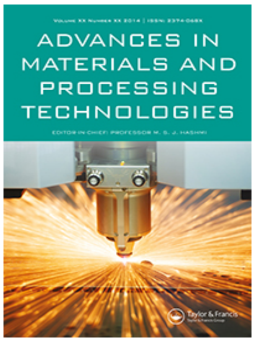
Development of the customised implant system using CAD/CAM/CAE tools

Article *in* Advances in Materials and Processing Technologies · March 2016

DOI: 10.1080/2374068X.2016.1159025

Durmus Ali Bircan
Cukurova University



Advances in Materials and Processing Technologies

ISSN: 2374-068X (Print) 2374-0698 (Online) Journal homepage: http://www.tandfonline.com/loi/tmpt20

Development of the customised implant system using CAD/CAM/CAE tools

D. A. Bircan, D. Dede & A. K. Ekşi

ADVANCES IN MATERIALS AND PROCESSING TECHNOLOGIES, 2016
http://dx.doi.org/10.1080/2374068X.2016.1159025

Taylor & Francis
Taylor & Francis Group

Development of the customised implant system using CAD/CAM/CAE tools

D. A. Bircan, D. Dede and A. K. Ekşi

Mechanical Engineering Department, Çukurova University, Adana, Turkey

ABSTRACT

In this paper, the 3-dimensional (3D) modelling of jaw and other artificial structures and most realistic simulation models are developed and studied. Optical scanning, computer aided design (CAD) and computer aided analysis (CAE) programs are integrated with each other are used. The DICOM (Siemens medical image format) files that are 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 are firstly converted to the solid model format and then combined in SolidWorks. In this way, specific jaw model of the patient is obtained. Finally, the appropriate implant, abutment and crown bridge for the jawbone is modelled. Created 3D solid models are assembled in SolidWorks and were 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 are assigned and a basic structure is formed that all types of improvements can be applied.

ARTICLE HISTORY
Accepted 24 February 2016

KEYWORDS
Implant; mandible; 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 the 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 Figure 1.

The long-term benefits of dental implants include improved appearance, comfort, speech and self-esteem. With the 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.

CONTACT D. A. Bircan abircan@cu.edu.tr

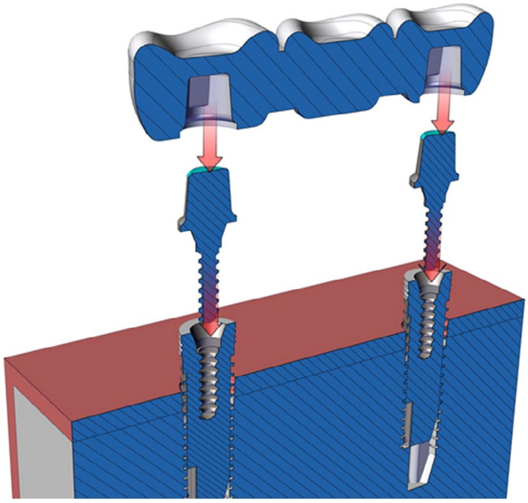


Figure 1. Dental implant system.

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 associated with using conventional prostheses operations can be eliminated.

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 to eliminate this negative situation. CT images of a patient's jawbone will be ripped and these images will be processed in a special programme by the method of reverse engineering(RE). In the end, 3-D solid jaw model will be created. Dentists will be able to determine the suitable implant system for the patient in the computer generated through this model. In the near future, with the aid of rapidly developing rapid prototyping (RP) method specific implants and prostheses can be produced or improvements can be made over existing implants.

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] The most appropriate implant system for the patient should be determined by the dentist to prevent undesirable situations.

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

Jawbone is one of the hardest parts in human anatomy to investigate 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. As in other biomechanical applications, finite element method is used in many studies related to the jawbone and implant surgery. Making actual experiments on human or animal in biomechanics applications is often a difficult and even impossible task. Finite element method is a very important method to determine the unknown behaviour of many structures.

In the previous studies, many modellings and analysis have been made with FEA methods. The stress distribution that occurs in the jawbone is investigated for different implant geometries.[3] The effect of implant diameter and implant length on the stress distribution is investigated.[4] The effect of different abutment designs on the implant systems is 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. In the end of the study, dentist will decide which type 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' type, 'frame' type and 'circular' type of three different thread forms is placed on the jawbone model, and were analysed.

2. Material and method

In this study, reverse engineering method is used to obtain 3D model of the lower jawbone. Firstly, a 35-year-old male patient's jawbone is scanned using a 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 this step, scanned DICOM files are converted into two different point cloud formats as cancellous and cortical bone layers by means of Mimics program (Figure 2).

3D models in point cloud format are not appropriate mathematical models to be used in finite element analysis method.[6] So the resulting two different 3D point cloud model are transferred into SolidWorks program which is 3D CAD program and wherein are converted into the solid model format. Obtained cancellous and cortical bone layer in the form of

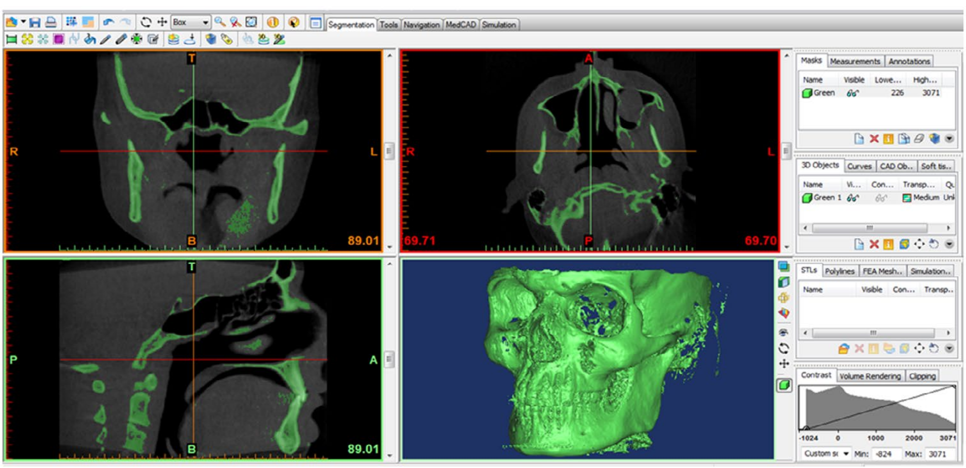


Figure 2. Obtaining of jawbone model as point cloud from the DICOM files.

solid models are combined in SolidWorks and 3D model of the lower jawbone has been obtained as can be seen in Figure 3.

After creating the jawbone model, implants with a diameter of 4.3 mm and an overall length of 14.25 mm is modelled 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] Full osseointegration is considered between implant and bone. Implants which have three different thread type can be seen in Figure 4.

Created geometric models are transferred to the ANSYS program to do finite element analysis. 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 Figure 5. After the meshing in ANSYS average 235,175 points and 139,420 elements are obtained for nine different models.

Some physical properties are to be entered for each structure in the model for the analysis after meshing process. These properties are modulus of elasticity (E) and Poisson's ratio

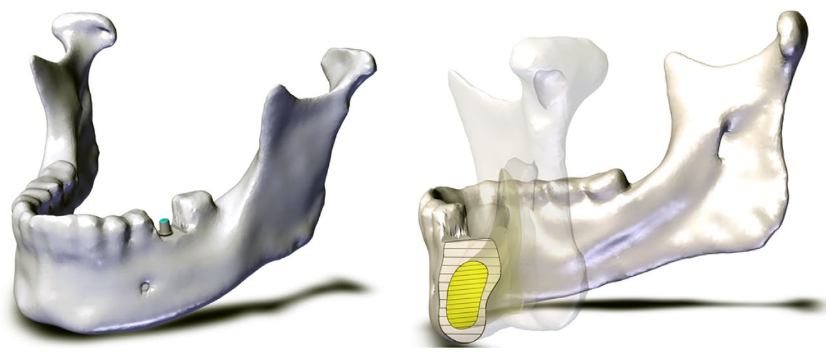


Figure 3. Obtaining the solid jawbone model.

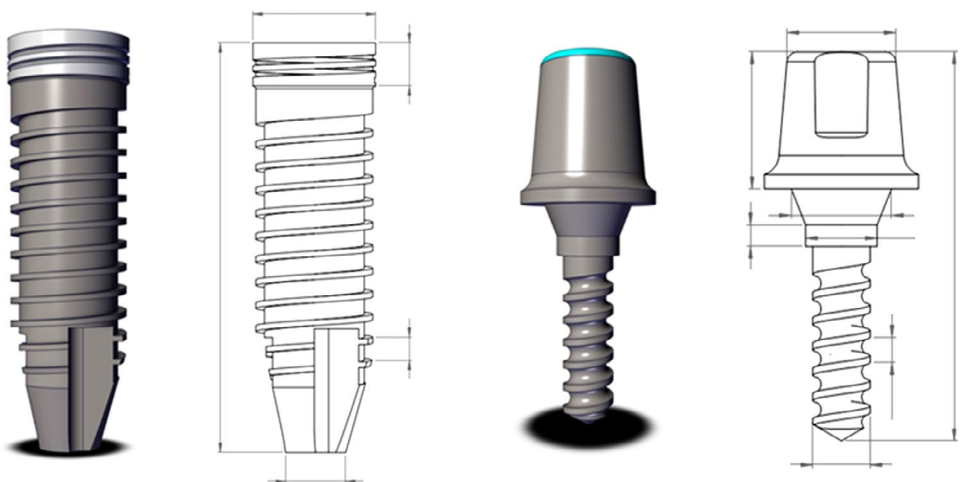


Figure 4. Implant and abutment designed.

(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, cortical and cancellous, depending on the density of bone. Cancellous bone layer located under a layer of cortical bone is a porous structure, less dense and less rigid than the cortical layer.[10] Hence, the two bone layers 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 teeth material (Table 1).

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 (Figure 6).

At the last step, two different occlusal forces, of 150 N in the horizontal direction and of 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]

3. Results and discussion

Most of the studies wherein finite element stress analysis method was used reported that using the Von Mises stress criterion which calculates numerically the stress condition is sufficient.[12, 13] In this study, the findings are compared according to this energy criterion.

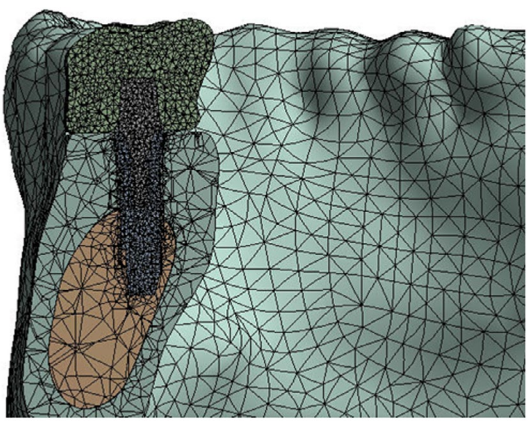


Figure 5. Tetrahedron mesh.

Table 1. Mechanical properties of the materials.

Material	Modulus of elasticity (GPa)	Poisson ratio
Cortical bone	15	0.3
Cancellous bone	1	0.3
Titanium	110	0.3
Zirconium	210	0.25
Cadmium	50	0.3
Porcelain	70	0.19

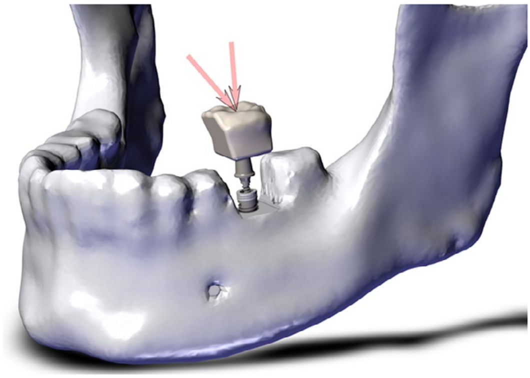


Figure 6. Boundary and loading conditions.

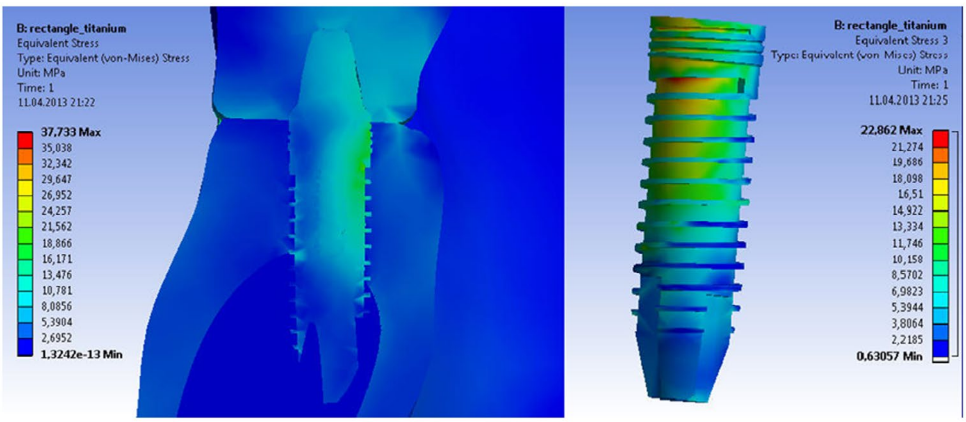


Figure 7. Stress distribution on the bone-implant interface and on the square thread type implant.

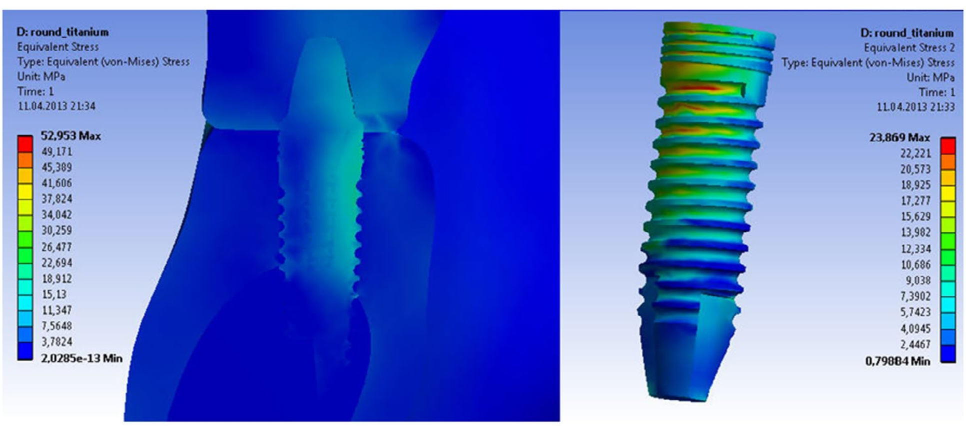


Figure 8. Stress distribution on the bone-implant interface and on the round thread type implant.

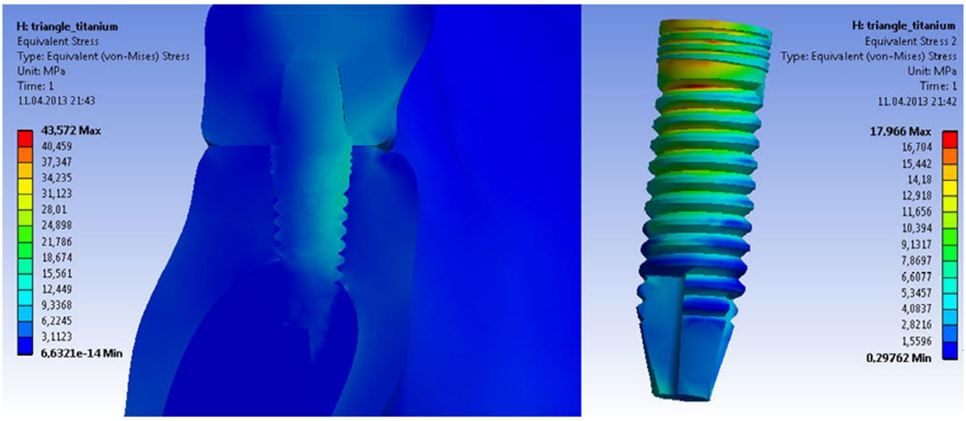
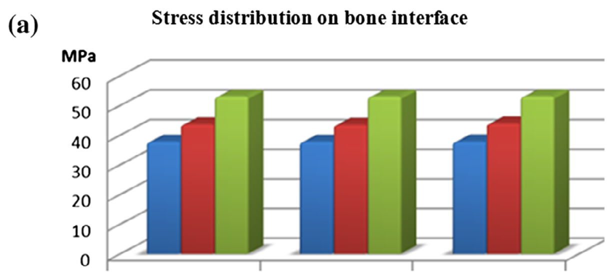
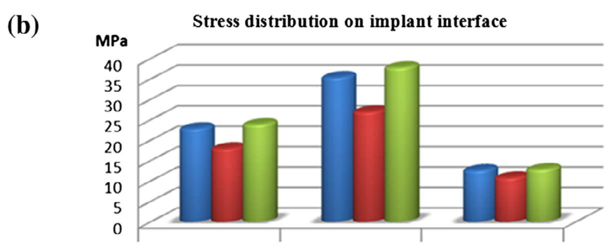


Figure 9. Stress distribution on the bone-implant interface and on the triangle thread type implant.



	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



	Titanium	Zirconium	Cadmium
Square thread	22,862	35,403	12,621
Triangle thread	17,966	26,965	10,823
Round thread	23,869	37,811	12,868

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

To summarize the results:

(1) Maximum stress value is observed on the cortical bone surface, especially first contacted side with the implant (Figures 7–9);

(2) It is determined that implant geometry with 'square' thread type causes least stress (Figure 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 (Figure 10);
(4) Stress accumulation is occurred on the hills of thread of used all of dental implants (Figures 7–9).

4. Conclusions

In this study, thread forms and materials of implants are taken as variable parameters and to compare these parameters 3D finite element stress analysis method is used. Stress distribution on the cortical, cancellous bone and implant is analysed. In this way optimal implant system is determined for patient who will be performed dental implant treatment

Disclosure statement

No potential conflict of interest was reported by the authors.

Funding

This study is supported by Çukurova University Research Fund [MMF2010BAP112].

References

[1] Masuda T, Liheikkila 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, Int J Interact Des Manuf, 2012, 6, 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 osseointegrated implants related to crestal bone geometry: A three-dimensional finite element analysis, J Prosthet Dent, 2008, 100:422–431.
[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 34, 7:817–25.
[6] Dilek M, Bircan DA, Ekşi AK. Construction of 3D finite element model of human mandible for biomechanical analyses. 2nd International Scientific Conference on Engineering "Manufacturing and Advanced Technologies" MAT 2012; 2012 November 22–24; Antalya, Turkey.
[7] Nobel Biocare Product Catalog, 2011.
[8] Geng JP, Tan KB, Liu GR. Application of finite element analysis in implant dentistry: a review of the literature. J. Prosthet. Dent. 2001 Jun;85:585–598.
[9] Gercek H. Poisson's ratio values for rocks. Int. J. Rock Mech. Min. Sci. Elsevier; 2007 Jan;44:1–13.
[10] Kurniawan D., Nor F. M., Lee H. Y., Lim Y., Finite element analysis of bone implant biomechanics: refinement through featuring various osseointegration conditions, Int. J. Oral Maxillofac. Surg. 2012, Sep, 41, 9:1090–1096.
[11] Richter EJ. Basic biomechanics of dental implants in prosthetic dentistry. J. Prosthet. Dent. 1989;61:602–609.

[12] Stegaroiu R, Kusakari H, Nishiyama S, et al. Influence of prosthesis material on stress distribution in bone and implant: a three dimensional finite element analysis. İnt. J. Oral. Maksillofac. İmplants. 1998;13:781–790.
[13] Huang H.L., Huang J.S., Ko C.C., Hsu J.T., Chang C.H., Chen M.Y.C., Effects of splinted prosthesis supported a wide implant or two implants: a three-dimensional finite element analysis, Clin. Oral Impl. Res., 2005, 16: 466–472.