Design and Analysis of the Customized Implant System Using CAD/CAM/CAE Tools

[1]Durmuş Ali Bircan, [1]Dinçer Dede and, [1]Abdul Kadir Ekşi
[1]Çukurova University, Mechanical Engineering Depth., 01330 Balcalı, Adana, TÜRKİYE, Tel: 322 3386084/2722/115, abircan@cu.edu.tr

Abstract

The components of dental implant system are biomedical element with complex geometries, and the design and production are complicated task. The aim of this study is to design and analyzes customized implant system for patient. First, 3-Dimensional (3D) modeling of jaw and other artificial structures and most realistic simulation models were created. To develop an accurate design and analysis; the DICOM (Siemens medical image format) files were obtained by scanning Computer Tomography (CT) images of jawbone are transformed into two different point cloud format as cortical and cancellous bone layers. Obtained point clouds were firstly converted to the solid model format and then combined using SolidWorks program. In this way, customized jaw model of the patient were obtained. Finally, the appropriate implant, abutment and crown bridge for the customized jawbone was modeled. Created 3D solid models were assembled and made suitable for the Finite Element Analysis (FEA).
Structural analysis of created 3D solid model was carried out in ANSYS with the finite element method, locations and amounts of stresses and displacements were determined by examining the stress distributions. Thus, the most appropriate geometry and material of implant, abutment and crown bridge that will be used in the implant treatment for the patient were assigned.

***Keywords*:** Implant, Customized Design, Computer Aided Design, Finite element analysis.

1. Introduction

Dental implantology, a special field of dentistry dealing with the rehabilitation of the damaged chewing apparatus due to loss of the natural teeth, is currently most intensively developing field of dentistry. Missing teeth can be replaced by dental implants (artificial roots), which are inserted into the root-bearing parts of the mandible or maxilla as presented in Fig.1.
The long-term benefits of dental implants include improved appearance, comfort, speech and self-esteem. With dental implant, the patient can eat more conveniently. In addition, the implant is able to protect the remaining natural teeth, stop bone loss and restore facial skeletal structure. As far as the costs are concerned, implants have shown to be less expensive overall than other types of prostheses such as crown and bridge restorations. Implants placed into bone fused with the bone in a certain recovery time and they support fixed or removable dentures made on them. In this way, many problems would be eliminated revealed by conventional prostheses.
Patients are going to the dentist many times to determine the appropriate implant design during the implant treatment process. This case causes loss of time and cost for the patient and also for the dentist. Within this study, it is aimed eliminate this negative situation. CT images of a patient's jawbone will be ripped and will be processed in a special program by the method of reverse engineering. At the end, 3-D solid jaw model will be created. Dentists will able to determine the suitable implant system for the patient in the computer generated through this

*Corresponding author: [1]Çukurova University, Mechanical Engineering Depth., 01330 Balcalı, Adana, TÜRKİYE.
E-mail address: abircan@cu.edu.tr, Phone: +90322 3386084/2722/115, Fax: +90322 3386126

model. In the near future, with the aid of rapidly developing CAD/CAM/CAE (Computer Aided Design/Computer Aided Manufacturing/Computer Aided Engineering) tools, specialized implants and prostheses can be produced or improvements can be made over existing implants.

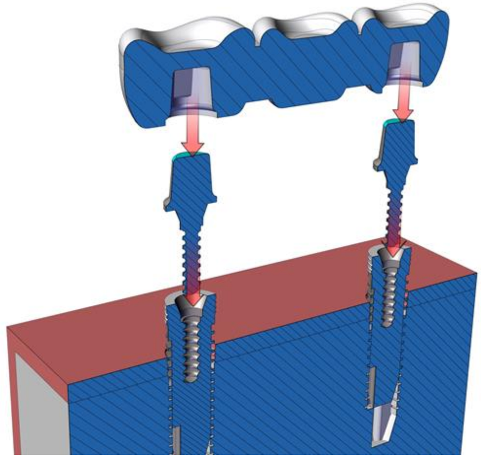


Figure 1. Dental implant system

The success and long term prognosis of implant prosthetic therapy depend primarily on the anchorage of the implant in the jawbone, i.e. on the osseointegration. Osseointegration may broadly be defined as the dynamic interaction and direct contact of living bone with a biocompatible implant in the absence of an interposing soft tissue layer [1]. In some cases, the implant system placed into the bone cannot withstand the stress during chewing and that may cause undesirable situations such as bone erosion or prolonged recovery after surgical operations [2].

Many complexities and variability occurs in human jawbone during chewing movements. Different analysis and simulations should be performed to understand these complex and variable situations. Doctors and engineers need to work together for the creation of models and simulations.

Jawbone is one of the hardest parts for investigation in human anatomy from a biomechanical aspect due to the lack of a uniform shape, bone structure varying from person to person and one of the most intense parts of the movement. Finite Element Method (FEM) is used in many studies related to the jawbone and implant surgery. Making actual experiments on human or animal in biomechanics applications is often difficult and impossible task. FEM is a very important method to determine the unknown behavior of many structures.

In the previous studies, many modeling and analysis have been made with FEA methods. The stress distribution that occurs in the jawbone was investigated for different implant geometries [3]. The effect of implant diameter and implant length on the stress distribution was investigated [4]. The effect of different abutment designs on the implant systems was also examined [5]. However, rectangular structures have been considered as the jawbone generally instead of the specific jawbone model in the majority of these studies.

In this study, different implant thread types are designed on the lower jawbone model. At the end of the study, dentist will decide which type of implant design is optimum for the patient based on bone density in patient’s jawbone. To prefer optimum thread design will increase the percentage of long-term success of the implant. In the study 13 mm in length and 3.5 mm diameter standard experimental implants "V", "frame" and "circular" type thread forms were placed on the jawbone model, and analyzed.

2. Materials and Method

In this study, Reverse Engineering (RE) method is used to obtain three-dimensional model of the lower jawbone. Firstly, 35-year-old male patient's jawbone is scanned in computed tomography device. The resulting data stored in the DICOM format that is medical image format of Siemens and has been made to transfer to the medical image processing program. After, scanned DICOM files are converted into two different point cloud formats as cancellous and cortical bone layers by means of Mimics program (Fig.2).

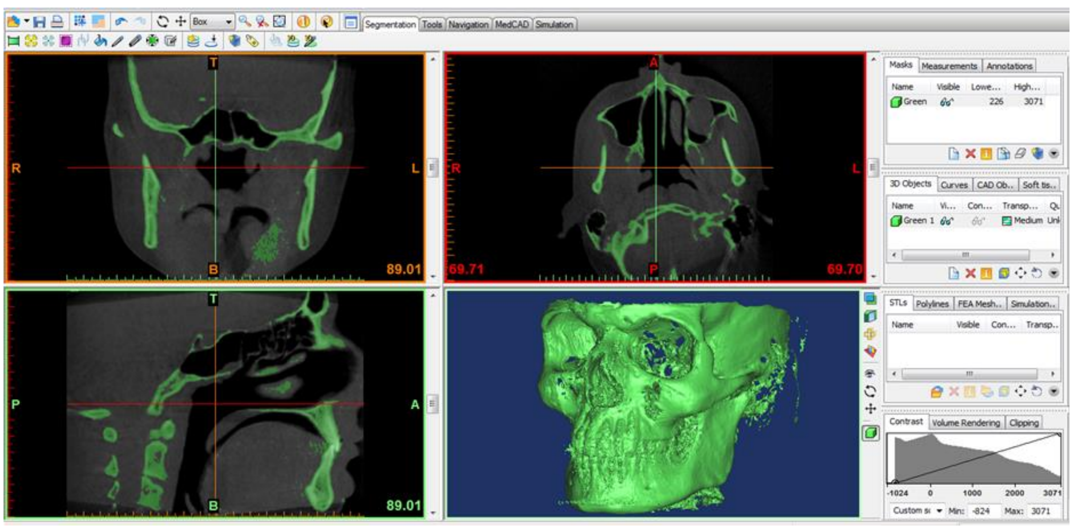


Figure 2. Obtaining of jawbone model as point cloud from the DICOM files

Three-dimensional models in point cloud format are not appropriate mathematical models to be used for FEA [6]. So, the resulting two different three-dimensional point cloud model are transferred into SolidWorks program which is 3-D CAD program and wherein are converted into the solid model format. Obtained cancellous and cortical bone layer in the form of solid models are combined in SolidWorks and three dimensional model of the lower jawbone has been obtained as can be seen in Fig.3.

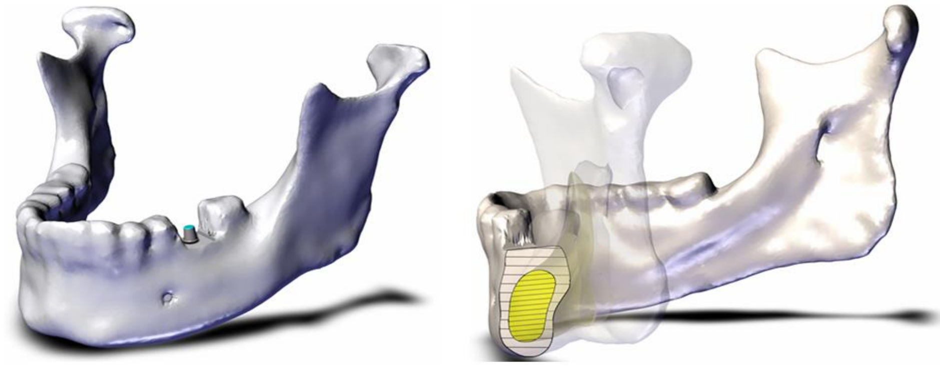


Figure 3. Obtaining of solid jawbone model

The jawbone model implants with a diameter of 4.3 mm and an overall length of 14.25 mm is modeled by using CAD software. SolidWorks by Dassault Systemes is used to model all the implant system. All implant forms are set as 0.8 mm pitch and 0.6 mm the depth of thread. These values are determined by the commercially used dental implants [7]. Fully osseointegration is considered between implant and bone. Implants which have different thread type can be seen in Fig.4.

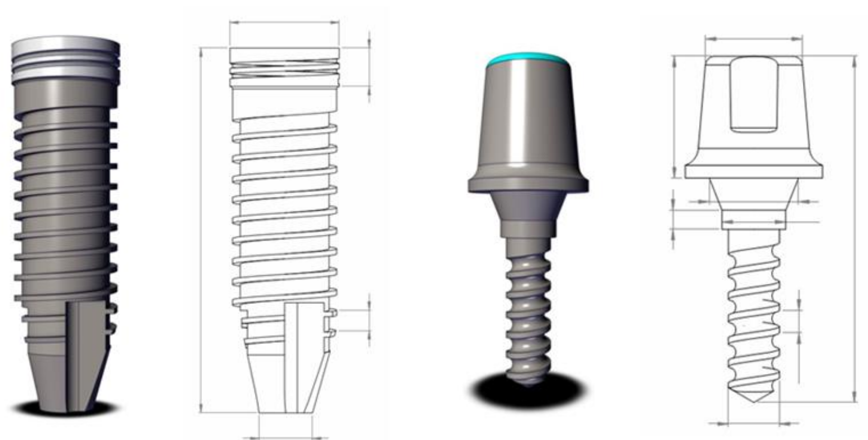


Figure 4. Implant and abutment designed

Created geometric models are transferred to the ANSYS program to be done for FEA. Selection of element type on the mathematical model, creating the mesh form, determining the contact areas, boundary conditions, environment and material properties and the type of analysis have been made in the program interface. Tetrahedron element is used as shown in Fig.5. After the meshing in ANSYS average 235175 points and 139420 elements are obtained for nine different models.

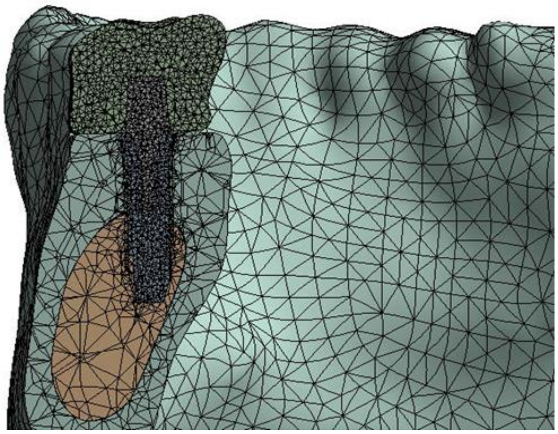


Figure 5. Tetrahedron Mesh

Some physical properties are to be entered of each structure in the model for the analysis after meshing process. These properties are Modulus of Elasticity (E) and Poisson's ratio (γ) which are the mechanical properties of materials. Modulus of Elasticity is a measure of elastic deformation of the material under the force [8].Poisson's ratio is the ratio of the relative contraction strain, or transverse strain normal to the applied load, to the relative extension strain, or axial strain in the direction of the applied load [9].

Bone is divided into two layers as cortical and cancellous according to the intensity of bone. Cancellous bone layer located under the layer of cortical bone is a porous structure and less dense and less rigid according to cortical layer [10]. Hence two bone layer have different mechanical properties. In this study, modulus of elasticity is taken as 15 GPa for the cortical bone and 1 GPa for the cancellous bone. Poisson's Ratio is taken as 0.3 for the cortical and cancellous bone. Titanium, Zirconium and Cadmium have been selected as three different materials for the implants and abutments. Porcelain is used as tooth material (Table 1).

Table 1. Mechanical properties of the materials

Material	**Modulus of Elasticity (GPa)**	**Poisson Ratio**
Cortical bone	15 GPa	0,3
Cancellous bone	1 GPa	0,3
Titanium	110 GPa	0,3
Zirconium	210 GPa	0,25
Cadmium	50 GPa	0,3
Porcelain	70 GPa	0,19

When determining the boundary conditions, considering that ramus section of the lower jawbone remains stationary in x, y and z directions, all the elements of the model in this region are given zero degrees of freedom (Fig.6). Two different occlusal forces, 150 N in the horizontal direction and the 300 N in the oblique direction that is the resultant of vertical forces are applied to the created model as the parallel to the long axis of the implant (Figure 6). The magnitude of the forces is given by considering the maximum masticatory forces in the mouth [11].

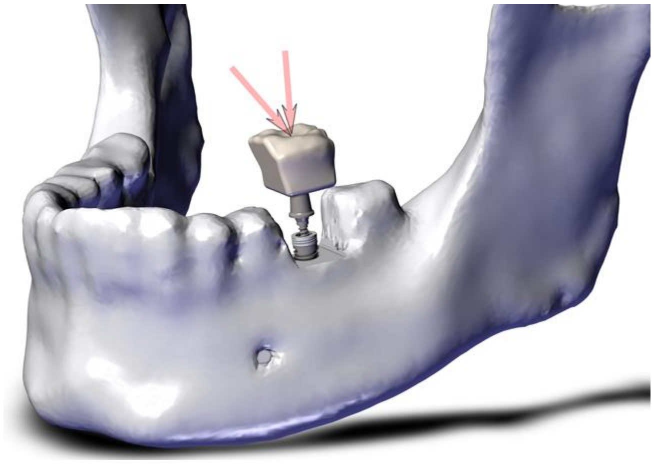


Figure 6. Boundary and loading conditions

3. Results and Discussion

It has been reported that Most of the studies using FEM used Von Mises stress criterion which calculates numerically the stress condition is sufficient [12]. In this study, the findings are compared according to this energy criterion.

To summarize the results;

1.) Maximum stress value is observed on the cortical bone surface, especially first contacted side with the implant (Fig.7-8-9).

2.) It is determined that implant geometry with 'square' thread type causes least stress (Fig.8).

3.) There is not a significant difference between the stress values created on the bone by the implants which have the same geometry but different materials (Fig.10).

4.) Stress accumulation is occurred on the hills of thread of used all of dental implants (Fig.7-8-9).

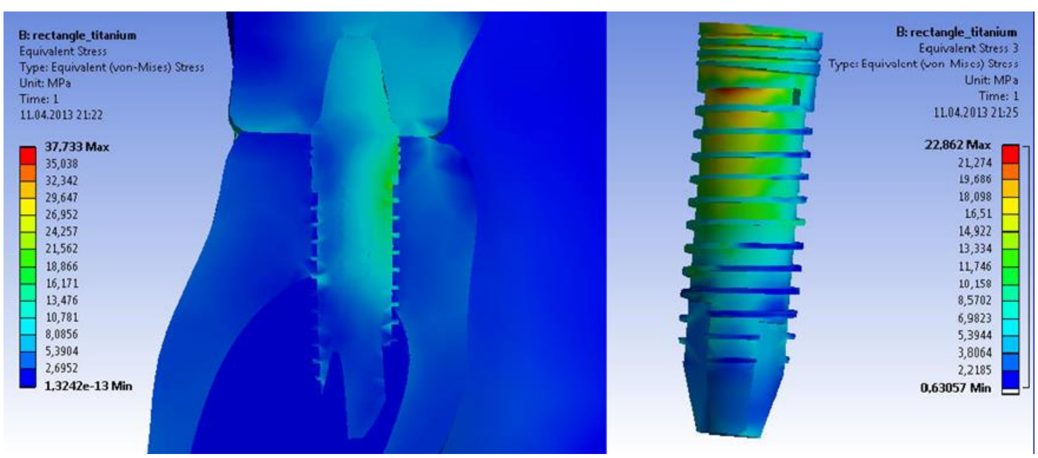


Figure 7. Stress distribution on the bone-implant interface and on the square thread type implant

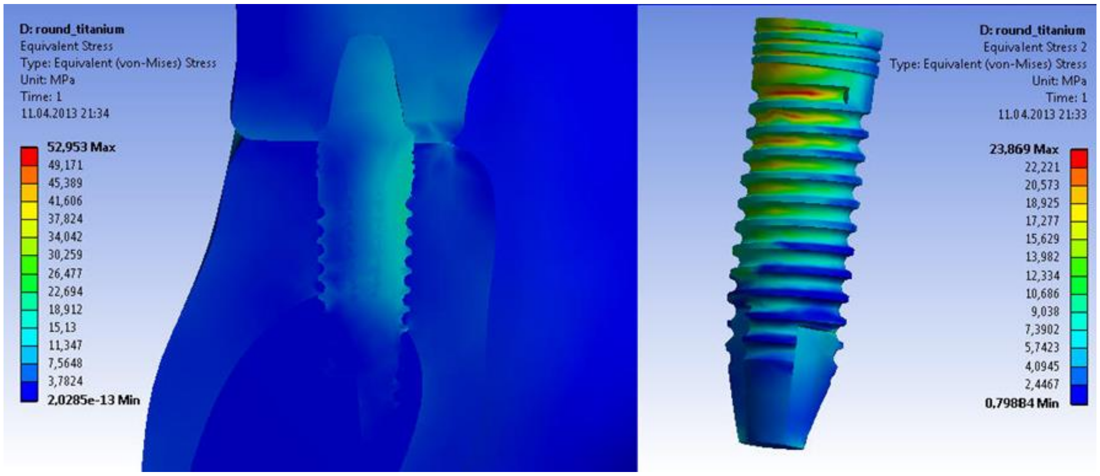


Figure 8. Stress distribution on the bone-implant interface and on the round thread type implant

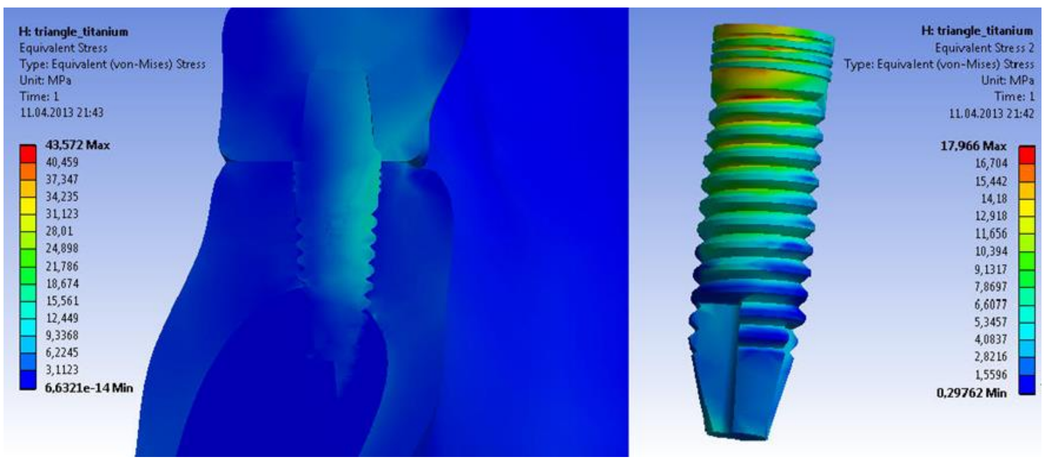
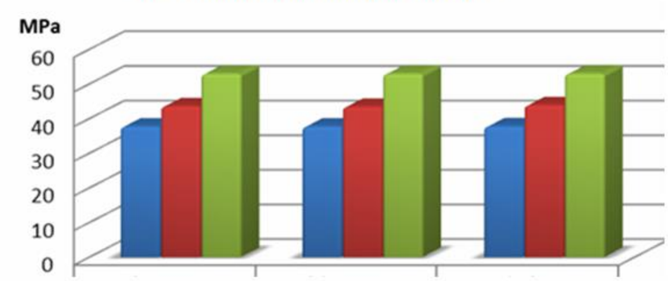


Figure 9. Stress distribution on the bone-implant interface and on the triangle thread type implant

Stress distribution on bone interface



	Titanium	Zirconium	Cadmium
Square thread	37,733	37,723	37,750
Triangle thread	43,572	43,391	43,864
Round thread	52,953	52,936	52,980

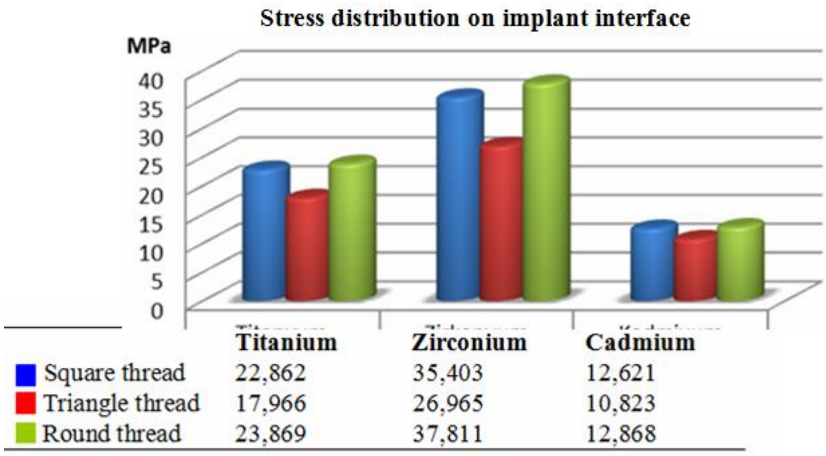


Figure 10. Maximum stress values occurred on the bone (a) and implant (b) by dental implants which have different geometries-materials

4. Conclusions

In this study, different thread forms and materials of implant were numerically investigated by means of FEM under occlusal and lateral loads. Stress distribution on the cortical, cancellous bone and implant is analyzed. Stress analyses were performed for perfect osseointegration configuration at bone-implant interface. The influence of implant‘s thread profile and material on implant mechanical performance and indications for the risk of bone weakening or loss due to local tissue overloading were determined.

Acknowledgement
This study is supported by Çukurova University Research Fund. (MMF2010BAP112).

5. References

1. Masuda T, Yliheikkila P, Felton D, Cooper L, Generalizations Regarding the Process and Phenomenon of Osseointegration. Part I, In Vivo Studies. The International Journal of Oral and Maxillofacial Implants 1998, 13:17-29.
2. Misch, C., Dental Implant Prosthetics St. Louis. Elsevier Mosby, 2005.
3. Franciosa P., Martorelli M., Stress-based performance comparison of dental implants by finite element analysis, International Journal on Interactive Design and Manufacturing (IJIDeM) May 2012, Volume 6, Issue 2, 123-129
4. Baggi L., Cappelloni I., Di Girolamo M., Maceri F., Vairo G., The influence of implant diameter and length on stress distribution of osseo integrated implants related to crestal bone geometry: A three-dimensional finite element analysis, J. Prosthet Dent. 2008 Dec; 100(6):422-31.

5. Hasan I., Röger B., Heinemann F., Keilig L., Bourauel C., Influence of abutment design on the success of immediately loaded dental implants: Experimental and numerical studies, Med Eng Phys.,2012 Sep;34(7):817-25.
6. Dilek M., Bircan D., A., Ekşi A. K., Construction of 3D Finite Element Model of Human Mandible For Biomechanical Analyses, 2nd International Scientific Conference on Engineering "Manufacturing and Advanced Technologies" MAT 2012, Antalya Turkey, 22-24 November 2012.
7. Nobel Biocare Product Catalog 2011 NobelSpeedy™ Replace RP.
8. Geng JP, Tan KB, Liu GR, Application of finite element analysis in implant dentistry: a review of the literature. J Prosthet Dent. Jun; 2001,85(6):585-98.
9. Gercek H., Poisson's ratio values for rocks. Int. Journal of Rock Mec. and Min. Sci.; Elsevier; January; 2007, 44 (1): pp. 1–13
10. Heng-Li Huang, Jehn-Shyun Huang, Ching-Chang Ko, Jui-Ting Hs, Chih-Han Chang, Michael Y. C. Chen, Effects of splinted prosthesis supported a wide implant or two implants: a three-dimensional finite element analysis, Clinical Oral Implants Research, 2004, Volume 16, Issue 4,
11. Kurniawan D., Nor F. M., Lee H. Y., Lim J. Y., Finite element analysis of bone implant biomechanics: refinement through featuring various osseointegration conditions, Int. J. Oral Maxillofac. Surg., 2012 Sep;41(9):1090-6
12. Richter EJ. Basic biomechanics of dental implants in prosthetic dentistry. J. Prosthet. Dent. 1989;61: 602-609.
13. Stegaroiu R, Kusakari H, Nishiyama S, Miyakawa O., Influence of prosthesis material on stress distribution in bone and implant: A three dimensional finite element analysis. Int J Oral Maksillofac. Implants 1998; 13: 781-790.