Erciyes Üniversitesi
Fen Bilimleri Enstitüsü Dergisi
Cilt 38, Sayı 2 , 2022

Erciyes University
Journal of Institue Of Science and Technology
Volume 38, Issue 2, 2022

Design of Neural Network Predictor for A Finite Element Analysis of Thermal and Mechanical Stresses Caused in the Jawbone of Implantology

Buket TURGUT [*1], **Aslı DURMUŞOĞLU** [2], **Çağlar SEVİM** [3] ,**Necdet ALTUNTOP** [4]

[*1,4] Erciyes Üniversitesi Mühendislik Fakültesi Makine Mühendisliği, KAYSERİ
[2] Hakkari Üniversitesi Mühendislik Fakültesi Makine Mühendisliği, HAKKARİ
[3] Niğde Ömer Halisdemir Üniversitesi Mühendislik Fakültesi Makine Mühendisliği, NİĞDE

(Alınış / Received: 18.04.2022, Kabul / Accepted: 31.05.2022, Online Yayınlanma / Published Online: 23.08.2022)

Keywords
Implantology,
Thermal stress,
Mechanical stress,
Finite element analysis,
Neural predictor

Abstract: As implant cavity preparation and healthy bone are essential for primary healing, prevention of mechanical and thermal damage, which occurs at bone, is crucial importance. So heat generation must be kept under control while preparing implant cavity. In this study, thermal and mechanical stresses were aimed to predict with computer software while drilling implant cavities. Thermal and mechanical stress analysis that arises around the jawbone while drilling implant cavities by using plot drills of four types of drills with a standard method was conducted by using Finite Elements Method (FEM). The real time estimation was made using data obtained from numerical analysis. The estimation analysis was performed with Adaptive-Network Based Fuzzy Inference Systems (ANFIS) approach that have a robust and adaptive structure. The results from the ANFIS simulation, the ability to show the same action as the results obtained with the Finite Element Method demonstrates that neural predictor can be used to estimate the system's thermal and Von- Mises stresses parameters. This presentation shows the estimation of thermal and mechanical stresses changes during implant surgery with artificial intelligence.

İmplantolojinin Çene Kemiğinde Neden Olduğu Termal ve Mekanik Gerilmelerin Sonlu Elemanlar Analizi için Yapay Sinir Ağı Tasarımı

Anahtar Kelimeler
İmplantoloji,
Termal gerilme,
Mekanik gerilme,
Sonlu elemanlar metodu,
Yapay sinir ağı tahmincisi

Öz: Primer iyileşme için implant yuvasının hazırlanması ve sağlıklı kemik gerekli olduğundan, kemikte oluşan mekanik ve ısıl hasarın önlenmesi kritik önem taşır. Bu nedenle implant yuvası hazırlanırken ısı oluşumu kontrol altında tutulması gerekmektedir. Bu çalışmada, implant kaviteleri hazırlanırken oluşan termal ve mekanik gerilmelerin bilgisayar yazılımı ile tahmin edilmesi amaçlanmıştır. standart yöntemle dört çeşit plot delici uç kullanılarak implant yuvası açılması esnasında çene kemiği çevresinde oluşan termal ve mekanik gerilme analizi Sonlu Elemanlar Metodu (FEM) kullanılarak yapılmıştır. Sayısal analizden elde edilen veriler kullanılarak gerçek zamanlı tahmin yapılmıştır. Tahmin analizi, sağlam ve uyarlanabilir bir yapıya sahip olan Uyarlamalı Ağ Tabanlı Bulanık Çıkarım Sistemleri (ANFIS) yaklaşımı ile yapılmıştır. ANFIS simülasyonundan elde edilen sonuçlar, Sonlu Elemanlar Metodu ile elde edilen sonuçlarla aynı davranışı gösterebilme yeteneği, sinirsel tahmincinin sistemin termal ve Von-Mises gerilme parametrelerini tahmin etmek için kullanılabileceğini göstermektedir. Bu çalışma, implant cerrahisi esnasındaki mekanik ve termal gerilme değişimlerinin yapay sinir ağları ile tahminini göstermektedir.

*Corresponding Author, email: bcinar@erciyes.edu.tr

1. Introduction

Dental implants applied for aesthetic and retrieving of lost functions in case of one or more missing teeth are quite popular at the present time. The term of osseointegration, which is necessary for a successful implant treatment, was first defined by Branemark et al. [1] The use of dental implants for restoration of lost teeth has significantly increased after the concept of the osseointegration. Achievement of osseointegration depends on many factors. The most important factor is the primary healing of the implant location. For this, thermal damage to the bone must be minimized [2]. Drilling tips used in preparing implant cavity can damage to the bone tissue due to the high temperature. It can also cause causing mechanical damage. As the heat transmission of bone is low, heat generation in the location where implant cavity is drilled cannot be distributed quickly [3]. While preparing the implant hole, heat generates in the bone due to the friction of the drills against the bone. As a result of this heat generation, necrosis, fibrosis and osteolytic degeneration may occur in the bone. These degenerations may cause implants may to be lost in the necrotic area. Studies have shown that if the temperature during preparation exceeds 47 °C in 1 minute, degeneration of fat cells and resorption of the bone that will come comes into direct contact with the implant increase. Eriksson and Alberktsson [4] investigated the susceptibility of lamellar bone pending heat generation on live rabbit bone. In their microscopic investigation, they showed that sub-bone resorption and fat cell degeneration started at temperatures above 47 °C [5]. The heat generated in the bone can be minimized by using a sharp drill bit, adjusting the speed, and by adequate external or internal cooling [6]. Shraway et al. [7] investigated the effects of thermal energy released during drilling implant cavity on bone healing and maximum temperature values that bone can bear without occurrence of necrosis. Sener et al. [8] reported in their study that thermal energy released during drilling has major influence on liveliness of cells and implant osseointegration. For this purpose, they created 12x6 cm rectangular cross-section pieces through lower jawbones of frozen toothles toothless cattle. They placed thermo resistors with depth of 3.7 and 12 mm into these pieces. A total of seven drilling tip groups with different surface properties were investigated in the study. Consequently, the lowest temperature increase was observed in boron nitride group. Nieri et al. [9] investigated the factors affecting the clinical approach to the affected maxillary canines by Bayesian network analysis in their study. Data were collected by comparing before and after treatment .Patient-related quantitative variables, metric variables, and nominal variables were collected, and the causal relationships between the variables were found through this analysis. This study supports that Artificial Intelligence (AI) could be used to assist dentists in decision making and that the possibility of substitution is quite high. Miladinovic et al. [10] applied data mining to analyze the indirect reason for the extraction based on large volumes of electronic medical records. According to the result of data mining of the electronic medical records of the selected subjects with a specific Artificial Intelligence (AI) algorithm, it was confirmed that the closeness of the extraction experience and the number of teeth extracted were statistically affected by gender, age, and occupation. Sukegawa et al. [11] used panoramic X-ray images to classify and clarify the correctness of different dental implant brands via deep convolutional neural networks (CNNs) with transfer-learning strategies. Turgut [12], Turgut et al. [13] investigated thermal stress and mechanical stress analysis that occurs around the jawbone during implant surgery by using drills of four different componies with a classical method without cooling was conducted in ANSYS programme by using FEM. The study shows that the heat measured while preparing implant cavity is about 65°C with the first model 58°C with the second model, 62°C with the third model, 64°C with the fourth model for 600 rpm. For 800 rpm; the temperature occurred was about 68°C for the first model, about 59°C for the second model, about 64°C for the third model, about 66°C for the forth model. Stress analysis is difficult to investigate in living tissue. Therefore, stress analysis studies are performed on a specific model with various methods. One of these methods is the FEM which is the most preferred method because of its advantages. Some of these advantages are as follows: complex geometries can be modeled, realistic model can be created with realistic material acceptances, stress distribution and strain can be obtained sensitively. Analysis can be performed effortlessly by changing boundary conditions, material properties and geometry [14]. Therefore, the FEM is used in many areas from vibration to impact. In this context Kosedag ve Ekici [15] numerically investigated free vibration analyzes of aluminum-foam sandwich structures. They performed the analyzes using the ABAQUS/Standard finite element package program. Ekici ve Kosedag [16] investigated the reinforcement effect of particle-reinforced metal matrix composites on the low-velocity impact behaviors. They used FEM to model and perform the low velocity impact analyzes of SiC and pumice particle reinforced aluminum 6061 matrix composites.

The novelty of this paper is to assist physicians in pre-diagnosis in implant surgery by using the stable and adaptive structure of the ANFIS approach. The purpose of this study is to investigate thermal and mechanical stresses caused by the heat generated during the drilling process. Numerical methods were used in the studies. The real time estimation of these stress parameters was analyzed with ANFIS approach. Thus, before the implant surgery is performed, necessary precaution can be taken by predicting the thermal stresses that may occur in the jawbone.

2. Materials and Methods

2.1. Finite Elements Model

Four different specially produced drilling tips, which are frequently used in implant surgery and therefore differ in geometries and materials and jawbone were modelled in the 3D solid model program. Jawbone was modelled in a cylindrical shape in order to ensure its mesh structure to be regular. The jawbone; in accordance with human anatomy; was modelled in two-stages consisting of 3 mm- thick cylindrical structured cortical bone layer and 12 mm- thick trabecular bone layer. The connection type between these two bone tissues was applied as bonded. In the jawbone, one of the elasto-plastic material models -piecewise-linear-plasticity- MAT24 material model was used and a rigid MAT20 material model was used for the drills. During the analyzes, movement of the jawbone in all axes is limited. The drill was movable in the z-axis direction. Both displacement and rotation of the drill in the x-y axis are limited. In this study, the eroding surface to surface contact algorithm was used. Solid models of drills, jawbone model and mesh structures have been given in the Figure 1. The geometric properties of drills have been given in Table 1.

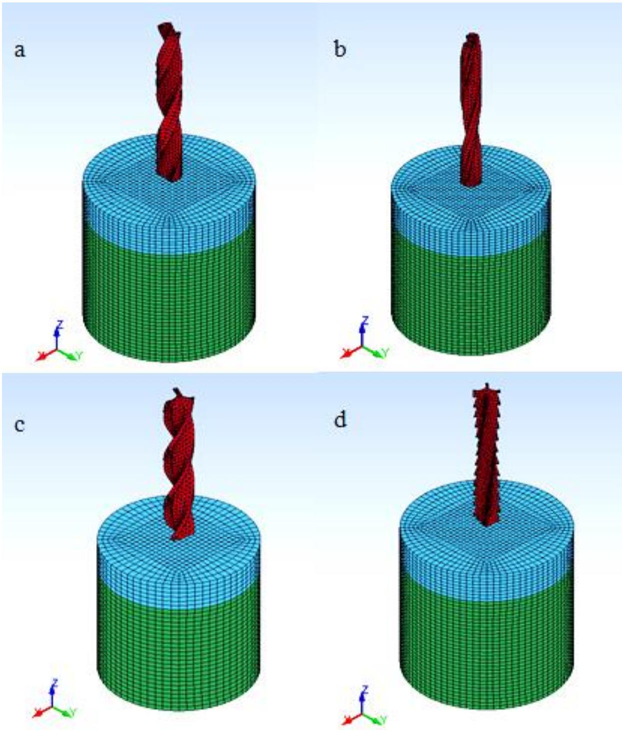

Figure 1. Mesh structures of jaw bone and drills a) First model, b) Second model, c) Third model, d) Fourth model

Table 1. The geometric properties of drills

	Materials	Diameter (mm)	Height (mm)	Angle (°)
First Model	Stainless Steell	2.2	17	60
Second Model	Tungsten Carbid Coating	2	18	60
Third Model	Titanium Nitride Coating	2	16.5	60
Forth Model	Titanium Nitride Coating	2.3	15	60

Mesh structures of drilling tips' solid models were formed using ANSYS - Mechanical APDL software. Hardware unit specifications used for analysis; It has 1.6 Ghz Intel i7 processor, 16 GB ram and 2 GB graphics card. Solid 186 element type is used in this study, the size of the element was determined as 0.50 mm. There are approximately 1.8 million elements in numerical models. Materials and tissues used in this study exhibit different mechanical and physical properties. The characteristics of the materials used in the study have been given in Table 2.

Table 2. The characteristics of the materials

Materials	Young's Modulus E (MPa)	Fail Parameter	Poison Ratio ν	Density ρ (kg/m³)	Friction Coefficient f_s	Thermal Conductivity k (W/m°K)	Ultimate Strenght σ_u(MPa)	Yield Stress σ_y(MPa)	Specific heat °C (J/kg°K)
Cortical Bone	13700 [17]	2,23 [17]	0,3 [17]	1100 [18]	0,35 [19]	0,56 [20]	250 [24]	122 [17]	1700 [18]
Trabecular Bone	1370 [17]	0,62 [17]	0,3 [17]	1800 [22]	0,3 [19]	0,3 [22]	194,6 [25]	162,5 [23]	1695 [21]
First Model	193000 [26]	-	0,29 [26]	7900 [26]	0,15 [26]	16,2 [26]	505 [26]	215 [26]	500 [28]
Second Model	420000 [27]	-	0,28 [27]	14800 [27]	0.95 [27]	163,3 [27]	620 [27]	550 [24]	134 [24]
Third Model	600000 [24]	-	0.25 [24]	5220 [24]	0.65 [24]	19 [24]	220 [24]	140 [75]	522 [28]
Forth Model	600000 [27]	-	0.25 [27]	5220 [27]	0.65 [27]	19 [27]	220 [27]	140 [27]	522 [27]

In order to demonstrate the post-yield behavior of the material, stress-strain data for both the cortical bone and the trabecular bone were taken from the literature. Stress strain diagram for cortical bone in Figure 2 and stress strain diagram for trabecular bone in Figure 3 are given.

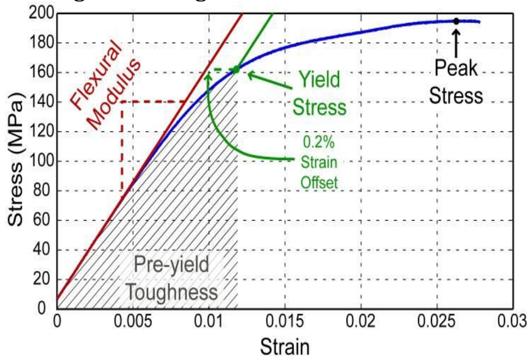

Figure 2. Stress and Strain diagram for cortical bone [30]

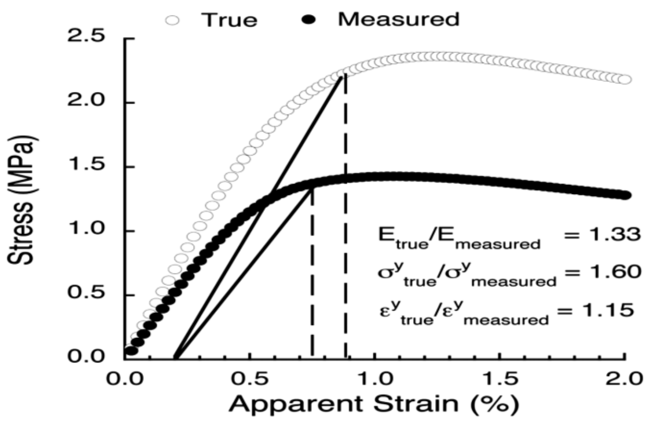

Figure 3. Stress and Strain diagram for trabecular bone [31]

2.2. Adaptive-Network Based Fuzzy Inference Systems (ANFIS) Model

ANFIS integrates the neural network with the Fuzzy Interface System (FIS). In Figure 4 the process flow sheet of learning and predicting stage of simulation study was given below. The method of ANFIS composes of 3 elements: a rule base, a database and a reasoning mechanism. A schematic representation of an ANFIS is outlined in Figure 5 [29].

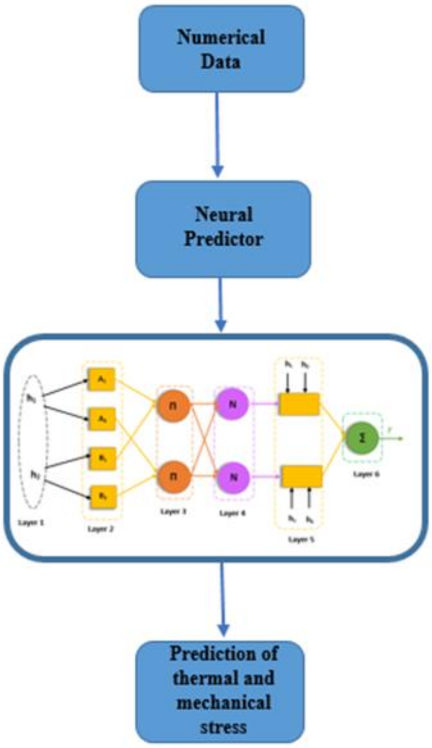

Figure 4. The flow chart of methodology used in the study

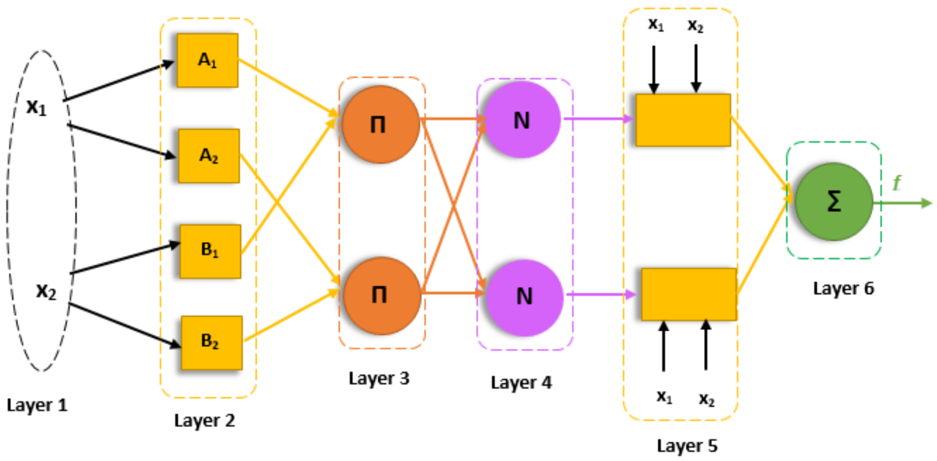

Figure 5. ANFIS Structure

Two different models were created in this study to define the fuzzy inference system. It is used two inputs as h_1 and h_2 (depth) for each model. Output values for two models are temperature (T) and stress (σ) respectively. The rule base contains two fuzzy if-then rules and they can be depicted such as:

Rule 1: if h_1 is A_1 and h_2 is B_2 then $T_1 = a_1h_1 + b_1h_2 + s_1$

Rule 2: if h_1 is A_2 and h_2 is B_2 then $T_2 = a_2h_1 + b_2h_2 + s_2$

where A_i and B_i are fuzzy membership sets, a_i, b_i and s_i are the design parameters which are defined during the train process the number of membership equations [29]. The ANFIS structure consist of six layers:

Layer 1: This is the input layer that determines actual data and desired data.

Layer 2: Every node in this layer is an adaptive node with a fuzzy membership equation. Node outputs for two inputs:

$$L_i^1 = \alpha\theta_i(h) \qquad i = 1,2 \tag{1}$$

$$L_i^1 = \alpha\beta_i(h) \qquad i = 1,2 \tag{2}$$

where αθ i and αβ i are membership functions.

$$\alpha\theta_i(h) = \frac{1}{1 + \left[\left(\frac{h-p}{r_i}\right)^2\right]} \tag{3}$$

where {p i , r i ,t i } is the coefficient group.

Layer 3: Each node in the third layer is a circle node called "Ŋ" which amplifies all the signals and sends out the product.

$$\omega_i = \alpha\theta_i(h).\alpha\beta_i(h) \qquad (\ i = 1,2 \ldots) \tag{4}$$

Layer 4: Each nodal in the fourth layer is a circle nodal called "N".

$$\bar{\omega}_i = \frac{\omega_i}{\omega_1 + \omega_2} \tag{5}$$

Layer 5: In this layer, each nodal i has the following function:

$$L_i^5 = \bar{w}_i.f_i = \bar{w}_i(a_i h_1 + b_i h_2 + s_i) \tag{6}$$

Layer 6: The single nodal in the sixth layer is a circle nodal called "∑".

$$L_i^6 = \sum \bar{\omega}_i \ f_i = \frac{\sum \omega_i f_i}{\sum \omega_i} \tag{7}$$

The data obtained from numerical analysis, they are divided into training data (80%) and testing data (20%). ANFIS is used for prediction the data in training data and testing data. Nonlinear parameters *(a.b)* and linear parameters *(p, r, t)* of ANFIS are optimized. The used ANFIS parameters are as follows:
Maximum epoch : 30
The number of membership function : 2 (input 1), 2 (input 2)
The number of rules : 2
The number of linear parameter: 12
The number of nonlinear parameter: 8
Maximum iteration : 500.

3. Simulation Results

3.1. Findings for Finite Elements Analysis

Initially, 20 Nm torque and 600 rpm rotational speed were applied to the drilling tip during drilling. 30 Nm torque and 800 rpm rotational speed were applied in the second case. The temperature of the jawbone was considered 36.5 ° C of body temperature. The cortical bone layer and trabecular bone layer were immobilized assuming jawbone to be motionless.

3.1.1. For ω=600 rpm The Occurring Temperature and Stress Changes

As it is seen in Figure 6 and 7 maximum temperature change which occurred in the jawbone was observed in the model with small bevel angle and titanium nitride coating. This is because its pitch structure has a small angle and heat transmission coefficient of the coating material is less. The reason why minimum temperature increase was observed in the model with saw tip-shaped pitch structure is that it has mainly a pitch structure and heat transmission coefficient of its tungsten carbide coating was higher as compared to other drilling tips.

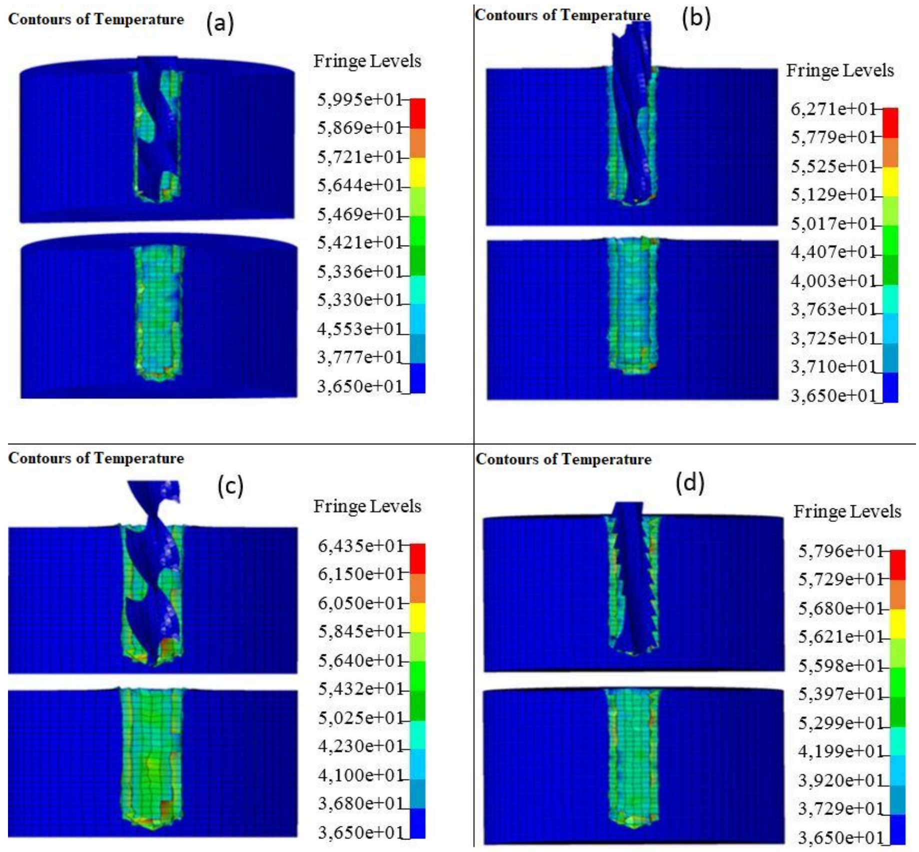

Figure 6. Temperature distributions in trabecular bone for; a) First model, b) Second model, c) Third model, d) Fourth model

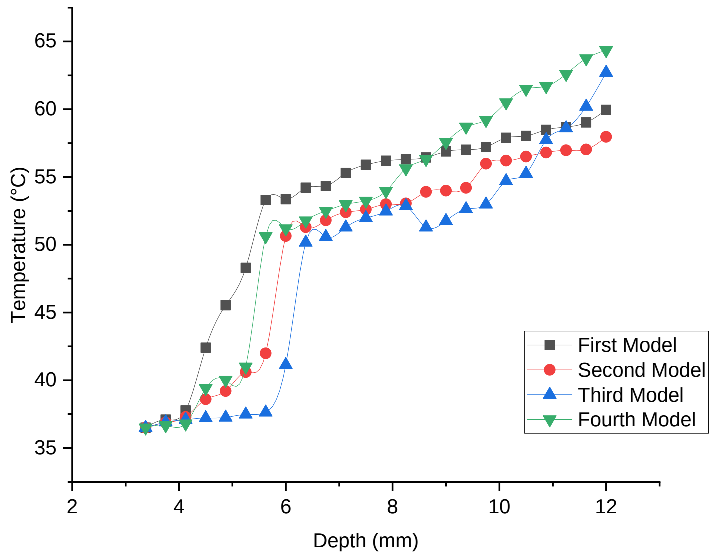

Figure 7. Temperature changes diagram in trabecular bone for ω=600 rpm

Stress distributions in trabecular bone for four drill tips have been represented in Figure 8 and variation diagram of stresses by depth, occurred by four different models on trabecular bone has been given in Figure 9. The similar stress distributions were observed at the same speed. Despite this, stress distributions occurred on trabecular bone yielded lower during drilling with first model drilling tip.

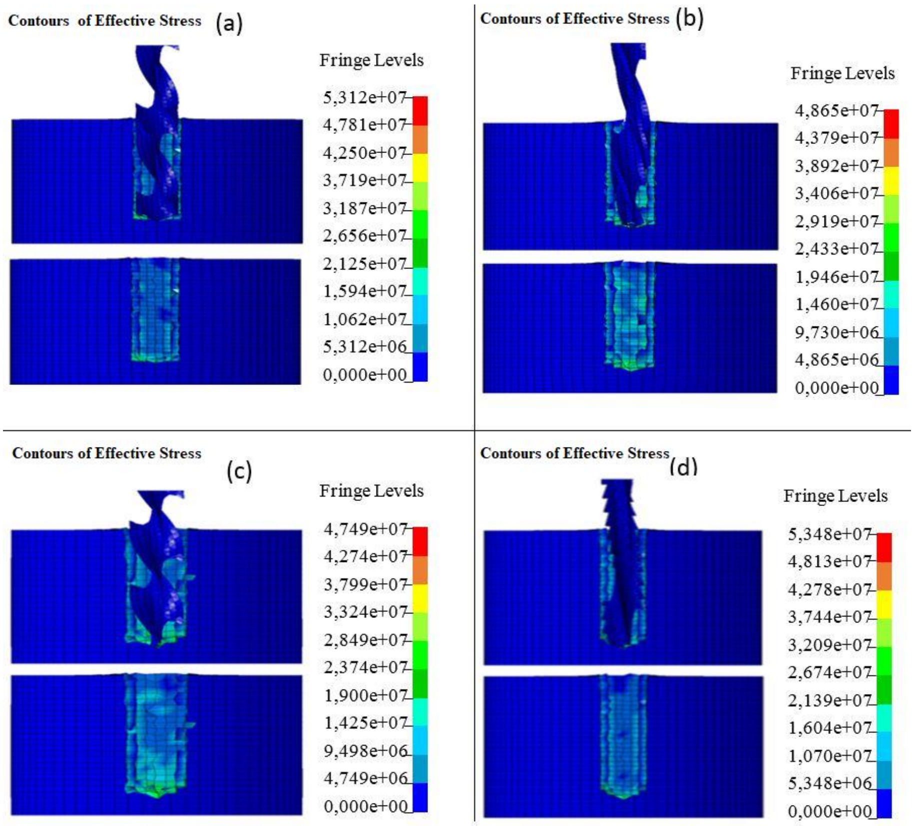

Figure 8. Stress distributions in trabecular bone for; a) First model, b) Second model, c) Third model, d) Fourth model

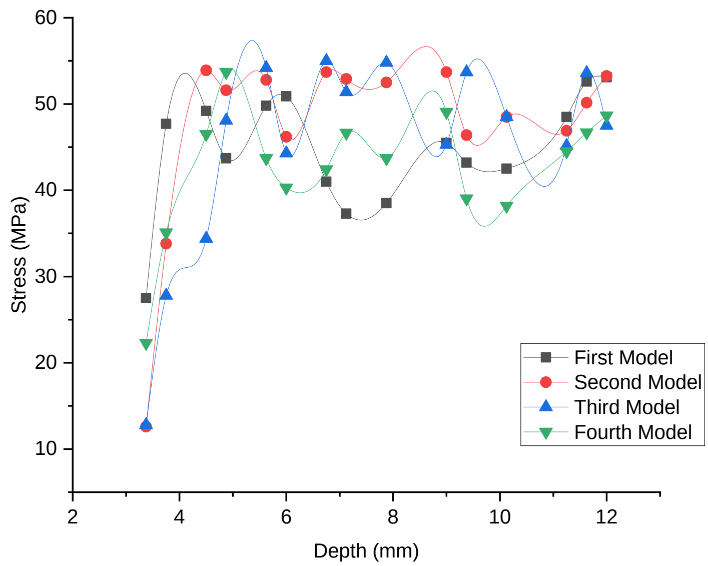

Figure 9. Stress changes diagram in trabecular bone for ω=600 rpm

3.1.2. For ω=800 rpm The Occurring Temperature and Stress Changes

As it seen Figure 10 and 11; the heat generated when drilling of implant cavity on jawbone depends on geometrical structure and material properties of drilling tip. Since the more drilling tip has screwed- structured geometrically, the more cavity will occur during drilling; less heat generates. Similarly, the more heat transmission coefficients of the coatings of drilling tips made of stainless steel is higher, the more effect of heat generated on jawbone will decrease.

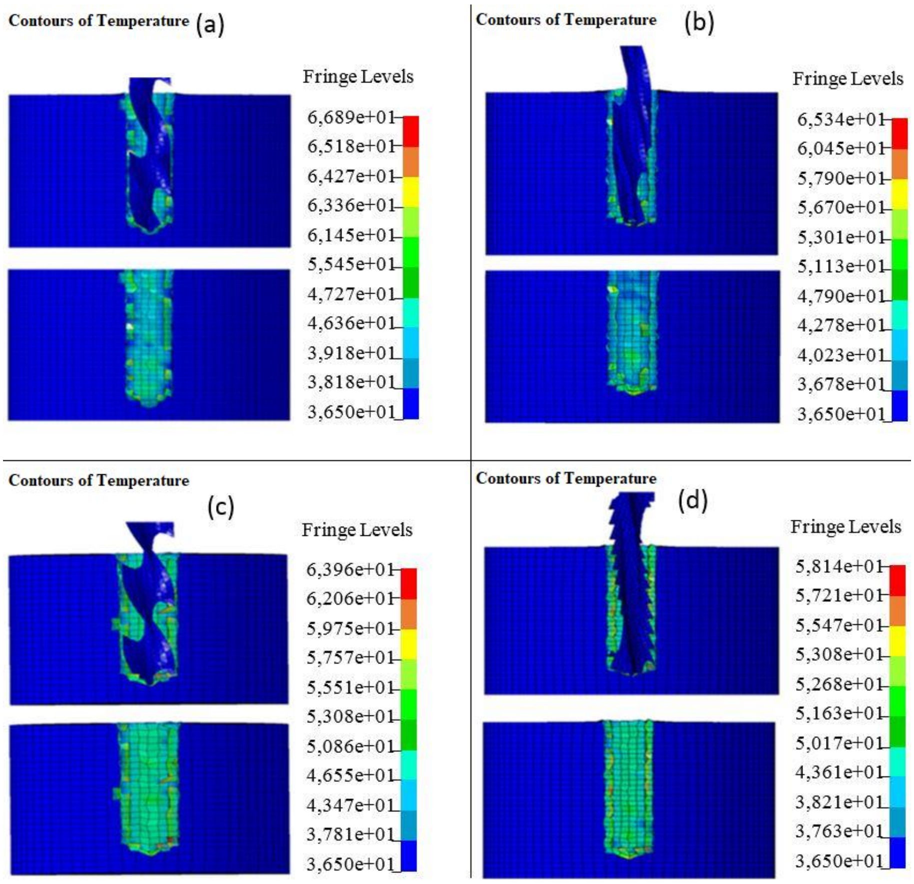

Figure 10. Temperature distributions in trabecular bone for; a) First model, b) Second model, c) Third model, d) Fourth model

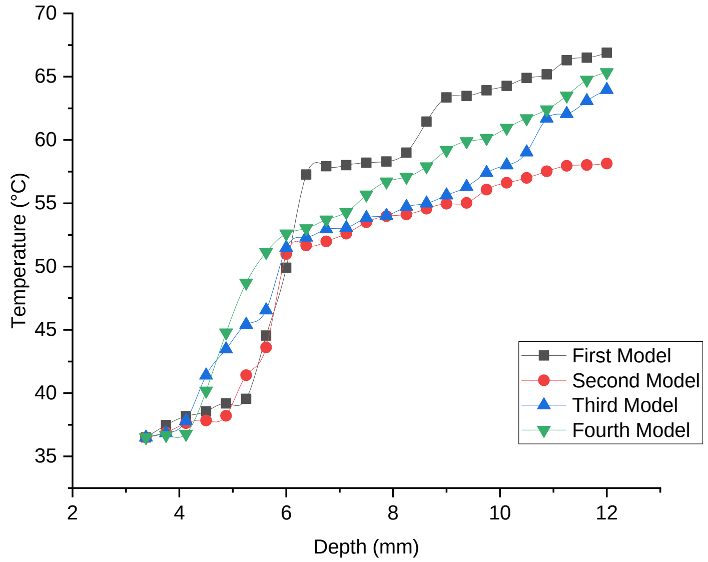

Figure 11. Temperature changes diagram in trabecular bone for ω=800 rpm

Stress distributions and variation diagram of stresses by depth occurred by four different models on trabecular bone have been given in Figure 12 and 13. The similar stress distributions were observed at the same speed. Despite this, stress distributions occurred on trabecular bone yielded lower during drilling with first model drilling tip.

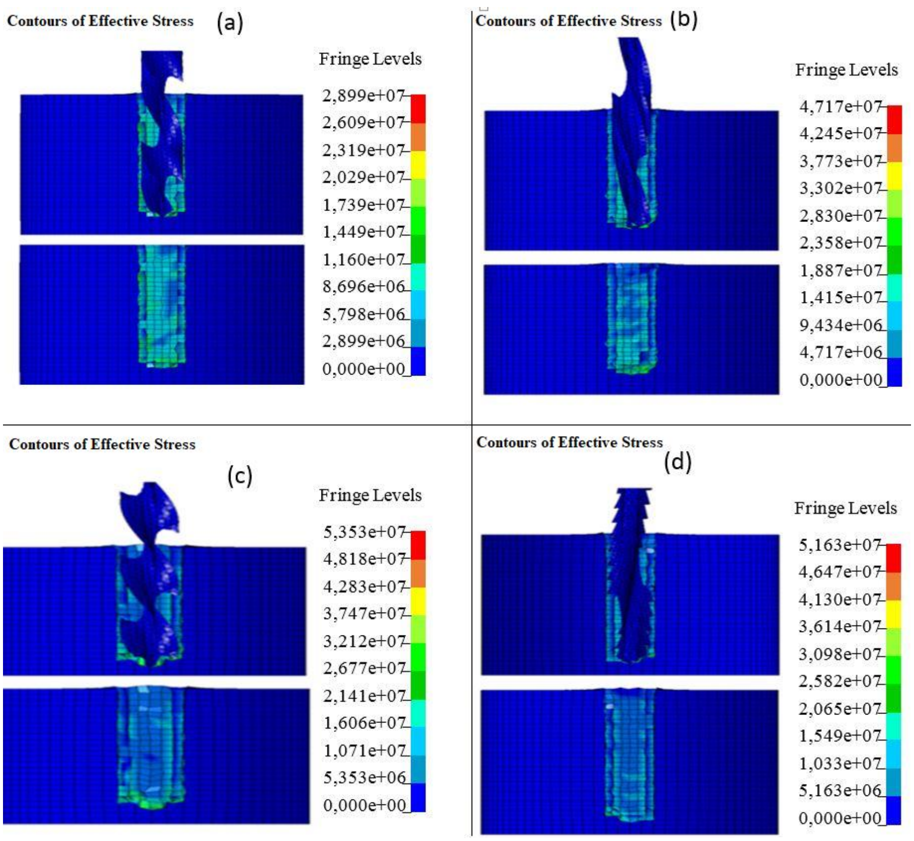

Figure 12. Stress distributions in trabecular bone for; a) First model, b) Second model, c) Third model, d) Fourth model

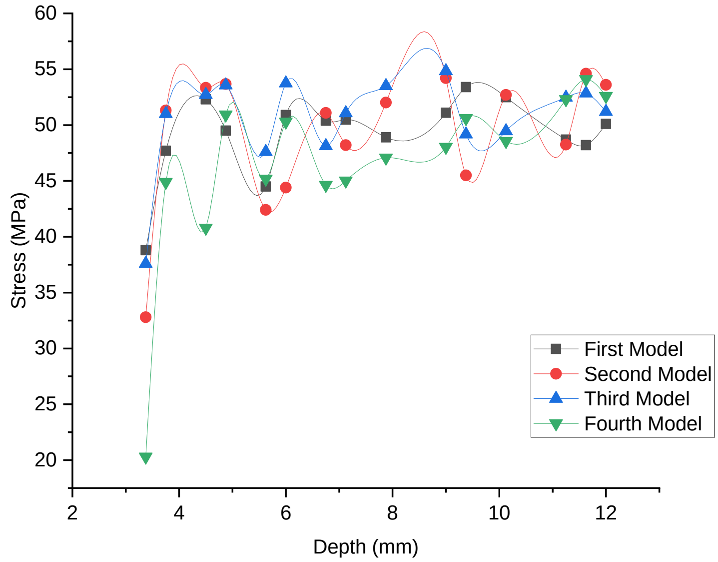

Figure 13. Stress changes diagram in trabecular bone for ω=800 rpm

3.2. Findings for Neural Predictor (ANFIS)

3.2.1. Prediction of Temperature Variation and Mechanical Stress for ω=600 rpm

Figure 14 and 15 render the prediction results of the thermal and mechanical stress values using the ANFIS approach. The simulation was realised for different type of the drills. As seen in the figures the prediction results of the jawbone parameters are quite good. Simulation results demonstrated that the root mean-squared error (RMSE) for any prediction was found to be 0.0003 which is completely low.

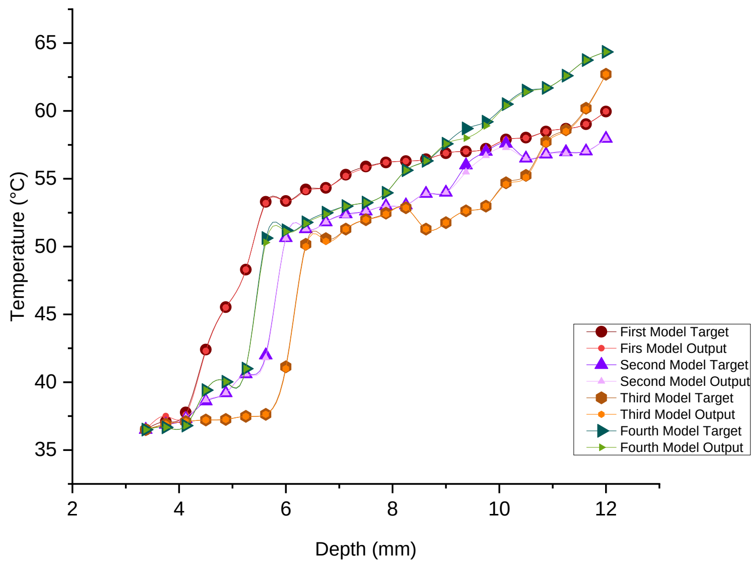

Figure 14. Prediction results of temperature variation for ω=600 rpm using ANFIS

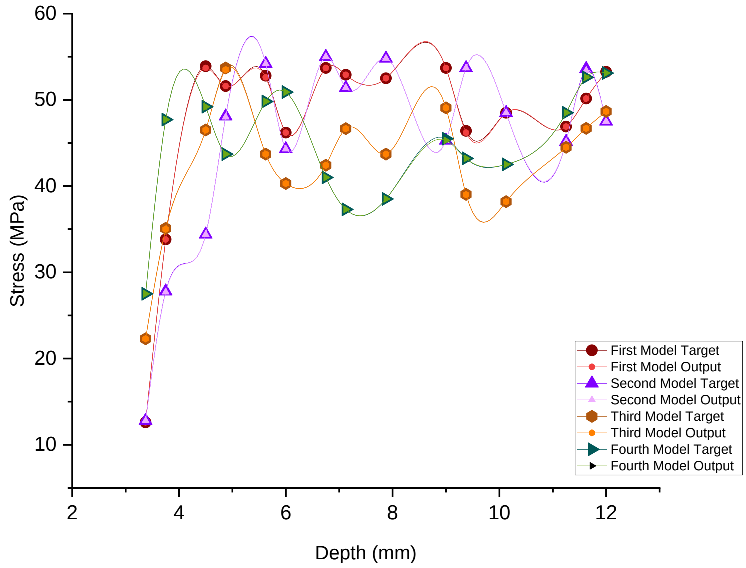

Figure 15. Prediction results of mechanical stress for ω=600 rpm using ANFIS

3.2.2. Prediction of Temperature Variation and Mechanical Stress for ω=800 rpm

Estimation results of the thermal and mechanical stress values for ω=800 rpm with ANFIS approach was given in Figure 16 and 17 for the different type of drills. As seen from the figures ANFIS structure is appropriate in prediction of the related thermal and mechanical parameters. And also RMSE error which is the model performance criteria was considered to be 0.0002.

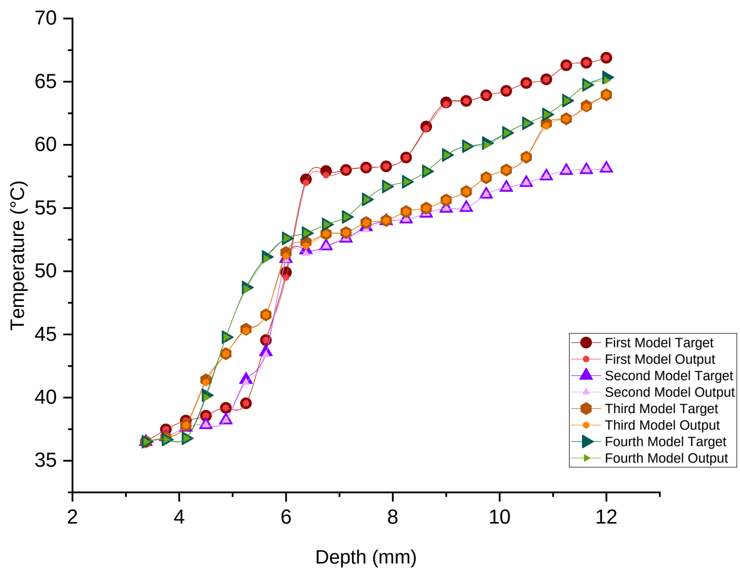

Figure 16. Prediction results of temperature variation for ω=800 rpm using ANFIS

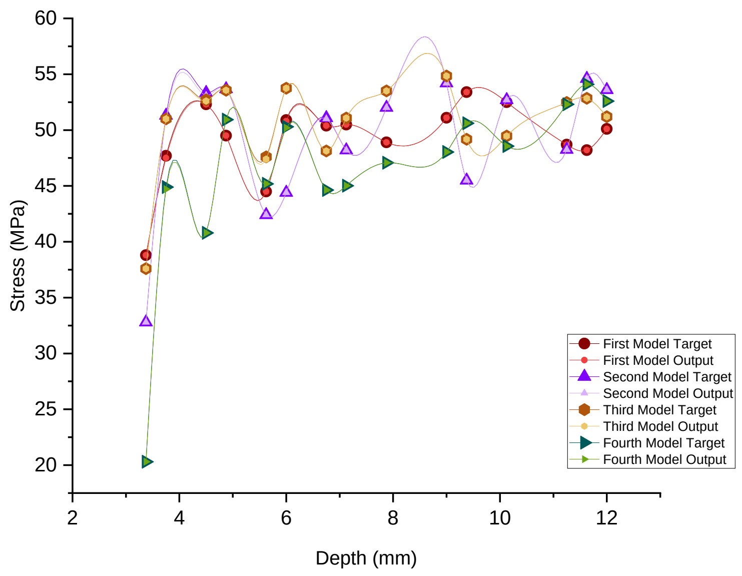

Figure 17. Prediction results of mechanical stress for ω=800 rpm using ANFIS

4. Conclusions and Discussions

The thermal and mechanical stresses occurring in the jawbone during the drilling process are of great importance in implant surgery. Therefore, critical parameters can be determined in advance with FEM and necessary measures can be taken.

The simulation results show that reveal how pitches structure of drilling tips affect temperature increase that occurs during drilling. During drilling with saw-tipped pitch structured drilling tip; maximum temperature occurred in rotational speed of 600 rpm at 38.8 °C and in rotational speed of 800 rpm at 39.8 °C. Maximum temperature increases occurred in rotational speed of 600 rpm at 57.96 °C and in rotational speed of 800 rpm at 58.14 °C on the trabecular bone. The temperature change increased with increasing of depth. It was determined that heat increase which occurred during drilling, with saw-tipped pitch structured drilling tip which has higher heat transfer coefficient, yielded less as compared with other groups.

In this study, similar mechanical stress distributions were observed at the same speed. For 600 rpm; maximum mechanical stress was about 3 Mpa in cortical bone, was about 52 MPa in trabecular bone. For 800 rpm; maximum mechanical stress was about 4 Mpa in cortical bone, was about 55 MPa in trabecular bone. This study has indicated that the maximum stress and maximum temperature increase with increasing speed.

Since cooling wasn't applied in the system, temperature due to increased heat generation is fairly above the critical temperature. When models were compared according to heat generation increases; because of that fourth model is less pitch structured as well as heat transmission coefficient of its coating material is lower, heat generation increase occurred more in this model. Since second model has saw-tipped pitch structure geometrically and also heat transmission coefficient of the coating material of the its drilling tip which is made of tungsten carbide was high, minimum temperature variation occurred. Since heat transfer will occur in case cooling liquid is added into the system, heat generation will increase less.

The findings of this study indicate that the unwanted thermal and mechanical stresses have arisen in the jawbone during the implant surgery. Accordingly, by predicting these stress values, necessary precautions can be taken before the surgical procedure. As the prediction of thermal and mechanical stresses, ANFIS approach has been used. Consequently, the proposed ANFIS predictor had a superior performance in showing the same behavior with numerical analysis outputs and shows that this approach can be used in real-time estimation of thermal and mechanical stresses changes in the during drilling bone.

References

[1] Branemark, P.I.; Hansson, BO;Adell, R.; et al.Osseointegrated implants in the treatment of the edentulous jaw. Experience from a 10-year period. Scand J Plast Reconstr Surg Suppl **1977**,16: 1-132.

[2] Albrektsson, T.; Eriksson, A. Thermally induced bone necrosis in rabbits: relation to implant failure in humans. Clinical Orthopaedics and Related Research **1985**, 195, 311-312.

[3] Lundskog, J. Heat and Bone Tissue. An Experimental Investigation of the Thermal Properties of Bone and Threshold Levels for Thermal Injury. Journal of Plastic and Reconstructive Surgery **1972**; 9, 1-80.

[4] Eriksson, R.A.; Adell, R. Temperatures during for the placement of implants using the osseointegration technique. Journal of Oral & Maxillofacial Surgery **1986**; 44(1), 4-7.

[5] Eriksson, R.A.; Albrektsson, T. Temperature threshold levels for heat- induced bone tissue injury: a vital-microscopic study in the rabbit. The Journal of Prosthetic Dentistry **1983**; Goteborg, 50(1), 101-107.

[6] Bachus, K.N.; Rondina, M.T; Hutchinson, D.T. The effects of drilling force on cortical temperatures and their duration: an in vitro study. Medical Engineering & Physics **2000**; 22: 685-69. https://doi.org/10.1016/S1350-4533(01)00016-9.

[7] Sharawy, M.; Misch, C.E.; Weller, N.; et al. Heat generation during implant drilling: the significance of motor speed. Journal of Oral & Maxillofacial Surgery **2002**; 60(10), 1160-1169. https://doi.org/10.1053/joms.2002.34992.

[8] Şener, B.C.; Dergin, G.; Gürsoy, B.; et al. effects of irrigation temperature on heat control in vitro at different drilling depths. Clinical Oral Implants Research **2009**; 20, 294-298. https://doi.org/10.1111/j.1600-0501.2008.01643.x.

[9] Nieri, M.; Crescini, A.; Rotundo, R.; et al. Factors affecting the clinical approach to impacted maxillary canines: a bayesian network analysis. Am Orthod Dentofac Orthop **2010**; 137:755-62. https://doi.org/10.1016/j.ajodo.2008.08.028.

[10] Miladinović, M.; Mihailović, B.; Janković, A.; et al.. Reasons for extraction obtained by artificial intelligence. Acta Fac Med Naissensis **2010**;27:143-58.

[11] Sukegawa, S.; Kazumasa, Y.; Takeshi, H.; et al. Deep Neural Networks for Dental Implant System Classification. Biomolecules **2020**; 10(7):984. doi: 10.3390/biom10070984.

[12] Turgut, B. Examination of mechanical and thermal changes caused by implant application in the jawbone, Msc, Erciyes University, 2016.

[13] Turgut, B., Erdemir, D., Altuntop, N. Examınatıon of mechanıcal and thermal varıatıons caused by the ımplantology ın the jawbone. International Conference on Advances in Mechanical Engineering **2016**; 788-795.

[14] Eskitaşcıoğlu G. Investigation of functional stress in different centric contact types in natural and prosthetic restorated teeth with structural analysis program, Phd, Ankara University, 1991.

[15] Kosedağ, E., Ekici, R., Free Vibration Analysis of Foam- Core Sandwich Structures, Journal of Polytechnic, **2021**;24(1):69-74. https://doi.org/10.2339/politeknik.571396

[16] Ekici, R., Kosedağ, E., Comparison Of The Low-Velocity Impact Behaviors Of Sic And Pumice Particle-Reinforced Metal Matrix Composites, International Journal of Mechanical And Production Engineering, **2017**,5(10)

[17] İmiroğlu, H.İ.; Tosun, Z.; Kaymaz, İ.; et al. A New TMJ Implant Design. Süleyman Demirel University Journal of Engineering Sciences and Design **2014**; 2(3), 199-210.

[18] Huiskes, J. Some Fundamental Aspects Of Human Joint Replacement. Acta Orthopaedica **1980**, 185, 44-108.

[19] Alam, K.; Mitrofanov, A.V.; Silberschmidt, V.V. Thermal analysis of orthogonal cutting of cortical bone using finite element simulations. International Journal of Experimental and Computational Biomechanics **2010**; 1(3).https://doi.org/10.1504/IJECB.2010.035259 .

[20] Davidson, S.R.H.; James, D.F. Measurement of thermal conductivity of bovine cortical bone. Medical Engineering and Physics **2000**; 22, 741-747. https://doi.org/10.1016/S1350-4533(01)00003-0.

[21] Clattenburg, R.; Cohen, J. Thermal properties of cancellous bone, Journal of Biomedical Materials Research **1975**; 9, 169-182.

[22] http://silver.neep.wisc.edu/~lakes/BoneTrab.html .(Accessed on 20 June 2021).

[23] Kopperdah, D.L.; Keaveny, T.M. Yield strain behavior of trabecular bone. Journal of Biomechanics **1998**; 31: 601-608. https://doi.org/10.1016/S0021-9290(98)00057-8.

[24] Tüfekçi, K.; Kayacan, R.; Kurbanoğlu, C. Investigation of the mechanical behavior of cortical bone sterilized by gamma radiation under dynamic load. Süleyman Demirel University Journal of Engineering Sciences and Design **2014**; 2(3), 299-302.

[25] Bevill, G.; Easley, S.K.; Keaveny, T.M. Side-artifact errors in yield strength and elastic modulus for human trabecular bone and their dependence on bone volume fraction and anatomic site. Journal of Biomechanics **2007**; 40(15). https://doi.org/10.1016/j.jbiomech.2007.05.008.

[26] http://www.asm.matweb.com. (Accessed on 20 June 2021).

[27] http://www.matweb.com. (Accessed on 20 June 2021).

[28] http://www.tasarimveimalat.com.(Accessed on 20 June 2021).

[29] Bachir, O.; Zoubir, A.F. Adaptive neuro-fuzzy inference system based control of puma 600 robot manipulator. Int J Electr Comput Eng **2012**; 2, 90–97.

[30] Bevill, G., Easley1, S.K., Keaveny, T.M., 2008. Side-Artifact Errors İn Yield Strength And Elastic Modulus For Human Trabecular Bone And Their Dependence On Bone Volume Fraction And Anatomic Site. NIH Public Access, University of California, Berkeley, CA, USA.15s

[31] Tunalı, B., 2000. Multi-Disipliner Bir Yaklaşımla Oral İmplantoloji. 2. baskı Nobel Tıp Kitabevleri Ltd. Şti İstanbul.