IJESRT
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

IMPACT OF IMPLANT PITCH AND LENGTH VARIATION ON STRESS INTENSITY AROUND DENTAL IMPLANT-BONE INTERFACE-BY 3D FINITE ELEMENT ANALYSIS

Mr.Durvesh J.Thakur *, Prof. T.Z.Quazi, Prof. Pankaj N. Dhatrak
* Department of Mechanical Engineering Saraswati College of Engg., Kharghar, Navi Mumbai, India
Department of Mechanical Engineering Saraswati College of Engg., Kharghar, Navi Mumbai, India
[2] Department of Mechanical Engineering, MAEER'S M.I.T. Pune, India

DOI: 10.5281/zenodo.47038

ABSTRACT
Purpose of the study is to determine the effect of pitch and length variation on stress intensity around dental-implant bone interface. Finite Elemental Analysis were conducted first for identification of stress intensity of v threaded commercial implant model and then used further for finding stress intensity results of each model with variation in length and pitch respectively under the combination of vertical and lateral loading conditions.Results showed that stress intensity decreased for cancellous bone when implant length increased where as magnitudeof stress intensity were incresed with increasing pitch of thread.

KEYWORDS: Dental Implants, Stress Intensity, Finite element analysis

INTRODUCTION
Dental implant is now become one of the important bio-mechanical area where lots of research is going on. Implants acts as an artificial root for tooth implantation in human jaws (Figure 1).These roots having direct contact with jaw bone and their stability depends upon the proper installation of them. The process of implantation is very complex and depends upon so biological conditions and engineering aspects. It fails generally due to improper installation, uneven stress distribution during mastication, wrong design features etc.

Figure 1. Dental Implant

Researcher's have already worked on the design parameters of dental implants such as thread patterns, taper angle, and Inner outer diameter etc. and their impact on stresses around dental implant bone interface[11][9][1]. They mostly preferred finite elemental analysis as an effective tool for finding the stresses around the implants under various loading conditions[19].

Most of the dentists had select the implants for fitting into human jaw on the basis of availability of bone, bone density, mandible or maxillary section of human jaw, various types of tooth locations such as molar, premolar canine and incisors etc. means their selection criteria is based on biological aspects not on engineering design aspects. But during selection of the dental implants along with biological terms design parameters must also take into consideration because they also have direct impact on stress distribution around dental implant-bone interface. Due combine these two aspects would definitely improve the success rate of implantation. There are so many design parameters are consider while selecting implants on the basis of their impact on stresses such as length, pitch, taper angle, diameters, thread type etc. In this study length and thread pitch of dental implant were taken as design parameters and molar tooth location was selected for implantation because it was subjected to maximum failures as per the suggestions obtained by various dentists and implantologists(Figure 2).

Purpose of the study is to find out impact of Length and pitch of thread variation on stress intensity around dental implant-bone interface in cancelloues bone region.

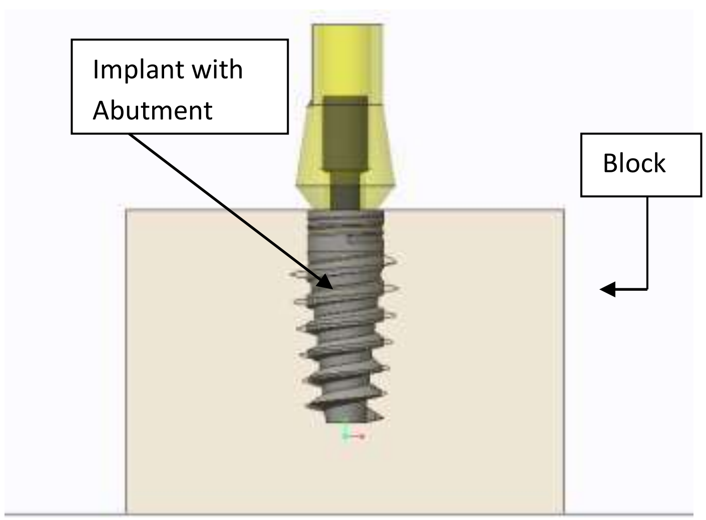


Figure 2. Implant Model Assembly

MATERIALS AND METHODS

1. **3D Model Design:**
   For study purpose ADIN TOURANG-TS Implant was selected having 10mm length and 1.15mm pitch. This implant was chosen for molar tooth location in mandibular jaw section[4]. All design specifications were studied and noted under advance profile measuring system named as video measuring machine. Actual 3d model assembly was prepared on personal computer using 3D program (pro/E Wildfire Parametric Technology Corporation, (USA) (Figure 2), [12][18].Then this main model was modified with variation in length and pitch by keeping other factor constant such as 4 models were made for length variation at constant pitch and another 4 models were made for pitch variation at constant length respectively. Model consists of implant with abutment fitted in block of size 10X 20X 20 mm size

2. **Material Properties:**

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

All materials used in the models were considered to be isotropic, homogeneous and linearly elastic in nature(Table1).The elastic properties were taken from literature[5] [12][18]. Material properties were adopted were specified in terms of young's modulus, poisons ratio .

Table 1. Linear material properties

Sr. No	Component	Material	Young's Modulus	Poisson's Ratio
01	Implant	Titanium Alloy	114000	0.33
02	Abutment	Titanium Alloy	114000	0.33
03	Screw	Titanium Alloy	114000	0.33
04	Crown	Porcelain	70000	0.38
05	Cancellous bone	Natural	1370	0.31
06	Cortical bone	Natural	13700	0.26

3. Interface Conditions:

Previous FEA studies assumed that immediate loaded implants are in direct contact with cancellous bone without blood interface and cancellous bone surrounded with cortical bone region. Bone Implant Interface Assumed to be bonded, so relative moment occurs between implant-bone interfaces [5][6].

4. Elements and Nodes:

Here models were meshed with 10 nodded tetrahedron elements for implant and and shell element for tooth. Element size kept 0.1 mm for fine meshing .A finer mesh was generated around the implant and coarse mesh was generated at remaining sections. Model assembly were composed of 7,42,679 elements and 1, 30,975 nodes in an average. Meshed Model was divided into following sections such as Cortical out/In, Cancellous Out/In, Implant, Abutment, Fitting Screw and crown (Figure 3).

5. Loading Conditions:

Present study focused only on the stresses within the cancellous bone, therefore implant system was simplified and only cancelloues section would observed for result analysis. Model were constrained in all directions at the nodes on the mesial and distal bone interface. Forces of 114.6 N and 40 N were applied on the top of the crown vertically and laterally respectively[14][20].Results were considered at combine effect of both the forces on implant under FEA analysis (Figure 4) .

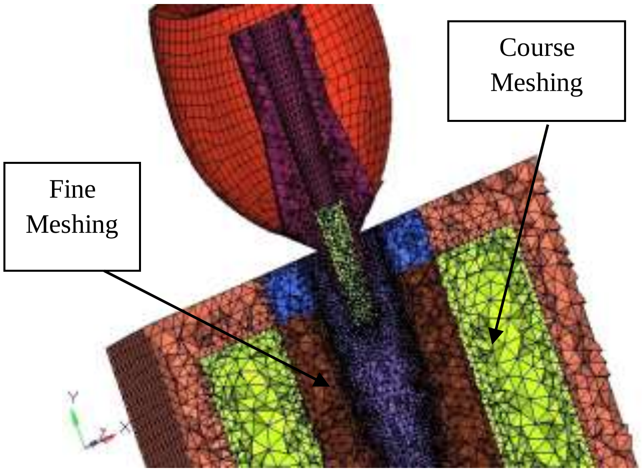


Figure 3. Meshed Model

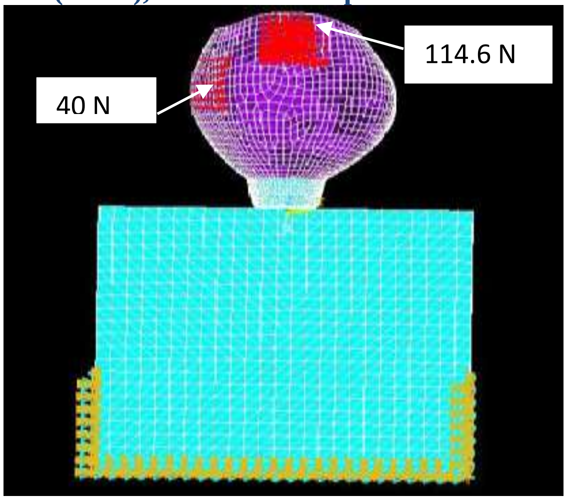


Figure 4. Loading Conditions

FEA ANALYSIS

Finite Element Analysis is a simulation Technique which evaluates the behavior of components and structures for various loading conditions including applied forces, pressures and different boundary conditions. Complex Engineering problems with non standard shapes and geometry can be solved using finite elemental analysis where closed formed solutions are not available. The finite element analysis method provides quantitative result of stress distribution, displacement and reaction loads at support etc for the subject[9][10].

Here FEA analysis was used to find stress intensity around dental implant at implant-bone interface for each model with variation in pitch and length. For original model with pitch 1.15mm and length10 mm result were observed in terms of stress intensity under combine loading conditions (Figure 5).

Table 2. FEA Load Plan

Loading Direction	Finite Element Load Magnitude
Vertical +Lateral	40N+114.6N

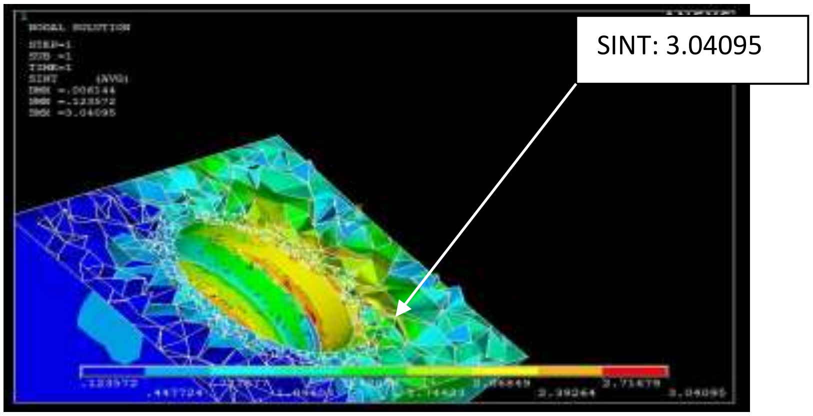


Figure 5. Stress Intensity of Original Model (Pitch1.15mm , Length10mm)

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

RESULTS AND DISCUSSIONS

For better understanding of the aim, the same FEA procedure should apply for all remaining models with variation in pitch and length for finding max stress intensity around dental implant interface [12][18]. Our original model is having length 10mm and pitch 1.15mm. So two iterations were taken on both sides of each factor as follows:

Table 3. Pitch and length Range

Factors	1	2	3	4	5
Pitch (mm)	0.85	1.0	**1.15**	1.3	1.5
Length (mm)	8	9	**10**	11	12

1. **Effect Of Pitch On Stress Intensity(SINT):**
   For this purpose, 4 models were made from original model by changing only pitch factor ranges from 0.85 mm to 1.5 mm. (Table 3) by keeping length constant Max. Stress Intensity was found as an output for each model under FEA analysis with given loading conditions (Table 4), [5]. All these results were observed at upper section of implant at cancellous bone interface (Figure 6). At 0.85 mm pitch stress intensity was found 2.7518 MPa and it increased upto 3.46519 MPa at 1.5 mm pitch.

Table 4. Effect Of Pitch

Sr. No.	**Factors**		**SINT (MPa)**
	Pitch (mm)	**Length (mm)**	
1	0.85	10	2.7518
2	1.0	10	2.88929
3	1.15	10	3.04095
4	1.3	10	3.01615
5	1.5	10	3.46519

2. **Effect Of length On Stress Intensity(SINT):**
   For this purpose, 4 models were made from original model by changing only length factor ranges from 8 mm to 12 mm. (Table 3) by keeping pitch constant[11][13]. Max. Stress Intensity was found as an output for each model under FEA analysis with given loading conditions (Table 5). All these results were observed at upper section of implant at cancellous bone interface (Figure 7). At 8 mm length stress intensity was found 3.46519 MPa and it decreased upto 2.85641 MPa at 12 mm length.

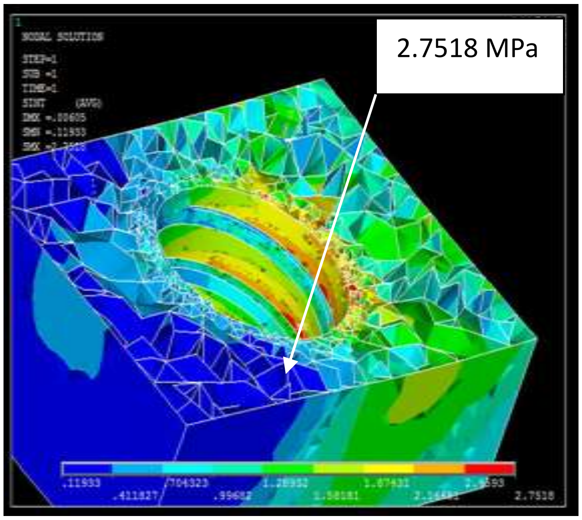
ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

Table 5. Effect Of Length

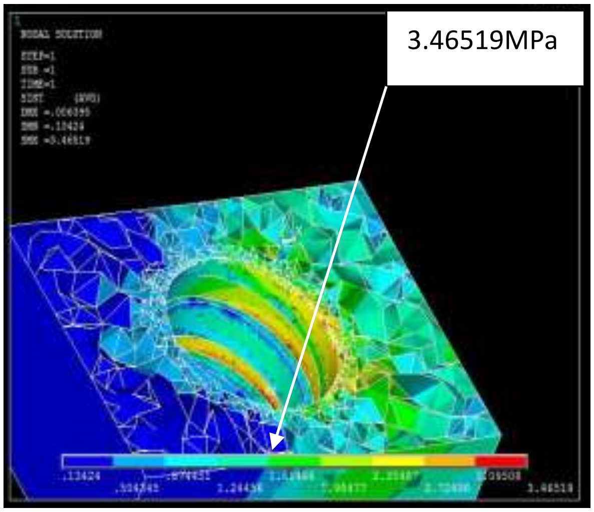
Sr. No.	Factors		SINT (MPa)
	Pitch (mm)	Length (mm)	
1	1.15	8	3.46519
2	1.15	9	3.23011
3	1.15	10	3.04095
4	1.15	11	2.96031
5	1.15	12	2.85641

3. **Analysis of Response Curve:**

Pitch and Length were taken as input variables and the max stress intensity was taken as output variable. (Table 4, 5). The response curve obtained by the analyzed data showed that as pitch increased stress intensity for each model were also increased (Figure 8). For length factor, response curve showed that stress intensity was reduced for each models with increment in length (Figure 9).



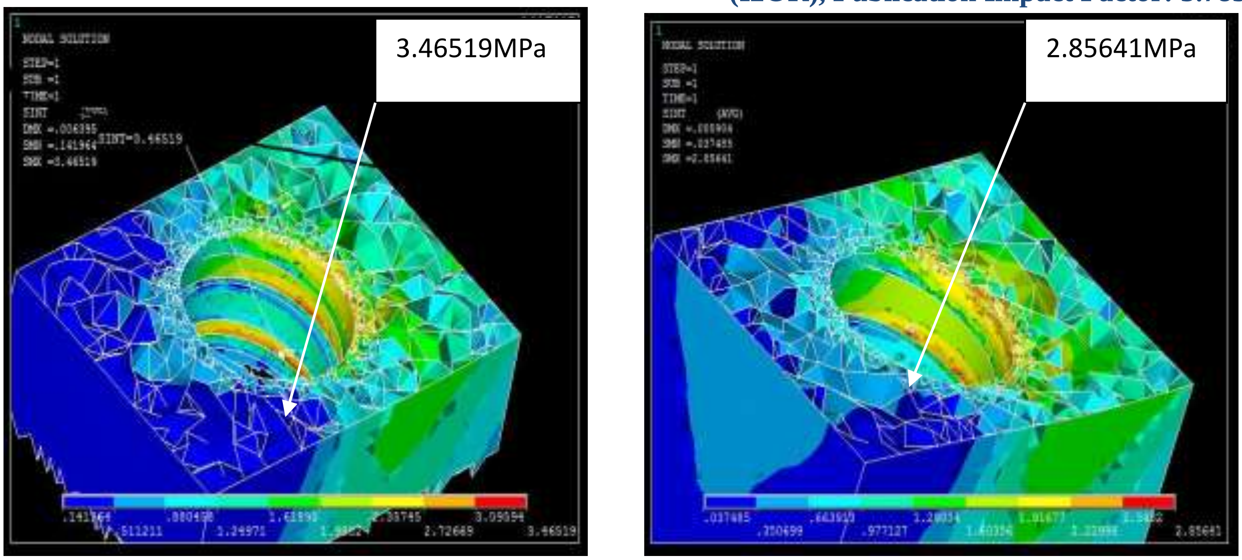
Pitch =0.85 mm



Pitch= 1.5 mm

Figure 6. FEA result observed under pitch variation

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785



Length= 8mm *Length = 12 mm*

Figure 7. FEA result observed under Length variation

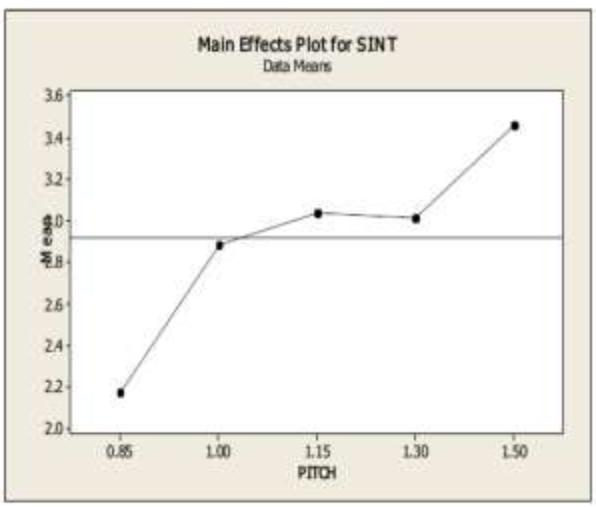


Figure 8. Pitch Effect

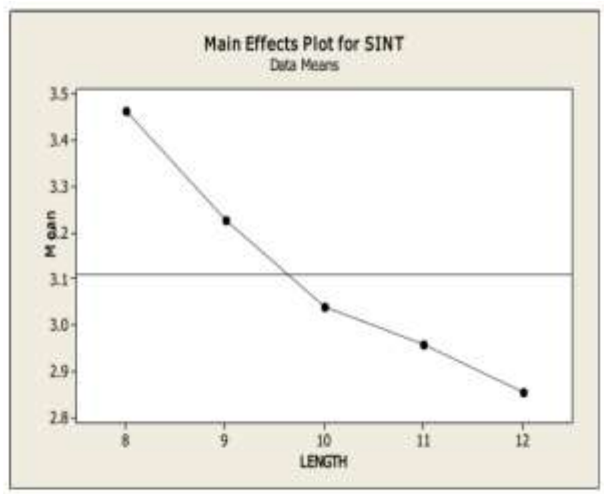


Figure 9. Length Effect

CONCLUSION

In order to achieve favorable success rate in implantation individual impact of design parameters on stress generation must understand and here we consider two parameters such as pitch of thread and length of dental implant.Within the limitations of the study, FEA simulation of dental implant we observed that,

An increase in implant length reduced magnitude of stress intensity around the dental implant-bone interface. Similarly as pitch of the dental implant increased magnitude of stress intensity also increased.

So we came to the conclusion that for reduction of stress intensity at implant bone interface the length should be maximum and pitch should be minimum for implant.

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

REFERENCES

[1] Benaissa Ali, Chikh, Meddah, 'Effects of overloading in mastication on the mechanical behavior of dental implants', Materials and Design- 2013, Vol.47; 210-217.

[2] Bijan Heidari, H. Bisadi, Behnam Heidari, Mahdi K., 'Influence of different tapered implants on stress and strain distribution in bone and implant: A Finite element analysis', Journal of Periodontology and Implant Dentistry-2009, 1(1); 11-19.

[3] Chih-Ling Chang,Chen-Sheng Chen, Chang-Hung Huang, Ming-Lun Hsu, 'Finite element analysis of the dental implant using a topology optimization Method', Medical Engineering & Physics 34 (2012) 999–1008

[4] El-Anwar, M.I., El-Zawahry, M.M. and E-Mofty, M., 'Load transfer on dental implants and surrounding bones', Australian Journal of Basic and Applied Sciences-2012, Vol.6(3); 551-560

[5] Hao-Sheng Chang , Yi-Chin Chen , Yao-Dung Hsieh , Ming-Lun Hsu, 'Stress distribution of two commercial dental implant systems: A three-dimensional finite element analysis', Journal of Dental Sciences (2013) 8, 261e271.

[6] Hong Guan, 'Influence of bone and dental implant parameters on stress distribution in the mandible: A finite element study', The International Journal of Oral and Maxillofacial Implants 2009, Vol.24 (5); 866-876.

[7] Heng-Lee Huang, 'Do Threaded Size and Surface Roughness Affect the Bone Stress and Bone-Implant Interfacial Sliding of Titanium Dental Implant? ', WCE-2011, Vol.III

[8] H. Y. Chou, Sinan Muftu, 'Simulation of peri-implant bone healing due to immediate loading of dental implant treatments', Journal of Biomechanics-2013,.2012.12.023.

[9] H. J. Chun, S. Cheong, J. Han, 'Evaluation of design parameters of osseointegrated dental implants using finite element analysis', Journal of Oral Rehabilitation–2002, Vol.29;565-574.

[10] Ibrahim, Nasser, 'Evaluation of Design Parameters of Dental Implant Shape, Diameter and Length on Stress Distribution: A Finite Element Analysis', J Indian Prosthodontic Society-2011, Vol.11(3);165-171.

[11] Jenny Zwei-Chieng Chang,Yi-Jane Chen,Yuan-Yi Tung,Yu-Ying Chiang, Eddie Hsiang-Hua Lai, Weng-Pin Chen, and Chun-Pin Lin,' Effects of thread depth, taper shape, and taper length on the mechanical properties of mini-implants', Am J Orthod Dentofacial Orthop 2012;141:279-88.

[12] Liang Kong , Yunzhuan Zhao , Kaijin Hu, Dehua Li , Hongzhi Zhou , Ziyan Wu, Baolin Liu, 'Selection of the implant thread pitch for optimal biomechanical properties: A three-dimensional finite element analysis', Advances in Engineering Software 40 (2009) 474–478.

[13] Luigi Baggi, Ilaria Cappelloni, Michele Di Girolamo, Franco Maceri ,Giuseppe Vairo,' The Influence Of Implant Diameter And Length On Stress Distribution Of Osseointegrated Implants Related ToCrestal Bone Geometry: A Three dimensional Finite Element Analysis', J Prosthet Dent 2008;100:422-431.

[14] N. Djebbar, B. Serier, B. Bachir Bouiadjra *, S. Benbarek, A. Drai, 'Analysis of the effect of load direction on the stress distribution in dental implant', Materials and Design 31 (2010) 2097–2101.

[15] O. Eraslan, 'The effect of thread design on stress distribution in a solid screw implant: a 3D finite element study', Clinical Oral Investigations-2010, Vol.14; 411-416.

[16] Samira Faegh, Sinan Muftu, 'Load transfer along the bone – dental implant interface', Journal of Biomechanics-2010,Vol.43; 1761-1770.

[17] Tae Hyun Baek, Myung Soo Kim, Dong Payo Hong, 'Fringe Analysis for photoelastic Using Image processing Technique', International Journal of Software Engineeringand Its Applications,Vol.8,No.4(2014),pp.91-102.

[18] T. Li, L. Kong, Y. Wang, K. Hu, L. Song, B. Liu, D. Li, J. Shao, Y. Ding, 'Selection of optimal dental implant diameter and length in type IV bone: a three-dimensional finite element analysis', Int. J. Oral Maxillofac. Surg. 2009; 38: 1077–1083.

[19] Yingying Sun, Liang Kong, Baolin Liu , Li Song, Shuicheng Yang, Taofeng Wei, 'Comparative study of single-thread, double-thread, and triple-thread dental implant: a three-dimensional finite element analysis', ISSN 1 746-7233, England, UK World Journal of Modelling and Simulation Vol. 3 (2007) No. 4, pp. 310-314.

[20] Zeinab Arsalanloo, Reza Telchi, and Kambiz Ghaemi Osgouie, '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, No. 3, May 2014.

AUTHOR BIBLIOGRAPHY

	**Durvesh Jitendra Thakur** I am Post Graduate student of Manufacturing system Engineering in Sarswati College Of Engineering Kharghar, Navi-Mumbai under Mumbai University, Maharashtra, India.
	**Prof.T. Z. Quazi** He is working as associate professor and HOD of Automobile engineering department in Sarswati College of Engineering, Kharghar, Navi-Mumbai.His work experience spans more than 17 years in teaching field.
	**Prof. Pankaj N. Dhatrak** He is working as an Assistant Professor in Mechanical Engineering Department of MAEER'S M.I.T. Pune. He is working on interdisciplinary projects viz. Numerical and experimental investigations of stress distribution pattern between implant and bone interface, Effect of surface roughness on fatigue life of dental implants. His work experience spans more than 10 years in teaching and 4 years in Industry. He is pursuing his PhD from Savitribai Phule Pune University, Maharashtra, India. He is a life member of The Indian Society for Technical Education (ISTE- LM 99151)