IJFANS INTERNATIONAL JOURNAL OF FOOD AND NUTRITIONAL SCIENCES
ISSN PRINT 2319 1775 Online 2320 7876
Research Paper , Journal Volume 11, Iss 09, 2022

MODELLING AND STATIC ANALYSIS OF DENTAL IMPLANT USING DIFFERENT BIOMEDICAL MATERIALS

[1]Mr. Kuna Naresh Raj,[2]Dr. Vodnala Veda Prakash,[3]Mr. P.Shivaraj
[123]Assistant Professor
Department Of Mechanical Engineering
Kshatriya College of Engineering

ABSTRACT

A Dental implant is a structure that replaces a missing tooth. With screw-likedevices, the surgeon inserts an implant into the jawbone, and it acts as an anchor for anartificial tooth, called a crown. A device called an abutment connects the artificial tooth tothe dentalimplant. The osseointegration of bone and implant at the interface is of utmost importanceas the success or failure of a dental implant depends on the manner in which stresses aretransferred to the surrounding bone. This could cause micro-fracture at the bone-implantinterface, fracture of implant, loosening of components of implant system and unwantedboneresorption.Therefore,itisessentialtounderstandstressconcentrationonimplantsatthe boneimplantinterface.Minimal Stress concentration on the contactsurface between the denture andvarious implants, abutment, and crown gives a quality of life to the implant. This can beachievedbymodellingandselecting theproperprostheticmaterial. A fully parametric 3D model of the Dental implant is created in SOLIDWORKS2016 software. Analysis of stress concentration on dental implant had done by usingANSYS 16.0. In this project work, four different materials YSZ, ABS, SS316L, TI-6AL-4V, which were used for analysis of dental implant using ANSYS 16.0 software in staticanalysismodule to findoutequivalentstresses&strains,totaldeformation. The objective of the project is to evaluate the performance of four different typesof materials and to select the best suitable material for dental implant by comparing thesefourmaterials.

I. INTRODUCTION OF DENTISTRY

Dental implants, which are used to replace lost teeth, play a significant function in dentistry. Dental implants are titanium posts that are surgically implanted into a patient's jawbone and to which an artificial tooth crown is affixed. The link between the implant and the bone must be strong enough to withstand stress from various loading circumstances. The success of an implant is determined by two essential factors: design and insertion technique.

In the biomechanical optimization of dental implants, thread form and geometry are key considerations. Threads are used to increase preliminary contact, improve initial stability, increase implant outer area, and promote interfacial stress dissipation. To increase clinical success, it is necessary to assess the thread design of dental implants.

The goal of the implant, according to the invention, is to achieve mechanical properties and stress distribution in bone that are similar to natural tooth attributes. Excessive tension from the implant to the bone can cause injury. As a result, the interface threaded component of the implant must be self-tapping for greater integrity and increased contact surface area for a better bone implant. Many elements have been discovered that influence the interfacial bonding between the implant and the bone, as well as the implant's success.

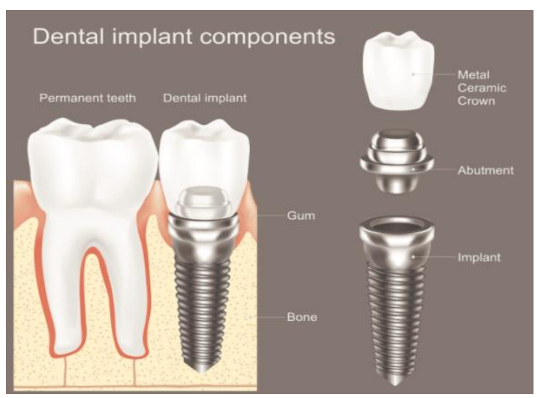


Figure 1 components of dental parts

TEETH ARE LOST BECAUSE OF:

- Tooth decay
- Root canal failure
- Gum disease (Periodontitis)
- Trauma to the mouth (tooth injury)
- Excessive wear and tear

IJFANS International Journal of Food And Nutritional Sciences Official Publication of International Association of Food and Nutrition Scientists

ISSN PRINT 2319 1775 Online 2320 7876
 Journal Volume 11, Iss 09, 2022

- Congenital defects

Because modeling a complete mandible is difficult, we employ a portion of the mandible that is considerably easier to model. We will use a cut portion of the mandible for our investigation, as illustrated below. A fragment of the mandible bone is chosen. The geometry of bone has been reduced and simulated as a rectangle, or brick, made up of two layers of bone. The inner bone is a spongy bone (size= 15 X 20 X 15 mm) that occupies the inner space of the outer bone, which is a cortical bone with thicknesses of 2mm.

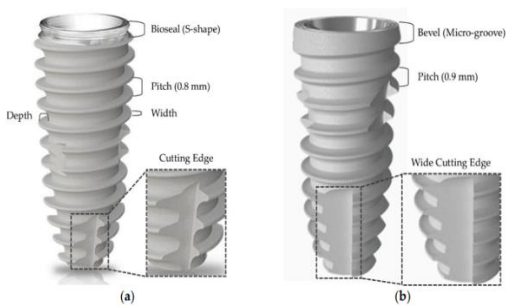


Figure 2implant macro design parameters in dental fixtures

Implant with progressive threads (greater depth in apical portion and lesser depth in coronal portion)

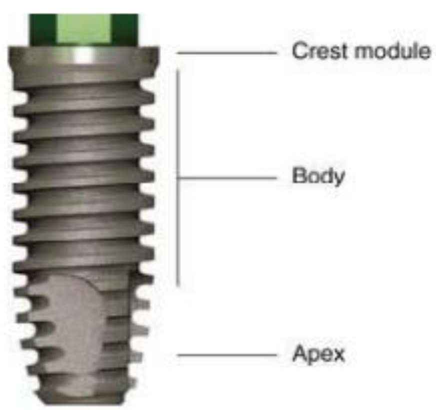


Figure 3 crest modul

APPLICATIONS OF THE DENTAL IMPLANTS

- Ability to bite and talk effectively
- Restoration of facial aesthetics
- Preservation of the ultimate jaw bone
- Restoration of the natural appearance and feature of tooth
- Increased confidence and self-self-assurance

This organic fusion system is called osseointegration and might help extensivephysical stress for extended periods of timesafe and effective manner to update the appearance and characteristic of missing enamel.

II. LITERATURE REVIEW

1. By building a 3D solid model of an Indigenous titanium dental implant and mandible and applying it to FEA using Ansys, Salujaet colleagues investigated the effect of length and diameter on stress distribution. Stress concentration and distribution are unaffected by implant length. As the diameter of the implant grows, the surface contact area grows, making the implant more stable and minimizing the stress pattern.

2. The influence of different thread types on loading in the implant abutment interface was examined by Arsalanlo et al. To better understand stress distribution, FEA was used to analyses models with changing diameter and length. They discovered that increasing thread depth improves stress distribution while decreasing thread pitch affects implant stability.

3. By modelling three screw type dental implant systems with three fixed screw diameter sizes and subjecting them to a static occlusal force of 100N with 15o for FEA, Zhidong Mao analyzed the influence of abutment and fixed screw on dental implant system. When the abutment and fixed screw are the wrong size, stress builds up.

4. By examining and comparing ten various types of implants, Giuseppe et al investigated the impact of implant design characteristics such as diameter, thread type, and length on the load transmission mechanism of Osseo integration in dental implants (varying length, diameter, thread shape and geometry). He came to the conclusion that design elements in load transfer on implants, such as implant diameter, length, and thread form, have a significant impact.

5. In FEA and experimental research, Ghorpade et al conducted a literature review to examine the influence of design parameters related to the bone–implant interface on stress distribution. The modelling method and implant geometry have an impact on the accuracy of the study.

6. Ausiello et al looked at the impact of implant design parameters on implant longevity. A DOE technique was used to generate different implant designs automatically and analyses them using FEA.

By linking design characteristics, implant stability is influenced by thread width and thickness.

7. Using various dental implant designs, Desai et al assessed stresses in the bone implant interface. Modeling and analysis of eight distinct types of implants with various thread patterns in order to assess stress and strains surrounding the implants.

8. Bahrami et al investigated stress distribution in the interaction between bone implants and dental prostheses. More thread depth can relieve stress in surrounding bone by increasing contact surface area.

9. Szajek et al. present a way for reducing the diameter of a two-component implantology system. The hybrid optimization approach was used to optimize a two-component implant model based on FE analysis (genetic and HookeJevees algorithm). The hybrid approach provided here optimized diameter while maintaining defined limits.

10. Mohammed et al investigated stress distribution in dental implants by modelling and analyzing various implant designs with changing length and diameter under 50N tensile, 100N compressive, and 20N bending loads. The load transfer mechanism is affected by implant diameter and length, crestal bone shape, and implantation site.

11. Using 3D finite element analysis, Kong et colleagues assessed the cylinder implant thread height and width. He came to the conclusion that when designing a screw type implant, proper thread height is more significant than thread width for reducing bone stress.

12. Mansour et al used three-dimensional finite element stress analysis to investigate stress distribution around tapered and cylindrical threaded implant designs.

13. Baggi et al looked into how implant diameter and length affected stress distribution. The weight transmission mechanism may be affected by implant shape, geometry, and bone resorption, resulting in failure.

OBJECTIVE OF THE PROJECT

Stress concentration on the contact surface between the Denture and various Implants, Abutment, and Crown is minimised. Finally, using the SOLIDWORKS ANSYS programme, design and static analysis are performed before selecting the appropriate Prosthetic Implant Material.

SPECIFIC OBJECTIVES:

1. A general assessment of mechanical reliability in terms of stress concentration for dental implants.

2. Using FEM to determine the best material to use.

3. Examining the stress concentration on various geometries of dental implants prosthesis Cylindrical Shape taper implant

4. Finally, utilizing various materials Ti-6AL-4V, YTZ, SS 316 L, ABS find out stress, total deformation, strain in static analysis

5. Recommending best material for the use in future implant surgeries.

III. METHODOLOGY

Step 1:collecting data and information on different dental implants.

Step 2: Create a fully parametric model of the artificialdental implants in SOLIDWORKS software, Cylindrical Shape taper implant

Step 3: In igs file imported in the ANSYS software (workbench) was used to observe stresses, Deformation, strain and shear stress for the following materialsTi-6AL-4V, YTZ, SS 316 L, ABS.

Step 4: Apply bucual load at to surface 100N as a boundary condition

Step 5: Finally find out the material is the best based on the values.

PROBLEMS OF THE DENTAL IMPLANTS:

A dental implant is a long-term tooth replacement that replaces a missing tooth. The implant is a titanium screw that is inserted into the jawbone by a dentist. The implant and jawbone fuse together over a period of weeks. Once united, the implant can support a set of artificial teeth or a crown.

Around 3 million people in the United States have dental implants, according to the American Academy of Implant Dentistry (AAID). Dental implants are likewise becoming increasingly popular. According to the AAID, the number of persons getting them is growing at a rate of roughly 500,000 each year.

This page discusses the capability issues and long-term problems that might arise as a result of DIS. It also includes information on implant success rates, aftercare, and recovery time.

COMMON PROBLEMS:

Some of the more typical issues that may arise as a result of DIS are listed below.

a. INFECTION
b. RECESSION OF GUM
c. IMPLANT WITH LOOSE IMPLANT
d. NERVE OR TISSUE DAMAGE

MATERIAL PROPERTIES:

MATERIAL	DENSITY (Kg/m³)	YOUNGS MODULUS (GPa)	POISSONS RATIO	ULTIMATE TENSILE STRENGT (MPa)
AISI 316L SS	8000	193	0.25	515
YSZ	8500	200	0.33	517.1
TI-6AL-4V	4420	114	0.32	930
ABS	1200	1.47	0.3	50.8

Table 1 material properties

DESIGN PROCEDURE OF IMPLANT IN SOLIDWORKS:

Initially open the SOLIDWORKS EXPLORER 2020 on computer and start a new part in sketch. A complete Tapered screw thread implant design was developed by using some line features then mark all dimensions as per standards of implants as per below figure.Once develop the sketch then apply revolved boss base feature as per its axis of rotation as its centerline of $360^{0.}$ Then after taking a reference of 1.8 mm to center of top plane and make a duplicate plane and draw a circle of 1.40 mm diameter for the helix base circle as shown in below figure.Now, drawn a helix by reference of above mentioned base circle from the feature tab of curves and helix. Apply a helix pitch of 0.50 mm and 12 numbers of revolutions are applied on it and finally make it as helix with groove structure by applying a swept cut feature.

Figure 4 Multi sectional view of a tapered implant screw thread

MESHING:

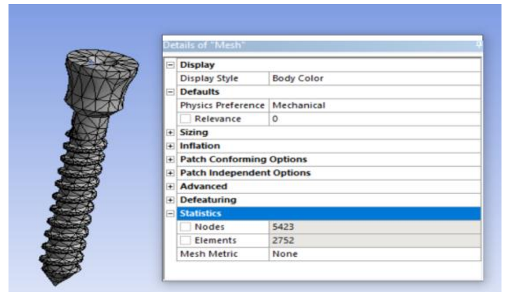


Figure 2nodes: 5423 and elements: 2752

BOUNDARY CONDITIONS:

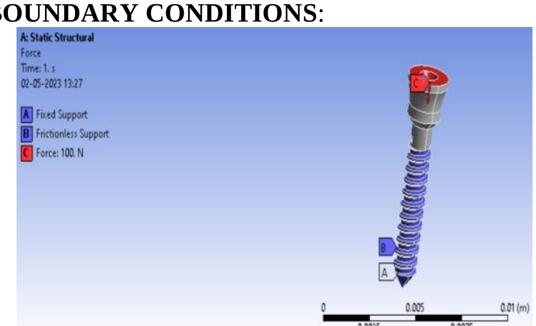


Figure 6 boundary conditions

After relating all boundary condition & material properties FEA model is solved with the help of ANSYS 18.0. We get the results of FEA analysis after achievement of solving procedure in ANSYS are as follows.

IV. RESULTS AND DISCUSSIONS

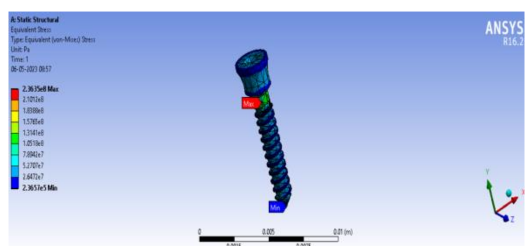
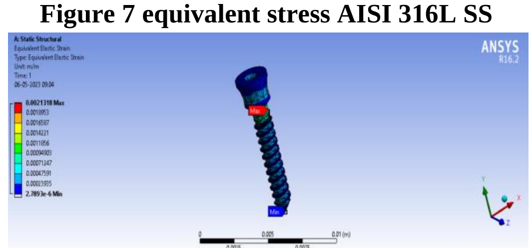


Figure 7 equivalent stress AISI 316L SS



IJFANS INTERNATIONAL JOURNAL OF FOOD AND NUTRITIONAL SCIENCES
ISSN PRINT 2319 1775 Online 2320 7876
 Journal Volume 11, Iss 09, 2022

Figure 8 equivalent strain AISI 316L SS

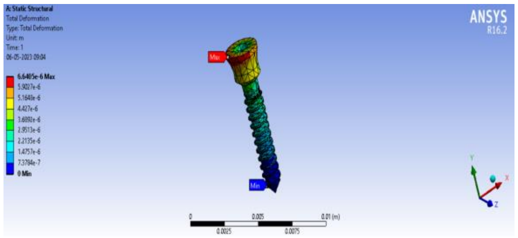


Figure 9 total deformation AISI 316L SS

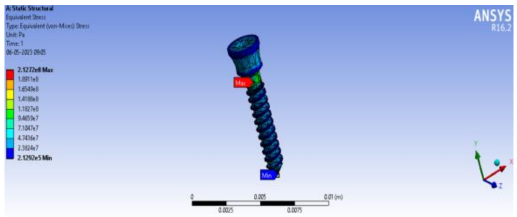


Figure 10 equivalent stress YSZ

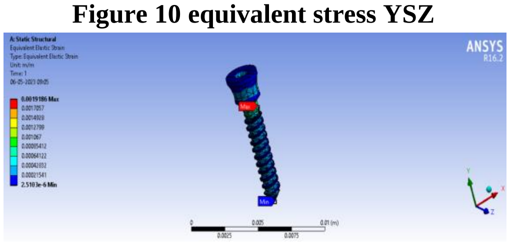


Figure 11 equivalent strainYSZ

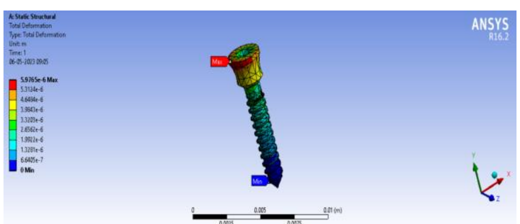


Figure 12 total deformation YSZ

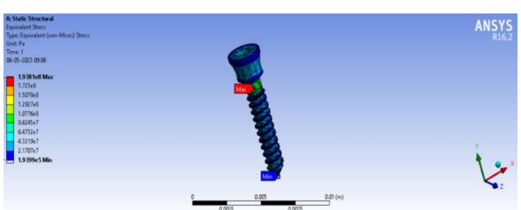


Figure 13 equivalent stresses Ti-6Al-4V

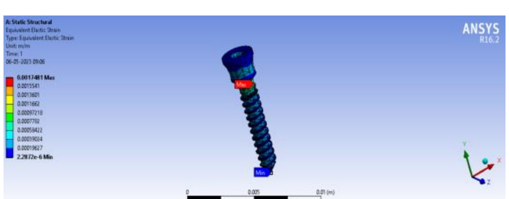


Figure 14 equivalent strain Ti-6Al-4V

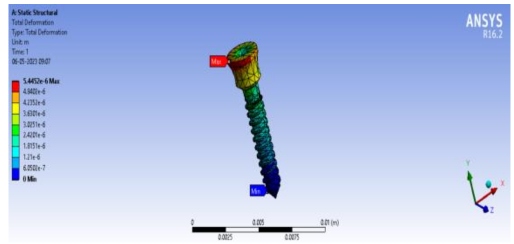


Figure 15 total deformation Ti-6Al-4V

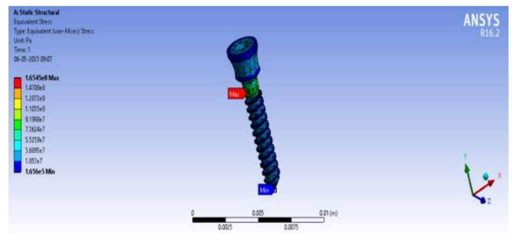


Figure 16 equivalent stressABS

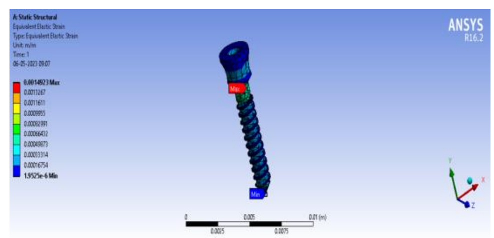


Figure 17 equivalent strainABS

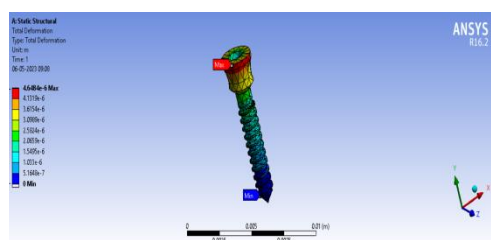
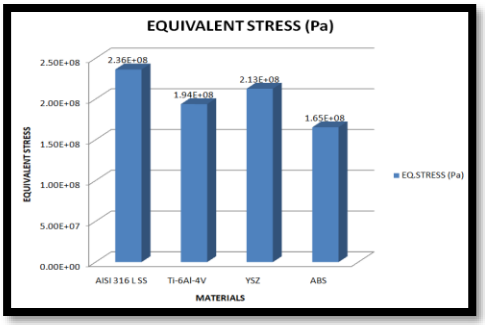


Figure 18 total deformation ABS

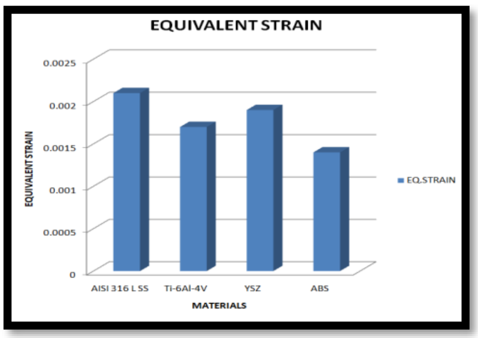
RESULTS AND DISSCUSSION

MATERIAL	EQ.STRESS (Pa)	TOTAL DEFORMATION(m)	EQ.STRAIN
AISI 316 L SS	2.363e8	6.40e-6	0.0021
Ti-6Al-4V	1.938e8	5.44e-6	0.0017
YSZ	2.127e8	5.97e-6	0.0019
ABS	1.654e8	4.64e-6	0.0014

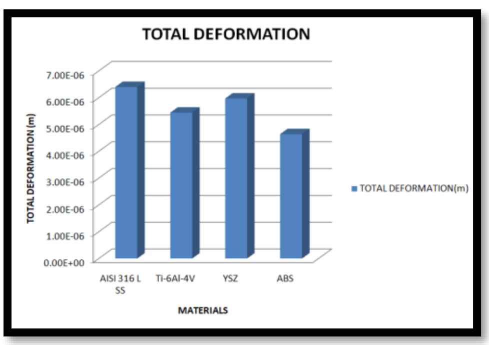
Table: static structural analysis results



Graph 1 equivalent stress graph



Graph 2 equivalent strain



Graph 3 total deformation

V. CONCLUSION

Design of dental denture, implant, teeth abutment using in SOLIDWORKS software and perform the static analysis in Ansys software. After Design and analysis finally find out the von misses stress, strain, total deformation and shear stress analysis on this dental implant's cylindrical taper implant using various material likely**TI-6AL-4V, SS 316 L, YSZ, ABS Materials** we are observing **ABS**materialas a better structure for all stress, strain, shear stress and total deformation point of view in static analysis In all tooth abutment designs, a homogeneous, isotropic, and linear elastic material is used which is titanium alloy because it has the best biocompatibility, bonding strength, and corrosion resistance.For better understanding, the biomechanics of dental implants and the use of computer technology alongside with more profound awareness about the concept, methodology, advantages, and limitations of FEA have to be assessed elaborately. As a result, clinicians can use this modern technology to enhance implant survival by well accepting the biomechanics of dental implantlogy. Similar to any other instrument used to resolve a problem, the explanation made can only be as robust as the suitable application of the instrument itself. Upcoming investigation ought to attempt to correlate results with clinical findings. In doing so, it enhances the validity of the models. In addition, simulate the consequence of saliva, infection.

REFERENCES

[1]. BobinSaluja, MasoodAlam, T Ravindranath, A Mubeen et.al , Effect of length and diameter on stress distribution pattern of INDIDENT dental implants by finite element analysis , Journal of Dental Implants, Vol 2 , Issue 1, PP 19-25, 2012.

[2]. ZeinabArsalanloo, Reza Telchi, and KambizGhaemiOsgouie, Selection of Optimum Thread Type in Implants to Achieve Optimal Biomechanical Properties by Using 3D Finite Element Method , International Journal of Bioscience, Biochemistry and Bioinformatics, Vol. 4, Issue 3, PP 185 – 190,May 2014

[3]. Zhihong Mao, Nanning, Design of dental implant system by FEA, 2014 IEEE Workshop on Electronics, Computer and Applications, 978-1-4799-4565-8, PP 675-677, 2014.

[4]. Giuseppe Vairo and GianpaoloSannino, Comparative Evaluation of Osseointegrated Dental Implants Based on Platform - Switching Concept : Influence of Diameter, Length, Thread Shape, and In-Bone Positioning Depth on Stress-Based Performance, Computational and Mathematical Methods in Medicine ,Volume 2013, Article ID 250929,PP 1-15,2013.

[5]. Ratnakar R. Ghorpade , Shailesh S. Yelekar, Computational And Experimental Studies In Threaded Dental Implant Research , e-Journal of Dentistry , Vol 3, Issue 4, PP 457 -465 , Oct - Dec 2013

[6]. PietroAusiello, Pasquale Franciosa , Massimo Martorelli , David C. Watts, Effects of thread features in osseo-integrated titanium implants using a statisticsbased finite element method ,ELSEVIER, Dental materials , Issue 28,PP 919–927(2012).

[7]. S.R Desai, MS Desai, G Katti, I Karthikeyan, Evaluation Of Design Parameters Of Eight Dental

Implant Designs: A Two-Dimensional Finite Element Analysis, NJPC Vol. 15,Issue 2,PP 176-181, 2012

[8]. PRAKASH, V. V., & KUMAR, B. S. IMPROVING ORC PLANT EFFICIENCY BY COMBUSTION PROCESS.

[9]. Krzysztof Szajek, MarcinWierszycki and Tomasz Łodygowski, Reduction of the tooth-implant components dimensions by optimization procedure, CMM-2011 – Computer Methods in Mechanics, Vol.4, 9–12 May 2011.

[10]. Mohamed I. El-Anwar, Mohamed M. El-Zawahry, A Three-dimensional Finite Element Study On Dental Implant Design,Journal of Genetic Engineering and Biotechnology, Issue 9, PP 77–82, 2011.

[11]. Liang Kong, Kaijin Hu, Dehua Li, Yingliang Song, Evaluation of Cylinder Implant Thread Height and Width – 3D Finite Element Analysis,The International Journal of Oral and Maxillofollical Implants, Volume 23,Number 1,PP 65-74,2008.

[12]. Luigi Baggi, IlariaCappelloni, Michele Di Girolamo, Franco Maceri, and Giuseppe Vairo, The influence of implant diameter and length on stress distribution of osseointegrated implants related to crestal bone geometry: A three dimensional finite element analysis, The Journal of Prosthetic Dentistry, Volume 10, Issue 6, pp 423-431, Dec 2008.

[13]. , A. K., Laxmaiah, G., & Babu, P. R. Process Parameters Optimization And Characterization Of RTM Manufacturing Process For High Performance Composites.

[14]. IniyanThiruselvam N, Dhananjay M. Kulkarni and Savio D.S.A Lourenco, Preoperative simulation of implant angulation effect on mandibular masticatory stresses- a finite element study, Procedia Engineering, Vol.64, pp. 8145-824, 2013.