Mechanical response of taper dental implants using finite element analysis

Vinamra Jain1,*, Vamsi Krishna Dommeti1, Emil Nutu2,3, Ali MERDJI3,4, Jayanta Kumar Biswas3 and Sandipan Roy1
1 Department of Mechanical Engineering, SRM Institute of Science and Technology, Kattankulathur, Chennai 603203, India
2 University Politehnica of Bucharest, Faculty of Engineering and Management of Technological Systems, Strength of Materials Department, Bucharest, Romania
3 Romanian Research and Development Institute for Gas Turbines COMOTI, Research and Development for Satellites and Space Equipment Department, Bucharest, Romania
4 Laboratory of Mechanics Physics of Materials (LMPM), Mechanical Engineering Department, Faculty of Technology, Djillali Liabes University, Sidi Bel-Abbes 22000, Algeria
5 Faculty of Sciences and Technology, University Mustapha Stambouli of Mascara, Mascara, Algeria
6 Department of Mechanical Engineering, JIS College of Engineering, Kalyani, India

Email: vinamrajain699@gmail.com

Abstract  Implants have created a great impact in the bio medical industry. There are different kinds of dental implants which are existing in the market. Every implant has a unique feature and withstands for different time period. This study on the dental implant creates a new model which includes the porous and tapered angle. Titanium alloy was used for making the implant and supporting bone tissue was considered. Computed tomography data were used to generate the mandible model. In the analysis both static and dynamic loading condition were considered. Three dimensional FEM was performed on the implant to find out the mechanical response for this specific type of implant. The von-Misses stress were calculated for implants and it was observed that safe when compared with yield strength of implant and abundant. In analysis it was also varied bone quality for cortical and cancellous bone and the effect of stress produced can be predated before the implantation done for the patients.

1. Introduction
Dentists have a number of crown and projection reclamations to choose from to provide their patients the best possible treatment. Titanium’s biocompatible and osteoinductive properties have benefited the dental implant system as they have resulted in a proper osteointegration [1]. The current treatments have seen a high success rate due to improved osteointegration.

An adequate outcome at the level of osteointegration depends on an array of variables. Firm anchorage of a dental implant’s embedding into bone tissue is influenced by its improper placement. The role of surface treatment for long term solidity of implants has been highlighted by some of the researchers [2]. With the success of embedded treatment majorly depending on auxiliary steadiness
and solidity, it is also dependent on osseointegration and a good primary stability [3]. In vivo mandibular implant retained overdentures [4] and reverberation recurrence examination [5] have been used by the researchers for measuring the solidity of dental implants. A larger surface area of contact between the implant and the bone increases stress loosening because of stack exchange along the bone-implant interface. In order to understand the relationship among different implant factors like diameter, length, shape and thread pitch, studies on behaviour of short implants were conducted [6,7]. With implant’s diameter being more influential on stress value than its length, it has also been found that trapezoidal thread profile is best suited for dental implants [8]. For a good long term stability of an implant, the determining factor is the implant’s design complimenting proper loading conditions. Bone resorption may occur at the bone-implant interface due to excessive stress generation leading to superior long term stability of implant [9,10,11]. But the implant design might not be a total success in case of pathological bone loss or failure of bone-implant interface even when the geometry of implant is designed to perform well under the assumption of osseointegration [12]. Even though, the design of implant using computational techniques has been worked upon by several research groups, a better osseointegration process occurs through an optimum 3-D structure is yet to be validated [13,14].

The objective of the current investigation is to observe the mechanical response of the tapered implant assembly when subjected to static analysis, dynamic analysis and three directional loading analysis.

2. Materials and Methods

2.1. Construction of three dimensional model.

The mandible cross section is taken from the Computerises Tomography (CT) scan and converted into a solid model as shown in figure 1(a). The section in figure 1(b) at the pre molar teeth is taken for the finite element method. The mandible of 17.2mm width and 23.2mm height is taken for the modelling as shown in fig. 1(c).

![Figure 1](image-url)

Figure 1. (a) Mandible volume with reference plane. (b) Profile Section. (c) Extruded profile section.

The cross section that is obtained from the mandible section is extruded to 12 mm and divided into two parts to form a cortical bone and a cancellous bone as shown in figure 2 and figure 3.

![Figure 2](image-url)

Figure 2. Cortical.

![Figure 3](image-url)

Figure 3. Cancellous.
There are 3 different models of implants. With the length of the implant remaining constant, the diameter varies from 3mm to 5mm (figure 4). The three models of designed implants have taper angle of 6.5 degrees [15] and the porous circle is of 0.7 mm diameter which has been included in the implant design keeping blood flow and bone growth through the pores in mind. The thread is helical in shape and has a pitch of .7 mm [15]. A locking system has also been included and fixed at the top of the implant, so the abutment may be removed whenever required. The models have been designed in PTC Creo software.

| Model | Diameter(mm) | Length(mm) | Percentage of Porosity |
|-------|-------------|------------|------------------------|
| 1     | 3           | 12         | 8.13                   |
| 2     | 4           | 12         | 6.3                    |
| 3     | 5           | 12         | 4.56                   |

Figure 4. Front view of the models with diameter 3, 4 and 5 mm respectively.

Figure 5. Isometric view of the implants with diameter 3, 4 and 5 mm respectively.

The height of the abutments are 4mm, 6mm and 8mm respectively as shown in figure 5 [6]. The cross-sectional view and the whole assembly are shown in figure 6 and figure 7. The thread isn’t continuous in this study of the implants and a gap of 3mm is left in the middle for the accommodation of pores. The percentage of porosity of all the models is mentioned in table 1.
2.2. **Finite Element Analysis.**

The 3D model that is shown in the figure 7 is imported for simulation in the software Ansys Workbench as a .STEP file. A SOLID 45 model that is designed in the designing software, the mandible which is divided as the two parts taken as cortical bone and cancellous bone. The material properties (table 2) of cortical bone, cancellous bone, implant and abutment were then assigned.

![Figure 6. Cross sectional view.](image1)

![Figure 7. Clear view of assembly.](image2)

### Table 2. Material Properties.

| Model         | Material   | Young’s Modulus(MPα) | Poisson’s Ratio | Density (gm/cm³) | Reference |
|---------------|------------|----------------------|-----------------|------------------|-----------|
| Cortical Bone | Cortical   | 14700                | 0.3             | 2.4              | [4]       |
| Cancellous    | Cancellous | 500                  | 0.3             | 1.1              | [4]       |
| Bone          |            |                      |                 |                  |           |
| Implant       | Titanium   | 110000               | 0.35            | 4780             | [4]       |
|               | Alloy      |                      |                 |                  |           |
| Abutment      | Poracellum | 68900                | 0.28            | 2.3              | [3]       |

Static analysis, dynamic analysis and three dimensional loading is performed to understand the mechanical response of the implant.

2.2.1. **Static Analysis and Dynamic Analysis.** For the static analysis the three dimensional model created in the designing software creo is imported in the ansys workbench in static structural. The material properties are given in the engineering data as mentioned in the above table 2. A compressive load of 150N is applied on the abutment (figure 8). For dynamic analysis, the end time given is 5 seconds so that the stress will vary in that interval of time. A compressive load of 150N is applied on the upper face of the abutment.
2.2.2. Three Directional Analysis. In this analysis, stress distribution patterns were compared and interfacial stresses were monitored for the bone. The load applied along x, y and z axis, respectively on the abutment is X= 23.4N, Y= 17.1N, Z= 114.6N (figure 9). This analysis is performed in order to understand the bone conditions when forces act in different directions.

2.2.3. Bone Conditions. Five different bone conditions have been considered for this study (table 3). The bone strength is different for each and every mandible.

| S.no | Bone     | Young’s Modulus (MPa) | Poisson’s Ratio |
|------|----------|-----------------------|----------------|
| 1    | Cortical | 14700                 | 0.3            |
|      | Cancellus| 500                   | 0.3            |
| 2    | Cortical | 16170                 | 0.35           |
|      | Cancellus| 550                   | 0.35           |
| 3    | Cortical | 14700                 | 0.3            |
|      | Cancellus| 550                   | 0.35           |
| 4    | Cortical | 14700                 | 0.3            |
|      | Cancellus| 450                   | 0.25           |
| 5    | Cortical | 13230                 | 0.25           |
3. Results and Discussion

The results are analysed and compared on the basis of von mises stress and maximum principle stress. Figure 10, figure 12 and figure 14 show the static analysis simulations, dynamic analysis simulations and three dimensional loading analysis simulations, respectively. Figure 11 clearly represents the results of the static analysis and figure 13 clearly represents the results of the dynamic analysis. When values of static analysis and dynamic analysis are compared, the stress values for the dynamic analysis is more than static analysis. The stress is maximum for the smaller diameter implant and minimum for the larger diameter implant and the values for the von misses stress and maximum principle stress decrease when implants’ diameter is increase.

![Figure 10. Static analysis simulations.](image1)

The Equivalent (von-mises) Stress is found out by dynamic analysis for the design Model 1 which has the implant diameter of 3 mm. The load of 150N applied on model 1 (3mm diameter) and end time of 5 seconds for 106 cycles time interval [16] resulted in maximum equivalent stress of 150.12 MPa. Here it is found that in dynamic analysis the equivalent stress value is are higher than the static stress value.

![Figure 11. Static analysis results.](image2)
Three directional loading analysis determined the stress acting along x, y and z axes for the given load. The three directional analysis for the design Model 1 has been referred here which has the implant diameter 3mm. The results of the three directional loading are shown in figure 15.

Figure 12. Dynamic analysis simulations.

Figure 13. Dynamic analysis results.

Figure 14. 3-D Loading analysis results.
Figure 15. 3-D Loading analysis results.

The mandible jaw is taken from the CT scan copy and converted into a solid model using mimics software and von mises stress and maximum principle stress have been calculated for different models. A comparative study has been performed between the available models.

CONCLUSION
In this study, static analysis, dynamic analysis and three directional loading has been performed on the implant assembly. It is concluded that the von mises stress values for implants are safe when compared with yield strength of implant and abutment. It is also concluded that relation between thread and pitch also has influence on bone damage and by varying material properties for cortical and cancellous bone, the effect of stress produced in the system can be predated before the patients are treated.

REFERENCES
[1] Branemark PI 1983 J. Prostheth. Dent.; 50 399-410
[2] Iezzi G, Piattelli A, Mangano C, Shibli JA, Vantaggiato G, Frossechi M, Di Chiara C and Perrotti V. Peri 2014 Odontolog 102 116-21
[3] Javed F, Ahmed HB, Crespi R and Romanos GE 2013 Interv. Med. Appl. Sci. 5 162-7
[4] Menicucci G, Ceruti P, Barabino E, Screti A, Bignardi C and Preti G 2006 Int. J. Prosthodont. 19 6
[5] Shokri M and Daraeighadikolaei 2003 Int. J. Dent. 2003
[6] Figueirêdo EP, Sigua-Rodriguez EA, Pimentel MJ, Oliveira Moreira AR, Nóbilo MA and de Albergaria-Barbosa JR, 2014 Int. J. Dent.
[7] Reingewirtz Y and Tenenbaum H 2015 Journal of Oral Implants
[8] Jain V, Domneti VK, Rana M and Roy S 2019 J. Long Term Eff. Med. Implants 29
[9] Berglundh T, Abrahamsson I, Lang NP and Lindhe J 2003 Clin. Oral Implants Res. 14 251-62
[10] Coelho PG, Suzuki M, Marin C, Granato R, Gil LF, Tovar N, Jimbo R, Neiva R and Bonfante EA 2015 In Engineering Mineralized and Load Bearing Tissues Springer, Cham pp. 111-128
[11] Misch CE, Bidez MW and Sharawy M 2001 J. Periodontol. 2001 72 1276-86
[12] Wolff J, Narra N, Antalainen AK, Valášek J, Kaiser J, Sándor GK and Marcián P 2014 Mater. Des. 1 77-84
[13] Roy S, Panda D, Khutia N and Chowdhury AR 2014 Int. J. Biomater
[14] Roy S, Dey S, Khutia N, Chowdhury AR, and Datta S 2018 Appl. Soft Comput. 1 272-9
[15] Franciosa P and Martorelli M. 2012 Int. J. Interact. Des. Manuf. 6 123-9
[16] Chang CL, Chen CS, Huang CH and Hsu ML 2012 *Med. Eng. Phys.* 1 *34* 999-1008.