitably with some simplifications in the design of the implant and the bone as well due to the design limitations which, of course, will not have much effect on the obtained results. Another limitation in this study was the lack of sufficient laboratory and numerical research carried out in preload review part. In the first part of this study, the stress distribution of the implant and in the next section the stress distribution of the jaw bone in different conditions has been studied which the most important results of this study are as follows:

- The presence of preload is completely effective on the stress distribution of the implant and bone.

- The value of the applied preload is highly effective on the surface stress created in the implant but it is not significantly effective on the critical points of the bone.

- In the analyses considering preload, the maximum stress occurs in the first involves thread root of the implant and bone after the contact point of the implant and abutment which we can conclude that with continuing of the fatigue loading the maximum stresses will be concentrated in this area and this area can have the minimum fatigue life in comparison with the other points. Also, in the most of the analyses, the fractured implants often fracture from the first involved thread area of the implant and the sample.

-The analysis amount of the Cortical bone is important at the location of the maximum stress. Therefore, it is recommended that the influence of the bone analysis and the materials to be considered on implant life due to the importance and impact of bone analysis on stress distribution. Additionally, it is also suggested that the life of the implants to be studied based on the presence of cracks theory due to the fact that real components always have natural defects in their structure. This study can also be carried on to evaluate the implants life considering Preload.

References

Adell, R., Eriksson, B., Lekholm, U., Branemark, P.I., Jemt, T. (1990). A Long-Term Follow-up Study of Osseointegrated Implants in the Treatment of Totally Edentulous Jaws. Int J Oral Maxillofac Implants, 5, 347-359.

Adell, R., Lekholm, U., Rockle, B., Branemark, P.I. (1981). A I5-year study of osseointegrated implants in the treatment of the edentulous jaw. Int. J. Oral Surg, 10, 387-416. https://doi.org/10.1016/S0300-9785(81)80077-4

Bulaqi, H.A., Mashhadi, M.M., Geramipanah, F., Safari, H., Paknejad, M. (2015). Effect of the coefficient of friction and tightening speed on the preload induced at the dental implant complex with the finite element method. The journal of Prosthetic Dentistry, 113, 405-411.

FilhoI, A.P.R., Fernandesi, F.S.F., StraiotoI, F.G., Silvai, W.J., Del Bel Cury, A.A. (2010). Preload loss and bacterial penetration on different implant-abutment connection systems. Braz. Dentistry Journal, 21 (2). https://doi.org/10.1590/S0103-64402010000200006

Guan, H., van Staden, R.C., Johnson, N.W., Chaye Loo, Y. (2011). Dynamic modelling and simulation of dental implant insertion process: A finite element study. Finite Elements in Analysis and Design, 47, 886-897. https://doi.org/10.1016/j.finel.2011.03.005

Guan, H., van Staden, R.C., Loo, Y.C., Johnson, N., Ivanovski, S., Meredith, N. (2009). Influence of bone and dental implant parameters on stress distribution in the mandible: A finite element study. Int Journal Oral Maxillofac Implants, 24 (5), 866-876.

Guda, T., Ross, T.A., Lang, L.A., Millwater, H.R. (2008). Probabilistic analysis of preload in the abutment screw of a dental implant complex. J Prosthet Dent, 100, 183-193. https://doi.org/10.1016/S0022-3913(08)60177-8

Kayabasi, O., Yuzbasioglu, E., Erzincanli, F. (2006). Static, dynamic and fatigue behaviors of dental implant using finite element method. Advances in Engineering Software, 37, 649-658. https://doi.org/10.1016/j.advengsoft.2006.02.004

Kong, L., Zhao, Y., Hua, K., Li, D., Zhou, H., Wu, Z., Liu, B. (2009). Selection of the implant thread pitch for optimal biomechanical properties: A three-dimensional finite element analysis. Advances in Engineering Software, 40, 474-478. https://doi.org/10.1016/j.advengsoft.2008.08.003 https://doi.org/10.1016/j.advengsoft.2008.12.010

Perez, M.A. (2012). Life prediction of different commercial dental implants as influence by uncertainties in their fatigue material properties and loading conditions. computer methods and programs in biomedicine, I08, 1277-1286. https://doi.org/10.1016/j.cmpb.2012.04.013

Su Bae, J., Jeong, H.Y. (2012). Effects of material properties and hole designs of the jig on the fatigue life of dental implants. Journal of Mechanical Science and Technology, 26 (3), 759-7668. https://doi.org/10.1007/s12206-011-1107-3

Yang, J., Xiang, H.J. (2007). A three-dimensional finite element study on the biomechanical behavior of an FGBM dental implant in surrounding bone. Journal of Biomechanics, 40 (11), 2377-2385. https://doi.org/10.1016/j.jbiomech.2006.11.019