Topological Optimization of Direct Metal Laser Sintered Ti64 Dental Implants

A. Suresh Babu, M. Jaivignesh*, R. Subramanian, M. P. Dilipkumar, R. Loganathan and S. Vijayakumar

Department of Manufacturing Engineering, College of Engineering Guindy, Anna University, Chennai – 600 025, India.

Abstract

Topology optimization (TO) is a mathematical method that optimizes the material layout within a given design space, for a given set of loads, boundary conditions and constraints with the goal of maximizing the performance of the system. In recent years, many attempts have been made to optimize the shape of dental implants. This study took advantage of the TO method to reduce material distribution on a threaded premolar dental implant. The implant macro geometry is redesigned by TO with the evaluation of its biomechanical functions and the volume of the traditional implant is decreased by approximately 19.11%. Still this new design is able to afford as much stiffness (of the implant–bone complex) as the traditional implant. The surface area of the newly designed dental implant has been increased from 185.75 mm^2 (that of traditional implant) to 225.58 mm^2. Also the surface area of the bone to implant contact is increased by about 24.92 mm^2. Ti64 being biocompatible is the selected implant material and is fabricated by Direct Metal Laser Sintering process. The fabricated implant is tested for compressive strength and the implant withstood a load of 900 N which is more than four times the required load to be withstood on a premolar dental implant (210 N). Hence, the designed implant can be concluded as safe. The advantage of new implant is that it increases the surface area to allow more new bone in-growth, thus sustaining implant stability and also cost of implant is decreased due to reduced material consumption.

Keywords: Topology Optimization, Ti 64, Direct Metal Laser Sintering, Dental Implant, Compressive Strength

1. INTRODUCTION

In recent years, many attempts have been made to optimize the shape of dental implants. The purpose of this study took advantage of the topology optimization in the finite element (FE) method to look for redundant material distribution on a dental threaded implant and a new implant is designed with the evaluation of its biomechanical functions. A dental implant is a surgical component that interfaces with the bone of the jaw or skull to support a dental prosthesis such as a crown, bridge and denture. The basis for modern dental implants is a biological process called osseointegration, in which materials such as titanium form an intimate bond with bone. Using topology optimization method, material consumption of the implant is reduced. Additive manufacturing is a proven technology for fabrication of implants and Ti64 is an ideal material for many high performance engineering applications especially for the production of biomedical implants subject to fulfilment of appropriate statutory validation requirements. Compression strength is important criteria of evaluation for a dental implant. So in this work, Ti64 dental implant is fabricated by direct metal laser sintering technique and is tested for compression loading condition.

2. DESIGN OF IMPLANT

The preliminary model of the dental implant is designed with 14 mm of cylinder height and 3.5 mm of cylinder diameter as whole which includes thread of height 7.5 mm [4]. The model dimensions are further researched for its basic performance targets to be achieved. The implant dimensions are finalized as cylinder of height 11 mm, diameter 3.5 mm and a frustum with lower base diameter as 2.6 mm at the end, to hold the tooth. The cylinder carries helical thread with pitch of 0.6 mm which runs for 8mm. Figure 1 shows designed dental implant.

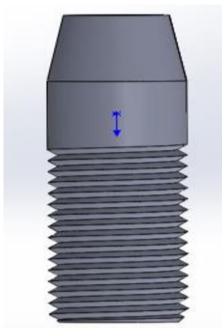


Fig. 1. Designed Dental Implant

3. SELECTION OF MATERIAL

Ti64 is selected based on biocompatibility and it has a good corrosion resistance. Table 1 shows material composition of Ti64. Desirable mechanical properties of Ti64 which are given as input for finite element analysis are tabulated in table 2.

Table 1

Material composition of Ti64

Material	Composition (%)
Titanium	balance
Aluminium	5.5 - 6.75
Vanadium	3.5 - 4.5
Oxygen	< 2000 ppm
Nitrogen	< 500 ppm
Carbon	< 800 ppm
Hydrogen	< 150 ppm
Iron	< 3000 ppm

* Author to whom correspondence should be made, Email: jaivignesh.murali@gmail.com

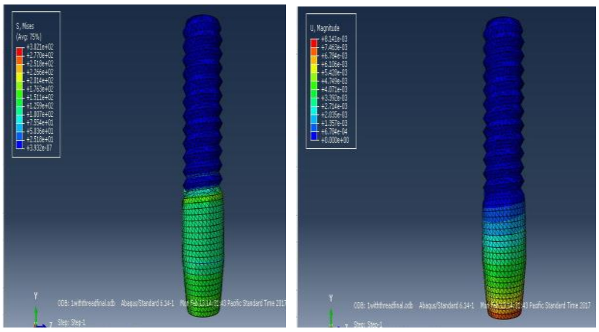
ISBN: 978-93-80689-28-9

Table 2. Desirable mechanical properties of Ti64

Property	Value (units)
Density	4.41 g/cm^3
Young's modulus	113.8 GPa
Tensile Strength	1290 MPa
Yield Strength	1140 MPa
Elongation at Break	7 %
Poisson's Ratio	0.342

4. FINITE ELEMENT ANALYSIS

The designed implant is analyzed for its performance by applying compressive load of 200 N, which is the maximum load acting on the premolar tooth during biting and chewing action. The Maximum stress distributed on the dental implant is 302.1 N/mm^2 and the maximum displacement suffered by implant design during analysis is 8.14 µm. The stress distributed (figure 2a) and the magnitude of displacement (figure 2b) are shown below. The Von-mises stress is used to predict yielding of materials under any loading condition in general and the same is used in this work to find the suitability of design. Since it is below the ultimate strength of Ti64, the design is concluded as safe and is suitable for design optimization.

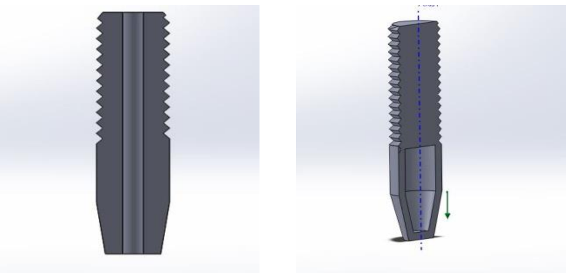


a) Stress distribution **b) Magnitude of displacement**

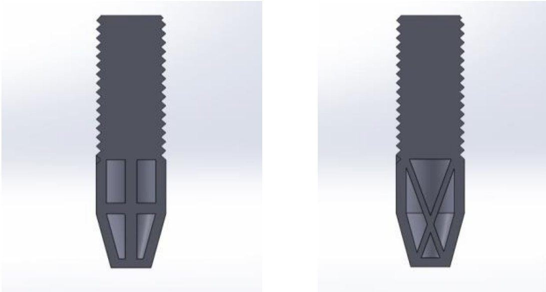
Fig. 2. Finite element analysis of dental implant model

5. SHAPE OPTMIZATION

In shape optimization, the shape of the structure is obtained by changing the shape of the used components with other components of different shape, in order to improve a desired variable within a system. The dental implant model is shape optimized on different types which are listed below (Figure 3a, 3b, 3c and 3d).



a) Model 1 **b) Model 2**



c) Model 3 **d) Model 4**

Fig. 3. Shape optimization models

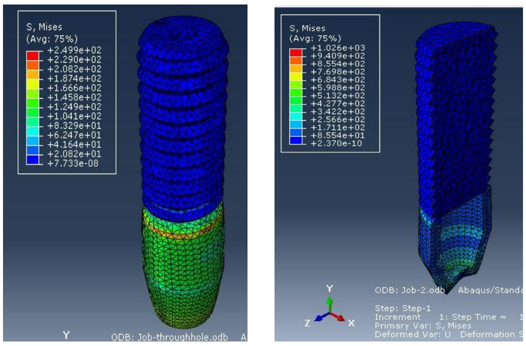
The shape optimization models are selected based on the performance criteria and constraints such as stress distribution and magnitude of displacement.

- Model 1 – Implant with through hole
- Model 2 – Implant with Hollow profile at bottom
- Model 3 – Implant with straight rib
- Model 4 – Implant with cross rib

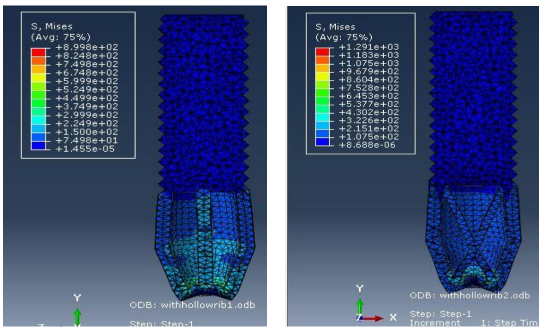
In this work, shape optimization is useful for implant in order to come up with a preliminary design with reduced material consumption before topologically optimizing it, since the implant geometry is smaller and also has lot of constraints in design and volume reduction.

6. ANALYSIS OF SHAPE OPTIMIZED MODELS

The shape optimized models are analyzed using Abaqus for its performance characteristics with the constraints such as stress distribution and magnitude of displacement. For a load of 200 N, the stress distribution for different type of shape optimized models are shown in the figures 4a, 4b, 4c and 4d and the magnitude of displacement in shape optimized models are shown in figures 5a, 5b, 5c and 5d.

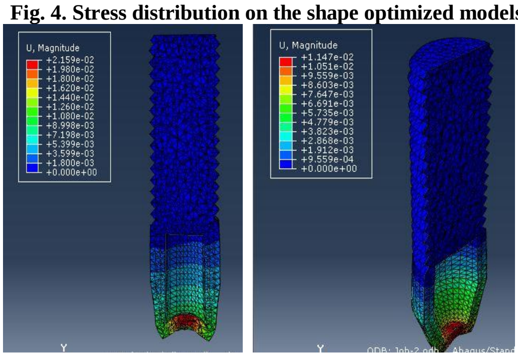


a) Model 1 **b) Model 2**

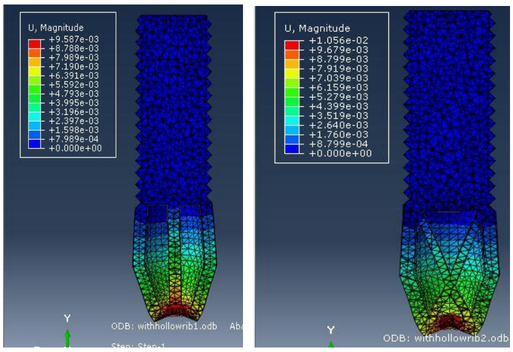


c) Model 3 **d) Model 4**

Fig. 4. Stress distribution on the shape optimized models



a) Model 1 b) Model 2



c) Model 3 d) Model 4

Fig. 5. Magnitude of displacement on dental implant

The various models, which are shape optimized, are studied for its performance characteristics under real time conditions. The stress distribution and magnitude of displacement pattern are analyzed for the selection of optimal shape optimized model. The inference from the shape optimization process is tabulated in table 3.

Table 3. Inference from shape optimization process

Models	Magnitude of Displacement (x 10^{-2} mm)	Stress Distribution (x 10^{2} N/mm^{2})
Model 1	2.159	2.49
Model 2	1.147	10.26
Model 3	0.9587	8.998
Model 4	1.056	12.91

Based on the constraints like stress distribution and magnitude of displacement, the dental implant model with through hole i.e model 1 has the optimum or nearby value when compared with the original dental implant model designed. Hence this implant with through hole model is taken for further considerations and topological optimization is carried out on that.

7. TOPOLOGICAL OPTIMIZATION OF DENTAL IMPLANT

The whole model is topologically optimized using Abaqus, by using basic constraints such as freezing load applied regions and boundary condition regions (figure 6). This is done because there should be no material loss in surfaces where boundary conditions are applied. The lower the strain energy, higher is the stiffness of the structure. So, the problem statement involves the objective functional of strain energy which is to be minimized. The volume fraction (i.e. the volume of the component to be reduced) is given as design response value.

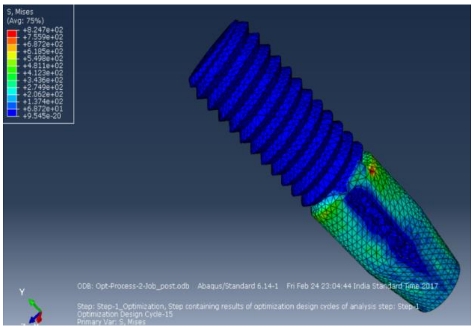
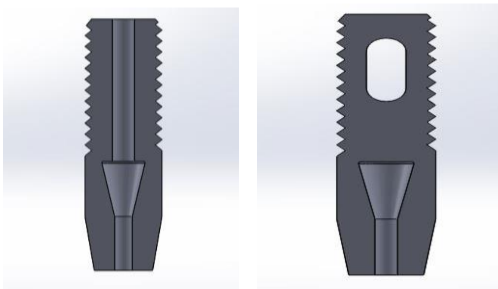


Fig. 6. Topology Optimization of model for volume fraction ≤ 0.48

8. REMODELLING THE TOPOLOGICALLY OPTIMIZED DESIGN

The final topologically optimized model has a void in the surface, which can cause dental infection. So, as per suggestions from medical experts, the void is removed and based on Topologically Optimized design, the original design of dental implant is remodeled as shown in figure 7a.



a) Redesign based on TO **b) Final remodeled design**

Fig. 7. Remodeling of Dental implant

The remodeled design is further studied and consulted with doctors. The suggestion is to replace the through hole in the stem with vent which has a proven application in implant design. Based on their suggestions, the implant is remodeled as shown in figure 7b. The purpose of the vent is to increase the contact surface area of the implant with the bone, which allows growth of new bone into it. This increases the stability and life time of the implant thus making it a more osseointegrated implant.

9. FABRICATION OF DENTAL IMPLANT MODEL

The dental implant model which is topologically optimized and remodeled based on suggestions and constraints is fabricated with Ti64 alloy by Direct Metal Laser Sintering process where the laser (with 400 W power) sinters the powder to build up the part layer by layer. After fabrication, they are subjected to heat treatment at 800 ^{o}C for 8 hours to remove residual stresses and the support structure is removed by machining. Finally the surface is cleaned by means of sandblasting. Figure 8 shows the fabricated specimen.

Fig. 8. Implant specimen fabricated by DMLS process

10. COMPRESSION TESTING OF DENTAL IMPLANT SPECIMEN

The dental implant specimen is tested for its compressive behaviour. Since the maximum compressive load acting on the tooth during biting and chewing action is 200N, the load condition is fixed based on that. The specimen is placed in the testing table and load is applied gradually. A maximum of 900 N (which is more than four times the load required to be withstood), is applied. Figure 9 shows compression testing set up used for specimen.

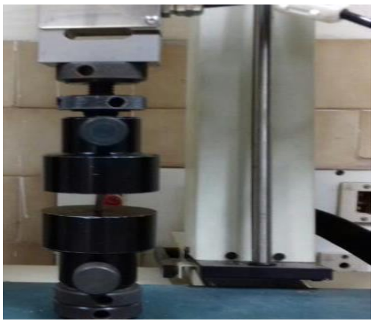


Fig. 9. Compression testing set up used for specimen

11. COMPARISON OF RESULTS

The Topologically Optimized model is compared with the original dental implant model and the results observed are discussed in the table 4.

Table 4. Results Inferred

Parameter	Original dental implant	Topologically Optimized model
Volume	116.61 mm^3	94.32 mm^3
Surface area	185.75 mm^2	225.58 mm^2
Maximum stress	302.1 N/mm^2	361.2 N/mm^2
Maximum displacement	8.14 µm	8.43 µm

The volume of the dental implant model is reduced from 116.61 mm^3 to 94.32 mm^3. The final volume reduction of the modified implant design is by 19.11%. The surface area of the dental implant has been increased from 185.75 mm^2 to 225.58 mm^2. The surface area of the bone to implant contact increased by about 24.92 mm^2 in the final design of this new implant, which is thus available to transmit more of the compressive load to the bone. The more the contact of implant with the bone, more the implant is osseointegrated, thus making the implant more sustainable. The maximum compressive load acting on the tooth during biting and chewing action is 200 N. But, testing for a load of 900N doesn't crack the specimen. Thus, the specimen is stable for load higher than the required value to be withstood and produces only less stress than the allowable stress of 895 N/mm^2.

12. CONCLUSIONS

The new implant is shaped by topology optimization and it decreases the volume of the traditional implant approximately by 19.11% and this new design is still able to afford as much stiffness of the implant–bone complex as the traditional implant. The surface area of the dental implant has been increased. The surface area of the bone to implant contact is increased in the final design of implant by about 24.92 mm^2. Additionally, the biomechanical parameters of the newly modified implant generated an almost similar performance, in terms of displacement and stress of the implant, as the traditional one. The advantage of the new implant with its large concavity is that it increased the space to allow more new bone ingrowth or assist in fusing more bone graft into the bone, thus sustaining implant stability, reducing the material costs of the implant and producing less displacement than the existing implant.

References

[1] Abbas A. and Nikzad S., 'The evolution of rapid prototyping in dentistry: a review', *Rapid Prototyping Journal*, **15** (2009) 216–225.

[2] Adindu E. and Marufuzzamana M., 'Additive manufacturing of biomedical implants: A feasibility assessment via supply-chain cost analysis', *Additive Manufacturing*, **11** (2016) 97–113.

[3] Bicudo P. and Reis J., 'Performance evaluation of dental implants: An experimental and numerical simulation study', *Theoretical and Applied Fracture Mechanics*, **85** (2016) 74–83.

[4] Chang C. and Chen C., 'Finite element analysis of the dental implant using a topology optimization Method', *Medical Engineering & Physics*, **34** (2012) 999– 1008.

[5] Chun H. and Cheong Y., 'Evaluation of design parameters of osseointegrated dental implants using finite element analysis', *Journal of Oral Rehabilitation*, **29** (2002) 565–574.

[6] Daniel H. and Andreana S., 'The role of the finite element model in dental implants', *Journal of Oral Implantology*, **26** (2000) 77-81.

[7] Gilbert C. and Koike M., 'The Design and Production of Ti-6Al-4V ELI Customized Dental Implants', Journal of the Minerals, Metals & Materials Society, **60** (2008) 50-55.

[8] Gustav S. and Christer D., 'The bone-implant interface of dental implants in humans on the atomic Scale', *Acta Biomaterialia*, **48** (2017) 445–450.

[9] Stig H. and Johanna L., 'Global biomechanical model for dental implants', *Journal of Biomechanics*, **44** (2011) 1059–1065.

[10] Xiuli H. and Franz R., 'Analysis of titanium and other metals in human jawbones with dental implants – A case series study', *Dental materials*, **32** (2016) 1042-1051.