Sigma J Eng & Nat Sci 38 (3), 2020, 1495-1507

Sigma Journal of Engineering and Natural Sciences
Sigma Mühendislik ve Fen Bilimleri Dergisi

Research Article

EFFECT OF DIFFERENT ABUTMENT MATERIALS ON STRESS DISTRIBUTION IN PERIPHERAL BONE AND DENTAL IMPLANT SYSTEM

Merve TERZİ[1], Yılmaz GÜVERCİN[2], Sabit Melih ATEŞ[3], Dursun Murat SEKBAN[4], Murat YAYLACI*[5]

[1]*Recep Tayyip Erdogan University, Department of Civil Engineering, RIZE;* ORCID: 0000-0002-9367-069X
[2]*Recep Tayyip Erdoğan University, Dept. of Orthopaed & Traumatol, RIZE;* ORCID: 0000-0003-1861-2083
[3]*Recep Tayyip Erdoğan University, Dept. of Prosthetic Dentistry, RIZE;* ORCID: 0000-0001-7137-2096
[4]*Karadeniz Technical University, Department of Marine Engineering Operations, TRABZON;*
ORCID: 0000-0002-7493-1081
[5]*Recep Tayyip Erdogan University, Department of Civil Engineering, RIZE;* ORCID: 0000-0003-0407-1685

Received: 17.06.2020 Revised: 06.07.2020 Accepted: 09.08.2020

ABSTRACT

In this study, the effect of different abutment materials on the stress distribution in bone tissue around the dental implant was investigated using ANSYS packet program based on finite element method. The study aims to examine the stresses, strains and deformations that will occur in the bone, dental implants and abutments with the replacement of abutment materials on the titanium dental implant. Modeling has been prepared in Solidworks program. Titanium was chosen as the dental implant material. Abutment materials were selected as titanium, zirconium, chrome-cobalt and polyetheretherketone. After modeling, necessary load was applied to the abutments and analyzed. This process was repeated for each abutment material. As a result of the analyzes, the stresses, strains and deformations that occurred in the bone, dental implants and abutments were combined into tables. In the conclusion, the differences between the abutment materials were evaluated. Generally, it was found that the lowest values were obtained in the group that chrome-cobalt was used as an abutment material. Therefore, chrome-cobalt can be preferred as an abutment material among all tested materials.
Keywords: Biomechanics, dental implant, finite element method.

1. INTRODUCTION

Since the existence of humanity, there has always been a research for new and artificial materials to replace different kinds of tissues because of the failure or loss of any part of the human body for various reasons. [1]. Similarly, endosteal dental implants are needed to restore the chewing system of patients suffering from tooth loss. With the development of techniques and materials used in dentistry, implants are used to rehabilitate cases of total, partial and single edentulism by using removable or fixed prosthodontic dentures [2-3].

Dental implants exhibit different biomechanical features compared to natural teeth because implants do not have periodontal ligaments and are in direct contact with surrounding bone.

* Corresponding Author: e-mail: murat.yaylaci@erdogan.edu.tr, tel: (464) 223 75 18 / 1219

Consequently, dental implants transfer functional chewing forces directly to the surrounding tissues [4]. These functional forces can be affected by the direction of the load, prosthetic superstructure materials, denture design, number of implants, macro and micro design of the implant and host bone characteristic [5]. Therefore, prosthetic design and material selection affects the stress distribution in implants and surrounding bone, which is one of the major factors determining implant su ccess. Most of the prosthetic superstructure materials such as implant abutments are made from titanium or chromium-cobalt alloys produced by using digital processing methods. However, the main disadvantage of these materials as an implant abutment material in implant dentistry is their grayish color which may cause aesthetic problems in areas with thin gingival tissue especially in the anterior region of the mouth [6]. To overcome such aesthetic problems polyetheretherketone (PEEK) polymer and zirconia based implant abutments which are having more tooth-like color than titanium become popular and their use in dental implantology are widespreading [1-2]. Therefore, biomechanical behavior of different types of implant abutment materials in dental implant system must be further investigated.

The finite element method is based on the principle of "going from piece to whole". The method can be defined as a form of solution that is achieved by dividing complex problems into simpler sub-problems and solving each problem in itself. With the development of technology over time, it has been used in medicine, orthopedics, aesthetic surgery, as well as various engineering fields such as machinery, construction, electricity, hydrodynamics, atoms, and spread to different fields [7]. The finite element method is a theoretical method used in mechanical system simulations to estimate the stresses that occur within the model. This computerized mathematical method is used as an important tool to evaluate these effects and to understand the biomechanical characteristics of implants [8]. Important preliminary information is obtained when planning treatment in dental implantology with finite element method. Many studies conducted to date have examined the amount of stress that occurs in the bone around the implant. Studies examining the amount of stress occurring in the implant and the bone around it with the finite element method are summarized below.

Jafarian et al. [9] examined the stress distribution around implants of different lengths and diameters with finite element analysis. Djebbar et al. [10] examined the stress distribution of the variable dynamic load in the dental implant using three-dimensional finite element analysis. Robau Porrua et al. [11] investigated the effect of the diameter, length and elasticity module in a dental implant on the stress and strain distribution in the implant bone by three-dimensional finite element analysis. The structural analysis of dental implants were examined by Karaman [12]. Tuzlalı et al. [13] compared the stress distribution in screw and cemented implant top fixed restorations using finite element analysis. Aksan et al. [14] gave general information about stress analysis techniques, finite element analysis used in dental research for a long time and their use in implant supported prostheses. Karabudak et al. [15] examined the stress and displacement distributions of implants placed on the lower jaw in a solid model made of titanium and zirconium materials. The effect of biomechanical markers and bone-implant interface properties on the stability of dental implants was investigated by Mathieu et al. [16]. Subaşı and Karataş [17] examined the general characterization of titanium and titanium alloys, application types, surface and interface properties of implants and various implant production techniques. Liu et al. [18] investigated the effect of the number of implants on the biomechanical behavior of latency times supported by a mandibular implant using three-dimensional finite element analysis. The effects of a quarter implant design in the mandible on implants and the surrounding bone have been explored by a three-dimensional finite element analysis by Deste and Durkan [19]. Different heights of prosthetic crowns supported by ultra-short implant were evaluated by three-dimensional finite element analysis by Elias et al. [20]. The effect of using four direct loaded implants in supporting a fixed prosthesis instead of two and secondarily the impact of straight and 208-angled abutment designs was investigated by Hasan et al. [21]. Batista et al. evaluated the influence of pontic and cantilever designs on 3-unit implant-retained prosthesis at maxillary posterior region verifying stress and strain distributions on bone tissue and stress distribution in abutments, implants and fixation screws, under axial and oblique loadings by finite element analysis [22]. Junior et al. examined to assess the effect of different dental implant designs, bone type, loading, and surface treatment on the stress distribution around the implant by using the finite element method [23].

When the studies conducted are examined, it is seen that many studies have been done about the stress distributions occurring in dental implants. On the other hand, although many different materials are used as the abutment material, it is seen that there is no study related to the stress, strain and deformation that will occur in the bone, dental implants and abutments after the change of these materials. In this context, in this study, Titanium, Chromium-Cobalt alloy, Zirconium and PEEK polymer were selected as the abutment materials and finite element based analyzes were performed.

2. MATERIAL METHOD

In this section, *a 3.9 mm* diameter prosthesis system is designed on bone tissue taken from a healthy person and this prosthesis is mounted. The geometry of the underlying system is introduced. The prosthesis and abutment system designed on the obtained bone tissue is connected under the first minor molar-second minor molar and first major molar teeth. Solidworks 2018 program was used to create the geometry. After the design of the three-dimensional model, the finite element method was used and analyzed. Stress, strain and deformation values were compared on the model by changing the selected material on the same model. In this study, ANSYS Workbench program (ANSYS, 2016) was used for finite element analysis [24].

The first step to be taken when starting finite element analysis; It is the creation of a three-dimensional model of the object to be worked on. While preparing these models, different imaging methods can be used depending on the complexity of the object. Computerized tomography and magnetic resonance imaging methods are examples. While transferring such images to the computer environment and creating the model, models can be created by scanning each detail of the object to be modeled with 3D scanners and transferring it to the computer environment or drawing the object by the researcher using three-dimensional modeling programs. Creating a mathematical model provides the researcher with easy controllability, test conditions, model parameters and geometry. The desired answers can be simulated through the mathematical model and test simulations can be repeated at any time. Therefore, a well-tested and validated mathematical model offers researches a very powerful tool for analysis [25].

The geometric model prepared in finite element analysis is divided into simple geometric infrastructures called element. Elements are classified according to certain features. As an example of these features, geometric shape; triangle, parallel edge, quadrilateral, size; one-dimensional, two-dimensional, three-dimensional and number of nodes can be given. Element types are shown in Figure 1. These elements are fully compatible with the geometry of the existing main structure. They show the desired mechanical properties in each region of the main structure.

After the desired model is transferred to the computer environment, division of the elements is called the creation of a network structure so that the model analysis can be made most realistically and reduced to a simpler model. Corner contact points between the elements are called knot points. In the second step; Data such as elasticity module, Poisson ratio of the material used, and the model that is formed and divided into elements are defined in the program. The boundary conditions of the object and where the force is applied to the object are defined during analysis. After entering this information, the analysis is performed by adjusting the direction, intensity and angle of the force for the loading conditions to be applied to the object. As a result of the analysis, after the analysis of the sub-elements of each object, the entire structure is analyzed. The result is reached by interpreting all the values obtained. These operations are done with computer programs. In the interpretation of the analysis made using the finite element method, the values that directly affect the results of the analysis are used. These; The geometry of the bone and implant, material properties, boundary conditions, force properties, properties of the implant-bone interface can be listed.

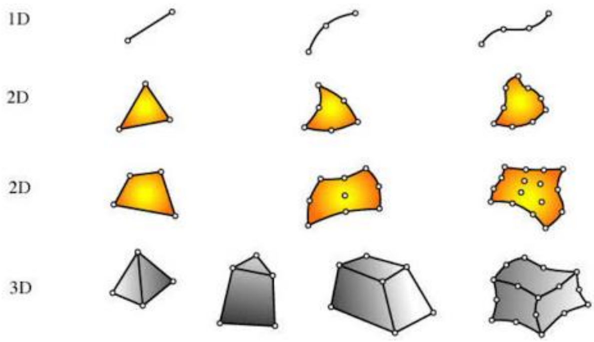

Figure 1. Finite element types (one-dimensional, two-dimensional and three-dimensional)

2.1. FINITE ELEMENT ANALYSIS

In this study, solid models of dental implants, abutments and mandible were drawn with the SolidWorks program. Then, these models were transferred to ANSYS Workbench software and analyzed.

In Figure 2, the bone tissue image obtained using a computerized tomography device was cleaned in a computer environment, and solid body was transferred to SolidWorks software by preserving three-dimensional coordinates and a bone tissue was designed in accordance with this image [26].

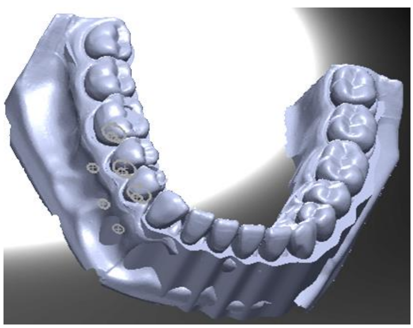

Figure 2. Mandible design

The abutments act as a fastener for dental implants. In this study, Nobel Active implants produced by Nobel Biocare company and supports suitable for implants were used. In Figure 3, the basis and dental implant design used in this study are given in SolidWorks software.

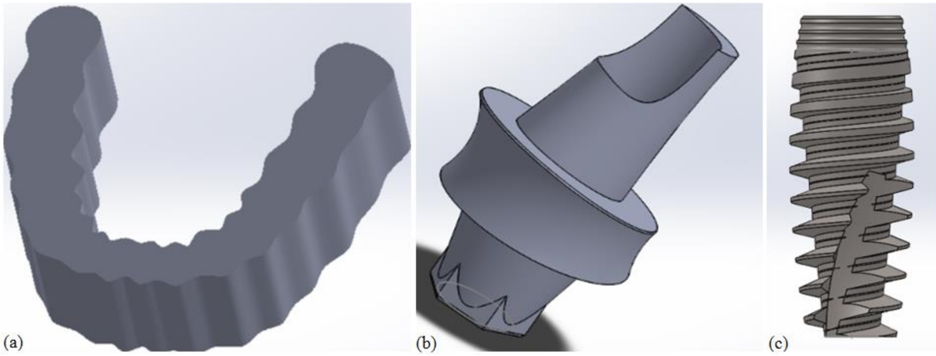

Figure 3. (a)Bone design, (b)Abutment design, (c) Dental implant design

After the solid model of each part was created, these parts were transferred to the assembly page. Bone, dental implants, abutments are combined by placing them in suitable positions for assembly. Firstly, the mandible and dental implants created in the SolidWorks software were transferred to the ANSYS Workbench program for analysis, as shown in Figure 4.

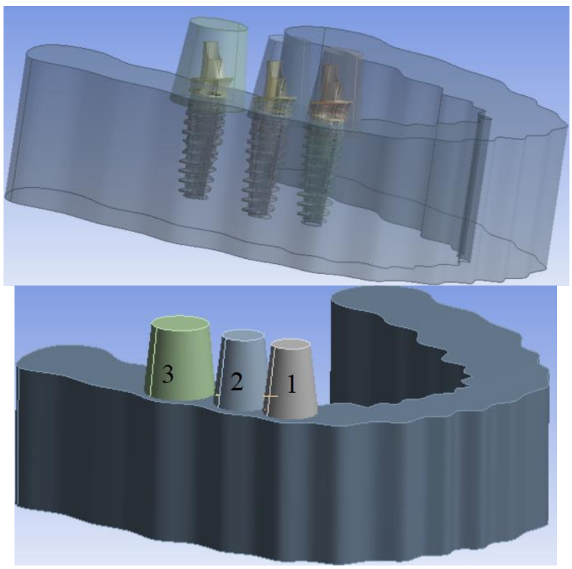

Figure 4. Importing geometry into ANSYS Workbench

Table 1. Material properties

Material	Elasticity Module (MPa)	Poisson Ratio	Literatures
Chrome-Cobalt (CoCr)	218000	0.33	[28]
Zirconium (Zr)	205000	0.22	[28]
Titanium (Ti)	110000	0.35	[29]
Polyetheretherketone (PEEK)	3500	0.36	[28]

Certain properties must be present in the material selected in the construction of dental implants. Required properties in the selected material; It can be listed as to be compatible with soft and hard tissues around it, be resistant to corrosion, be non-allergic, be able to form mechanical, functional, resist thermal stresses, be shaped, coated on the surface, be easy to build and apply, and be biocompatible [27].

In this study, four different materials were used. These materials are; It has been chosen as Titanium, Chrome-Cobalt, Zirconium, Polyether-Ether-Ketone (Table 1).

In Figure 5, the mesh structure of the solid model is given on ANSYS Workbench. After the mesh structure was created, 50 N load was applied on the abutments. 191622 elements and 321422 nodes are used in the model.

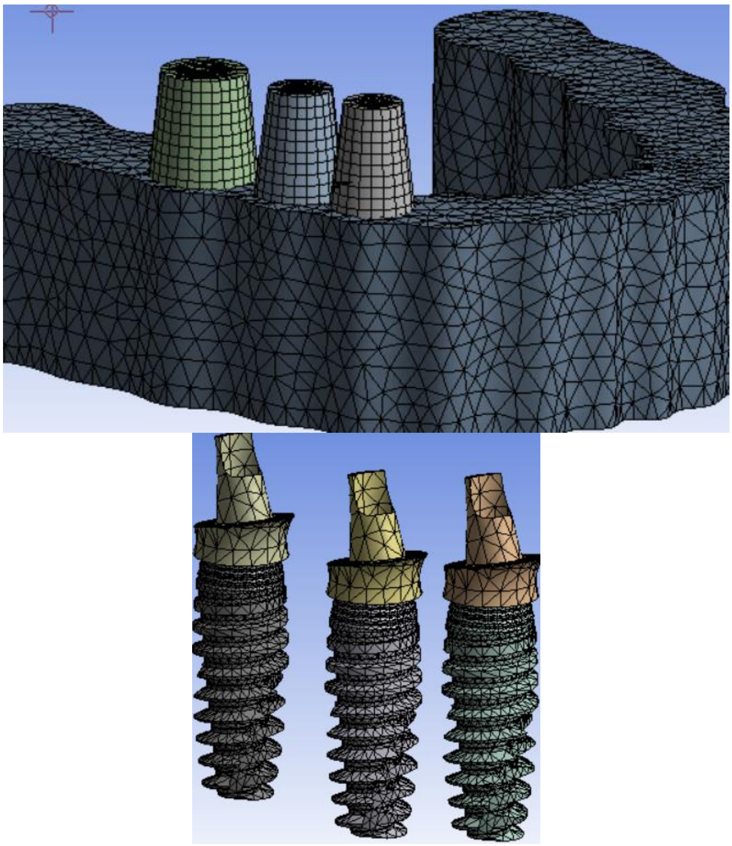

Figure 5. Mesh structure of the 3D

After all the necessary data such as boundary conditions and downloads are defined in the ANSYS Workbench software, the analysis phase has been started and the obtained results are described in the numerical results.

3. NUMERICAL RESULTS

Finite element method was used in this study to obtain stress, strain and deformation values in each region of the bone, dental implant and abutment. Titanium was used as a dental implant material on all models. Titanium, Zirconium, Chromium-Cobalt and Polyetheretherketone materials were chosen as the abutment material. The analysis images and results of the stress, strain and deformation in the bone when using CoCr, Zr, Ti and PEEK as abutment materials are shown in Figure 6 and Table 2, respectively. As can be seen from the table, the greatest values in the stress, strain and deformation amounts occurred in the bone when PEEK material was used as the abutment material. The reason for this is that PEEK material has the lowest modulus of elasticity among the 4 materials used as the abutment material. Similarly, the lowest values in stress, strain and deformation were obtained when CoCr material with the highest modulus of elasticity was used as an abutment material.

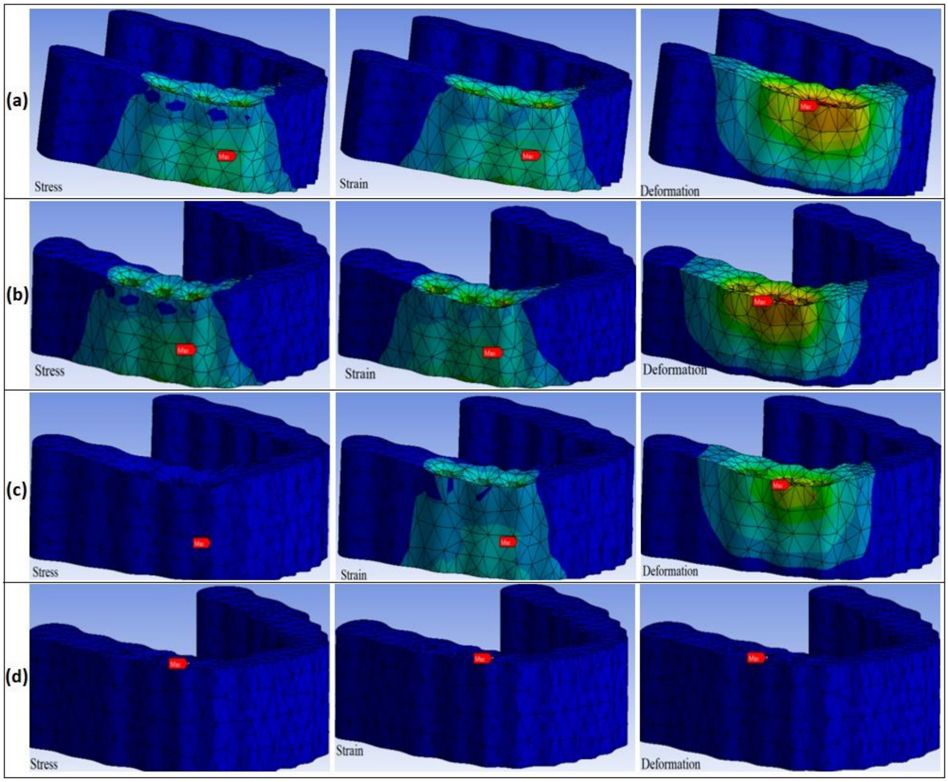

Figure 6. Analysis images of the stress, strain and deformation in the bone when using; a) CoCr, b) Zr, c) Ti and d) PEEK as the abutment material.

Table 2. Results of the stress, strain and deformation in the bone when using CoCr, Zr, Ti and PEEK as abutment material

Material	Max. Von-Mises Stress (MPa)	Max. Strain (10^{-4})	Max. Deformation (mm) (10^{-4})
Chrome-Cobalt	1.3952	1.0122	7.3662
Zirconium	1.3976	1.0140	7.4660
Titanium	1.3981	1.0144	7.4699
PEEK	1.4333	1.0940	7.8799

The analysis images and results of the stress, strain and deformation in the different position implants when using CoCr, Zr, Ti and PEEK as abutment material are shown in Figure 7 and Table 3, respectively. As can be seen, stress, strain and deformation values for all 3 implant values were obtained when PEEK was used as the abutment material. The reason that the highest values are obtained by using PEEK is that this material has a very low modulus of elasticity compared to the other 3 materials.

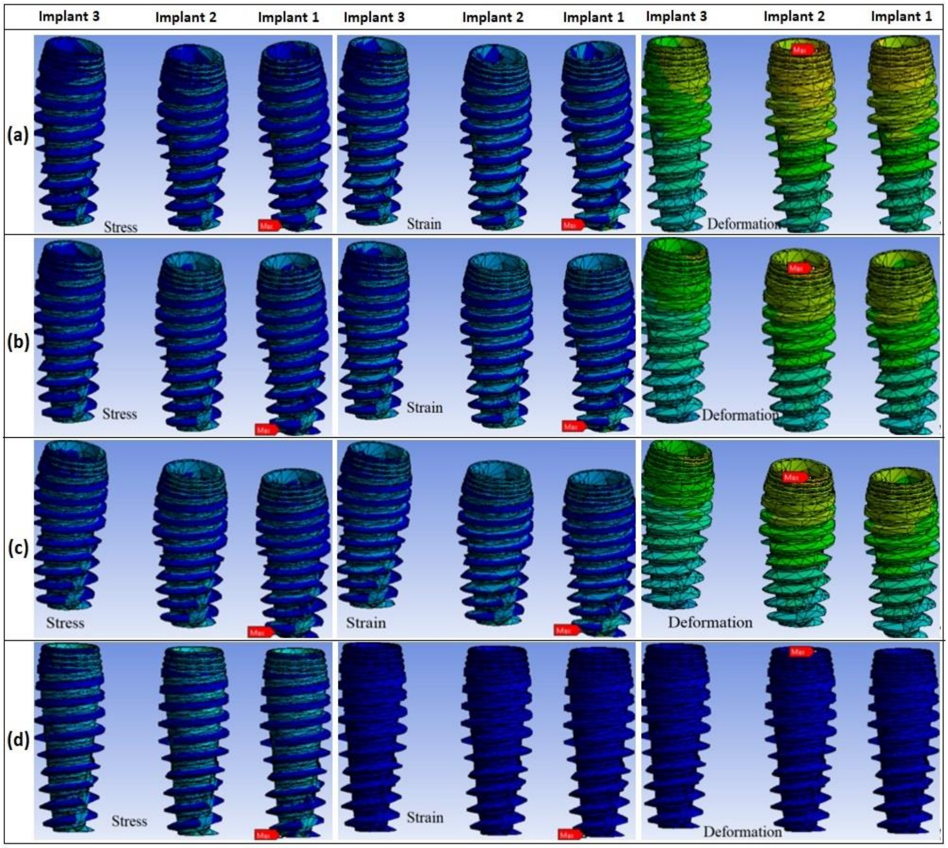

Figure 7. Stress, strain and deformation in the dental implant: (a)CoCr, (b)Zr, (c)Ti, (d)PEEK.

The analysis images and results of the stress, strain and deformation in the different position abutment when using CoCr, Zr, Ti and PEEK as abutment material are shown in Figure 8 and Table 4, respectively. As can be seen from the results, the highest values in stress, strain and deformation were achieved when used PEEK material with the lowest modulus of elasticity, while the highest values were achieved when CoCr material with the highest modulus of elasticity was used as the abutment material.

Table 3. Results of stress, strain and deformation in the different position implants when using CoCr, Zr, Ti and PEEK as abutment material

For Third Dental Implant			
Material	**Max. Von-Mises Stress (MPa)**	**Max. Strain (10^{-4})**	**Max. Deformation (mm)(10^{-4})**
Chrome-Cobalt	8.4159	0.7949	6.0387
Zirconium	9.5685	0.9081	6.0502
Titanium	9.6140	0.9130	6.1572
PEEK	10.2310	0.9742	6.5811
For Second Dental Implant			
Material	**Max. Von-Mises Stress (MPa)**	**Max. Strain (10^{-4})**	**Max. Deformation (mm)(10^{-4})**
Chrome-Cobalt	8.8579	0.8188	7.2957
Zirconium	8.8016	0.8145	7.3733
Titanium	8.8032	0.8147	7.3773
PEEK	10.3380	0.9622	7.7397
For First Dental Implant			
Material	**Max. Von-Mises Stress (MPa)**	**Max. Strain (10^{-4})**	**Max. Deformation (mm) (10^{-4})**
Chrome-Cobalt	16.0560	1.5297	6.9627
Zirconium	16.7060	1.5343	7.0537
Titanium	16.7090	1.5345	7.0567
PEEK	17.0300	1.5640	7.4305

Table 4. Results of the stress, strain and deformation in the different position abutments when using CoCr, Zr, Ti and PEEK as abutment material

For Third Dental Abutment			
Material	**Max. Von-Mises Stress (MPa)**	**Max. Strain (10^{-4})**	**Max. Deformation (mm)(10^{-4})**
Chrome-Cobalt	16.0880	0.8028	7.9698
Zirconium	12.2100	0.9189	8.1666
Titanium	10.5810	1.0424	7.4376
PEEK	9.4477	27.4950	96.6000
For Second Dental Abutment			
Material	**Max. Von-Mises Stress (MPa)**	**Max. Strain (10^{-4})**	**Max. Deformation (mm)(10^{-4})**
Chrome-Cobalt	18.7850	0.9345	9.2252
Zirconium	11.9250	1.1713	10.6080
Titanium	11.9320	1.1721	10.6140
PEEK	9.4323	27.4490	97.3290
For First Dental Abutment			
Material	**Max. Von-Mises Stress (MPa)**	**Max. Strain (10^{-4})**	**Max. Deformation (mm)(10^{-4})**
Chrome-Cobalt	17.3240	0.8733	8.2668
Zirconium	11.6300	1.1094	9.5786
Titanium	11.6700	1.1098	9.5823
PEEK	9.4481	27.4970	96.1230

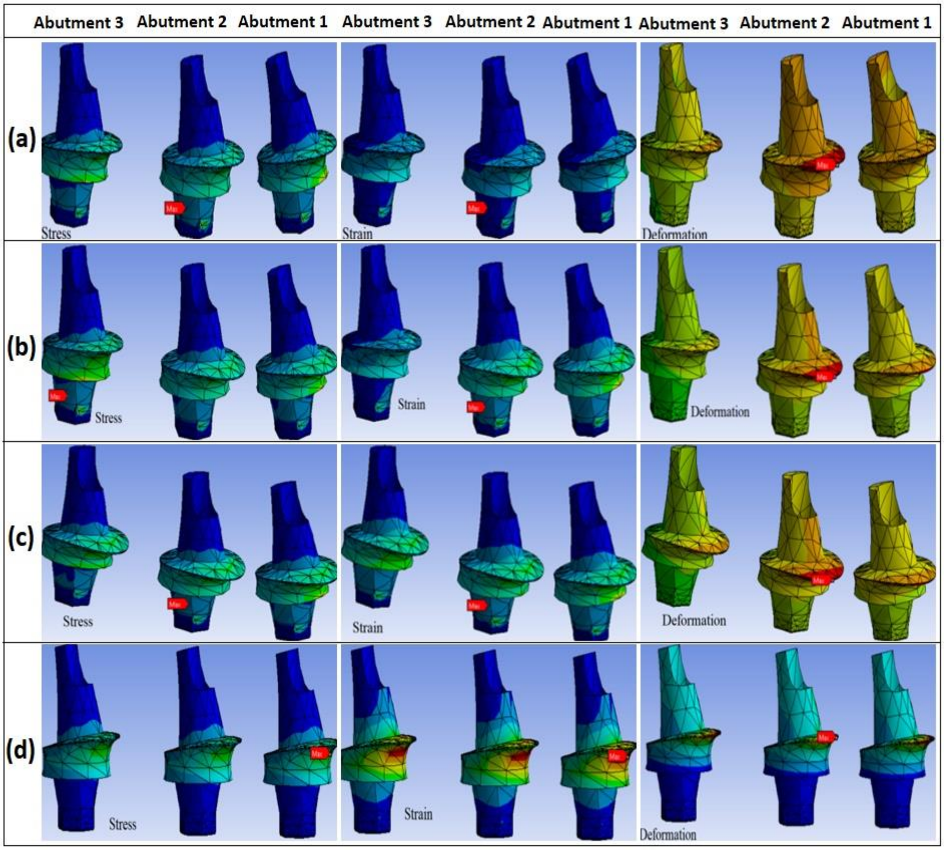

Figure 8. Stress, strain and deformation in the abutment: (a)CoCr, (b)Zr, (c)Ti, (d)PEEK.

When Table 5 is considered, it is seen that the stress values for bone are close to each other when using Chrome-Cobalt, Zirconium and Titanium as the abutment material, but it has a higher value when using PEEK. In dental implants, the stress values are found to be close to the second and third dental implants in all materials, but the stress values in the first dental implant are higher than other implants. Maximum von-Mises stress values for all abutments are obtained when using PEEK, while minimum stresses are obtained when using CoCr alloy.

As can be seen from the data in Table 5, deformation values were obtained when using PEEK as the basis for the highest value in bone and dental implants, and the lowest values when using CoCr. The results obtained when using Zirconium and Titanium abutments were close to each other. The maximum strain values for the abutments were obtained when PEEK was used as the abutment material.

When the numerical data in Table 5 are examined, the maximum deformation values are obtained when using PEEK as abutment material in bone, dental implants and abutments, while the lowest values are obtained in CoCr using as abutment. These values were found to be close to each other in Titanium and Zirconium abutment using.

When all the results are examined the lowest stress, strain and deformation results among the analyzed materials were generally seen in the CoCr alloy. The reason for this is that this material has the highest modulus of elasticity among the materials analyzed. According to the results obtained, it is possible to say that the most suitable of these 4 materials as a prosthetic material is the CoCr alloy.

Table 5. Stress, strain and deformation values occurred in the bone, implant and abutment when different abutment materials are used.

Maximum Von-Mises stress values (MPa)				
	CoCr	**Zr**	**Ti**	**PEEK**
Bone	1.3952	1.3976	1.3981	**1.4333**
1.Dental implant	16.0560	16.7060	16.7090	**17.0300**
2.Dental implant	8.8579	8.8016	8.8032	**10.3380**
3.Dental implant	8.4159	9.5685	9.6140	**10.2310**
1.Abutment	**17.3240**	11.6300	11.6700	9.4481
2. Abutment	**18.7850**	11.9250	11.9320	9.4323
3. Abutment	**16.0880**	12.2100	10.5810	9.4477
Maximum strain values (10^{-4})				
	CoCr	**Zr**	**Ti**	**PEEK**
Bone	1.0122	1.0140	1.0144	**1.0940**
1.Dental implant	1.5297	1.5343	1.5345	**1.5640**
2.Dental implant	0.8188	0.8145	0.8147	**0.9622**
3.Dental implant	0.7949	0.9081	0.9130	**0.9742**
1.Abutment	0.8733	1.1094	1.1098	**27.4970**
2. Abutment	0.9345	1.1713	1.1721	**27.4490**
3. Abutment	0.8028	0.9189	1.0424	**27.4950**
Maximum deformation values ($mm10^{-4}$)				
	CoCr	**Zr**	**Ti**	**PEEK**
Bone	7.3662	7.4660	7.4699	**7.8799**
1.Dental implant	6.9627	7.0537	7.0568	**7.4305**
2.Dental implant	7.2957	7.3733	7.3773	**7.7397**
3.Dental implant	6.0387	6.0502	6.1572	**6.5811**
1.Abutment	8.2668	9.5786	9.5823	**96.1230**
2. Abutment	9.2252	10.6080	10.6140	**97.3290**
3. Abutment	7.9698	8.1666	7.4376	**96.6000**

4. CONCLUSION

In this study, the stress, strain and deformation distributions caused by replacing the abutment materials in the dental implant supported prosthesis were examined. A model with three implants was designed as the implant layout. The model obtained was subjected to stress tests with ANSYS Workbench finite element software. The results obtained are described below.

It has been observed that the maximum von-Mises stress, strain and deformation values for bone tissue occur in PEEK, while the minimum values occur in the Chromium-Cobalt alloy.

Considering the results of titanium dental implants, the greatest stress, strain and deformation values were seen in PEEK; the smallest stress, strain and deformation values were observed in the Chrome-Cobalt alloy. The results were similar to the evaluation of bone tissue.

For Abutment, CoCr received the highest stress values while PEEK received the lowest values. The Elasticity Modules of the materials affected the stress values on the abutment.

When the maximum strain and maximum deformation results in the abutments were examined, all maximum values were seen in PEEK; all minimum values were observed in Zirconium. Volume change is associated with the Poisson ratio. As the Poisson ratio increases, the material stiffness increases. This view supports the results obtained.

REFERENCES

[1] Ring, M.E., (1995) A Thousand Years of Dental Implants: A Definitive History- Part 1, Compendium Continuing Education Dental 16, 1060-1069.

[2] Kaleli, N., Sarac, D., Külünk, S. and Öztürk, Ö., (2018) Effect of Different Restorative Crown and Customized Abutment Materials on Stress Distribution in Single Implants and Peripheral Bone: A Three-Dimensional Finite Element Analysis Study, J Prosthet Dent.119(3), 437-445.

[3] Tekin, S., Değer ,Y. and Demirci, F., (2019) Evaluation of the Use of PEEK Material in Implant-Supported Fixed Restorations by Finite Element Analysis, Niger J Clin Pract 22, 1252-1258.

[4] Bankoğlu Güngör, M. and Yılmaz, H., (2016) Evaluation of Stress Distributions Occurring on Zirconia and Titanium Implant-Supported Prostheses: A Three-Dimensional Finite Element Analysis, J Prosthet Dent. 116(3), 346-55.

[5] Pesqueira, A.A., Goiato, M.C., Filho, H.G., Monteiro, D.R., Santos, D.M., Haddad, M.F., et al. (2014) Use of Stress Analysis Methods to Evaluate the Biomechanics of Oral Rehabilitation with Implants, J Oral Implantol 40, 217-228.

[6] Fuh, L.J., Hsu, J.T., Huang, H.L., Chen, M.Y. and Shen, Y.W., (2013) Biomechanical Investigation of Thread Designs and Interface Conditions of Zirconia and Titanium Dental Implants with Bone: Three-Dimensional Numeric Analysis, Int J Oral Maxillofac Implants 28(2), 64-71.

[7] Fish, J. and Belytschko T., (2007) A First Course in Finite Elements. USA. 319s., 15-23.

[8] Damlar, İ., Özyılmaz, E., Altan, A. and Özyılmaz, E., (2014) Üç Boyutlu Sonlu Eleman Analiz Yöntemiyle İki Ticari Implant Sisteminin Gerilme Dağılımlarının İncelenmesi, Süleyman Demirel Üniversitesi Mühendislik Bilimleri ve Tasarım Dergisi 2(3), 175-180.

[9] Jafarian, M., Mirhashemi, F.S. and Emadi, N., (2019) Finite Element Analysis of Stress Distribution Around a Dental Implant with Different Amounts of Bone Loss: An in Vitro Study, Dental and Medical Problems 56(1), 27-32.

[10] Djebbar, N., Serier, B. and Bouiadjra, B.A.B., (2018) Stress Distribution of The Variable Dynamic Loading in the Dental Implant: A Three-Dimensional Finite Element Analysis, Journal of Biomimetics, Biomaterials and Biomedical Engineering 31, 44-52.

[11] Robau Porrua, A., Perez Rodriquez, Y., Soris Rodriquez, L.M. and Perez Acosta O., (2020) The Effect of Diameter, Length and Elastic Modulus of a Dental Implant on Stress and Strain Levels in Peri-Implant Bone: A 3D Finite Element Analysis, Bio-Medical Materials and Engineering 30, 541-558.

[12] Karaman, H., (2018) Dental Implantların Yapısal Analizi, Master Thesis. Uludağ Üniversitesi, Fen Bilimleri Enstitüsü, Bursa, Türkiye, 117s.

[13] Tuzlalı, M., Öztürk, C. and Zortuk, M., (2018) Vidalı ve Simante Implant Üstü Sabit Restorasyonlarda Stres Dağılımının Karşılaştırılması: Sonlu Elemanlar Analizi Çalışması, Türkiye Klinikleri Journal of the Dental Sciences 24(2), 106-113.

[14] Aksan, M.E., Atsü, S. and Bulut, A.C., (2018) İmplant-Protez Bağlantısında Sonlu Eleman Analiz Yöntemi, Atatürk Üniviversitesi Diş Hekimliği Fakültesi Dergisi 28(1), 91-97.

[15] Karabudak, F., Zamanlou, H., Yeşildal, R., Bayındır, F. and Şen, S., (2014) Düz ve Açılı Abutmentlere Sahip Titanyum ve Zirkonyum Dental Implantların Gerilme Analizlerinin Karşılaştırılması. Mühendislik ve Makina 55(652), 34-42.

[16] Mathieu, V., Vayron, R., Richard, G., Lambert, G., Naili, S., Meningaud, J.P. and Haiat, G., (2014) Biomechanical Determinants of The Stability of Dental Implants: Influence of The Bone–Implant Interface Properties, Journal of Biomechanics 47, 3-13.

[17] Subaşı, M. and Karataş, Ç., (2012) Titanyum ve Titanyum Alaşımlarından Yapılan Implantlar Üzerine Inceleme, Politeknik Dergisi 15(2), 87-103.

[18] Liu, J., Pan, S., Dong, J., Mob, Z., Fan, Y. and Feng, H., (2013) Influence of Implant Number on The Biomechanical Behaviour of Mandibular Implant-Retained/Supported Overdentures: A Three-Dimensional Finite Element Analysis, Journal of Dentistry 41, 241-249.
[19] Deste, G. and Durkan, R., (2020) Effects of all-on-Four Implant Designs in Mandible on Implants and the Surrounding Bone: A 3-D Finite Element Analysis, Nigerian Journal of Clinical Practice 23(4), 456-463.
[20] Elias, DM., Valerio, CS., Oliveria, DD., Manzi, FR., Zenobio, EG and Seraidarian, PI., (2020) Evaluation of Different Heights of Prosthetic Crowns Supported by an Ultra-Short Implant Using Three-Dimensional Finite Element Analysis, Fundamental Research 33(1), 81-90.
[21] Hasan, I., Bourauel, C., Keilig, L., Reimann, S. nd Heinemann, F., (2011) The influence of implant number and abutment design on the biomechanical behaviour of bone for an implant-supported fixed prosthesis: a finite element study in the upper anterior region, Computer Methods in Biomechanics and Biomedical Engineering 14(12), 1113-1116.
[22] Batista, V.E.S., Verri, F.R., Almeida, D.A.F., Junior, J.F.S., Lemos, C.A.A. and Pellizzer, E.P., (2017) Finite element analysis of implant-supported prosthesis with pontic and cantilever in the posterior maxilla, Computer Methods in Biomechanics and Biomedical Engineering 20(6), 663-670.
[23] Junior, J.F.S., Verri, F.R., Almeida, D.A.F., Batista, V.E.S., Lemos, C.A.A. and Pellizzer, E.P., (2017) Finite element analysis on influence of implant surface treatments, connection and bone types, Materials Science and Engineering C 63, 292–300.
[24] ANSYS 16.0, (2016). Swanson Analysis Systems Inc., Houston PA, USA.
[25] Menicucci, G., Mossolav, A., Mozzati, M., Lornzetti, M. and Preti, G., (2002) Tooth-Implant Connection: Some Biomechanical Aspects Based on Finite Element Analysis, Clinical Oral Implants Research 13, 334-341.
[26] Solidworks 2018, (2018). Dassault Systèmes Solidworks Corporation. Waltham MA, USA.
[27] Sauk, J.J., O’Neal, R.B., and Someman, M.J., (1992) Biological Requirements for Material Integration, Journal of Oral Implants 18, 243-255.
[28] Callister, W.D. and Rethwisch, D.D., (2013) Materials Science and Engineering. Nobel Akademik Yayıncılık, 8. Baskı, ISBN: 978-605-133-418-9, 992p.
[29] Colling, E.W., (1984) The Physical Metallurgy of Titanium Alloys, Metals Park (OH), American Society for Metals. 65-72.