

# **Optimizing Dental Implant Model by Comparison of Three Dimensional Finite Element**

Sushant Mahajan<sup>1</sup>, Prof. Raosaheb Patil<sup>2</sup>

P.G. Student, Dept. of Mechanical Engineering, Jawaharlal Nehru Engineering College, Aurangabad, India<sup>1</sup>

Associate Professor, Dept. of Mechanical Engineering, Jawaharlal Nehru Engineering College, Aurangabad, India<sup>2</sup>

**ABSTRACT:** The application of finite element ranging from biomedical engineering. Biomechanics is fundamental to any dental implant design. Strengths at any points in the construction are critical and govern failure of the prostheses, remoulding of bone and type of tooth movement. In our study Finite Element Analysis were performed to find out the best thread shape by comparing stress induced in cortical and cancellous bone. We have taken two different thread shape implant namely Implant A: Tapered cylindrical implant with alternate thread shape of triangular and Square. 2. Implant B: Tapered cylindrical implant with alternate thread shape of triangular andTrapezoidal. To investigate effect of stress induced in bone we carried out structural static analysis of Implant, cortical and cancellous bone assembly created in 3-D modelling application. After creating 3-D model imported that model in CAE application namely ANSYS for Static structural analysis.

After comparing results of both implants we found that stresses induced in bone of Implant A is less as compared to Implant B. From this study we may conclude that it is first time we are using taper implant design to investigate the stress distribution inside the bone and it is observed that due to the tapered implant design and combination of thread shape stresses decreases in depth as where thread taper angle increases. As thread taper angle increases stress induced in bone is reduced.

**KEYWORDS:** Thread shape of implant, Dental Implant, ANSYS, Loading Conditions, stress distribution, threads design Static, Structural Analysis.

## **I. INTRODUCTION**

Biomechanics is one of the most significant aspects for the long-term stability of an osseointegrated implant, because mechanical stress by functional loading inevitably influences long-term peri-implant bone remodeling [1]. Finite element analysis (FEA) has been extensively applied to studies on stress sharing in the bone around the loaded osseointegrated implant and these studies indicate that induced stress by vertical and/or oblique loading is mostly concentrated at the crestal bone. This has led to the interpretation that such stress concentration in the bone is the most possible cause of crestal bone loss in vivo [1]. However, such an interpretation appears to be premature, because little attention has been paid to the extent to which the modelling method significantly influences the results.

Thread shape plays vital role in biomechanical optimization of dental implants. Threads are used to maximize or minimize contact between dental implant and bone, that's why it is very important to improve initial stability, enlarge implant surface area and dissipation of interfacial stress. It is essential to evaluate the thread design of dental implant to boost further clinical success. In fact, the FEA models used in these studies help to conclude the optimum geometry solution for dental implant.

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 11, November 2015

## II. RELATED WORK

1. Y. Akagawa, Y. Sato, E. R. Teixeira, N. Shindoi & M. Wadamoto ; " A mimic osseointegrated implant model for Three-dimensional finite element analysis " [1]

The purpose of this study was to develop a new three-dimensional (3D) mimic model of an osseointegrated implant for finite element analysis (FEA) and to evaluate stress distributions in comparison with a model commonly used in most studies as a control. The findings, that stress distribution at the peri-implant bone were quite different between the mimic and control models, suggest the need to carefully interpret stress distribution in previous studies with models of uniform cancellous peri-implant bone.

2. Ming-Lun Hsu and Chih-Ling Chang; "Application of finite element analysis in dentistry"[3]

Since Brånemark introduced the concept of osseointegration and the possibility of anchoring dental prostheses by intraosseous implantation in 1969, the clinical use of implants for oral and maxillofacial rehabilitation has rapidly expanded over the past 20 years. Biomechanical factors play a substantial role in implant success or failure. The application of occlusal forces induces stresses and strains within the implant-prosthesis complex and affects the bone remodeling process around implants. To achieve optimized biomechanical conditions for implant-supported prostheses, conscientious consideration of the biomechanical factors that influence prosthesis success is essential.

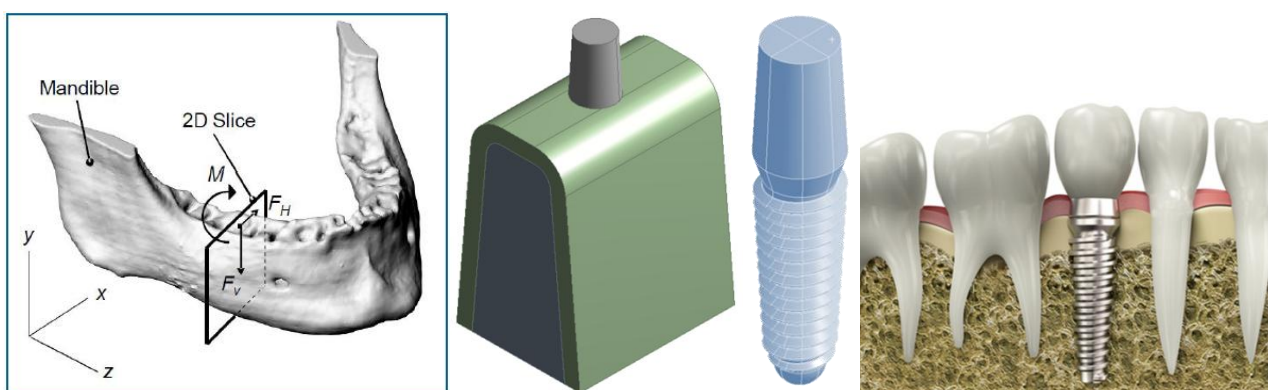
3. Yingying Sun, Liang Kong, Baolin Liu , Li Song, Shuicheng Yang, Taofeng Wei, " Comparative study of single-thread, double-thread, and triple-thread dental implant: a three-dimensional finite element analysis"[5]

The purpose of this study was to determine the optimal thread design for an experimental cylinder implant. Three-dimensional finite element models with an implant of single-thread, double-thread, and triple-thread were created. The results imply that single-thread implant shows the best stress transmission under axial load. Single-thread and triple-thread implant show the better stress transmission under buccolingual load. Different thread designs show similar effect on the osseointegrated implant stability.

## III. STUDY CRITERIA

Our main aim is to study the bone stress by varying implant thread design. So for easier comparison we keep simplify geometry on inner and outer bone. Create model of complete mandible is relatively difficult so we use particular segment of mandible which is considerably easier. For our study we take cut section from mandible as shown below.

Fig.1. Cut section from mandible



We select a segment from mandible bone. Bone geometry was simplified and simulated as rectangle, or brick consist of two layers of bone. The inner bone represents the spongy bone (size= 15 X 20 X 15 mm) which fills inner space of

**International Journal of Innovative Research in Science,  
Engineering and Technology**

**(An ISO 3297: 2007 Certified Organization)**

**Vol. 4, Issue 11, November 2015**

outer bone of thicknesses of 2mm which represents a cortical bone. As shown in diagram of Bone geometry with dental implant.

#### IV. IMPLANT GEOMETRY (3-D MODEL)

To construct a three-dimensional (3-D) finite element model for the study is an important operation in FEA. It is usually recommended that, when comparing the qualitative results of one case with respect to another, a 2-D model is effective. Although the time required to create finite element models is reduced with advanced computer technology, there is still an acceptable time and cost savings when using a 2-D model over 3-D, when appropriated. Still, 2-D models cannot simulate the 3-D complexity within structures and result is of little experimental standards. The group of 3-D regional FE models is systematic and well proven practice in mandible related researches. This is because modelling only the particular segment of mandible is much easier than modelling the whole mandible. In many of these regional models, reproduced boundary conditions are frequently over simplified and yield too much significance to their predictive, quantitative result.

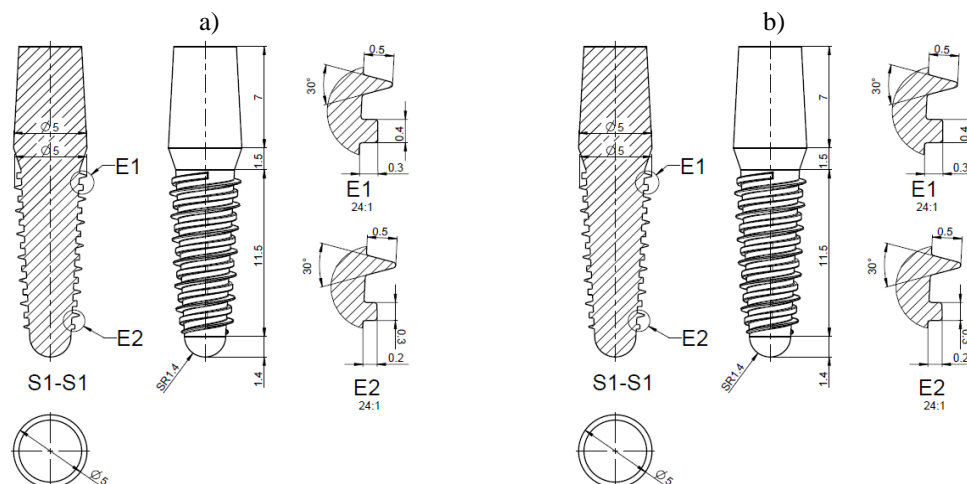
In 3-D analysis, the stress, strain and deformation condition can be calculated in all three directions (x, y, and z). The 1<sup>st</sup> step in FEA modelling is to create the geometry of interest in the computer with the help of CAD application. A mandibular segment with an dental implant and a superstructure was modelled with the help of CAD tool Creo 2.0 (Parametric Technology Corporation, USA). and CATIA [Computer Aided Three dimensional Interactive Application] V5 R20 ( Dassault System, France). After creation of complete 3-D model assembly of 1. Implant, 2. Cortical bone, 3. Cancellous bone that model geometry is transformed into STEP (Standard for the Exchange of Product model data) file format which is neutral CAD file format which can be easily used in FEA environment. For Finite Element Analysis (FEA) we used ANSYS program. With the help of ANSYS we have done meshing and structural analysis of 3-D model of implant to get required results.

For this study we have considered two different thread shape implants the details of this two thread implants are as below

- Implant A:** Tapered cylindrical implant with alternate thread shape of triangular and Square.
- Implant B:** Tapered cylindrical implant with alternate thread shape of triangular and Trapezoidal.

The detail geometry of both implants is as shown below.

Fig.2. Detail drawing of a) Implant A b) Implant B



# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 11, November 2015

## V. ACCURACY OF FINITE ELEMENT MODELS (FEM)

Accuracy of FEA is determined by comparing its result to the experimental tests it is quite difficult to made exact set up for experimental test as taken in FEA model. In FEA we consider as all materials are homogeneous and 100% defect free. But for experimental test the mandible bone may have defects. In experimental test we uses strain gages and then calculate the stresses in the bone and implant. Accuracy of FEA is depends in the person that how much tolerance he allowed with comparing experimental result. For our study we will allow 20% of results varying from experimental and simulation values. FEA is very good tool to predict the behaviour of one model with relative to another model. In FEA there is good flexibility of changing geometry, material properties and loading conditions.

Different CAE software's are used like STRAND 7, Nastran (MSC software partners solutions Marburg, Germany), Patran (MSC software corporation, USA), ANSYS. In our study we have used ANSYS.

## VI. MESHING OF FEA MODEL

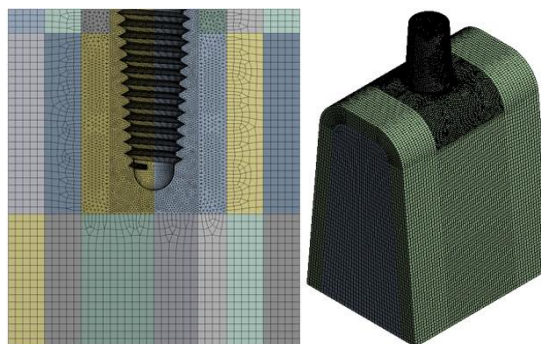
Mesh generation is the exercise of generating a polygonal or polyhedral mesh that approximates a geometric domain. The term "grid generation" is frequently used interchangeably. Typical uses are for execution to a computer screen or for physical simulation such as finite element analysis or computational fluid dynamics. The input model form can vary greatly but common sources are CAD, NURBS, B-rep, STL (file format) or apoint cloud. The field is extremely interdisciplinary, with contributions found in mathematics, computer science, and engineering.

Three-dimensional meshes formed for finite element analysis must to consist of tetrahedra, pyramids, prisms or hexahedra. Those used for the finite volume method can consist of random polyhedra. Those used for finite difference methods typically need to consist of piecewise structured arrays of hexahedra known as multi-block structured meshes. A mesh is otherwise a discretization of a area existing in one, two or three dimensions.

## VII. BEST MESH SIZE AND FINENESS FOR A MESH REFINEMENT (CONVERGENCE) STUDY

The Finite Element Method (FEM) to compute single- and multiphysics simulations. Whenever we use the finite element method, it is important to remember that the accuracy of our solution is linked to the mesh size. As mesh size decreases towards zero (leading to a model of infinite size), we move toward the exact solution for the equations we are solving. However, since we are limited by finite computational resources and time, we will have to rely on an approximation of the real solution. The goal of simulation, therefore, is to minimize the difference ("error") between the exact and the approximated solution, and to ensure that the inaccuracy is below some accepted tolerance level that will vary from project to project based on our design and analysis goals.

Fig.3. Mesh Model of Implant Assembly



# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 11, November 2015

## VIII. BOUNDARY CONDITIONS

Currently FEA programs provide several types of contact algorithms for simulation of contacts. For our study we have used frictional contact - to obtain initial stability for the situation of immediate loading after implantation, it was modelled using nonlinear frictional contact elements, which allowed minor displacements between implant and bone. Under these conditions, the contact zone transfers pressure and tangential forces (i.e., friction), but no tension. The friction coefficient was set to 0.3. The friction between contact surfaces can also be modelled with contact algorithms. The coefficient of friction (COF), often symbolized by the Greek letter  $\mu$ , is a dimensionless scalar value which describes the ratio of the force of friction between two bodies and the force pressing them together. The coefficient of friction depends on the materials used; for example, ice on steel has a low coefficient of friction, while rubber on pavement has a high coefficient of friction. Coefficients of friction range from near zero to greater than one.

Potential energy and the solutions can be delivered by applying boundary conditions of FEA models. Boundary conditions means constraints applied on CAD models. In FEA we can easily change in force, magnitude and directions. We can consider infinite changes but to limit our study we go only below boundary conditions.

1. Base of FEA model is fixed
2. Sides of FEA model is frictionless supports
3. Loading was applied on the top of abutment on horizontal surface of implant assembly

Model were constraint in all directions on the mesial and distal bones since this study was aimed to investigating bone effects to loads within the physiological limits rather than to overloads.

There are four different loading conditions are used details of same are as shown below

1. Axial downward (Compressive-100 N) <sup>[6]</sup>
2. Angular force of 100N from buccal (cheek) to lingual (tongue) <sup>[6]</sup> side

Fig.4. Vertical loading conditions applied on Implant assembly

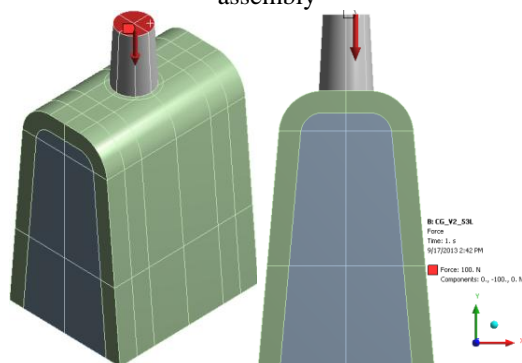
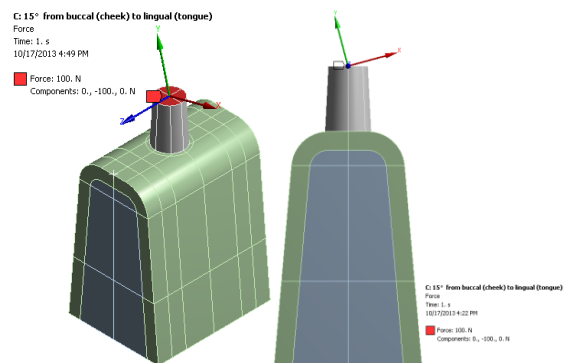


Fig.5. Angular forces applied on Implant assembly



3. Axial upward (Tension- 50N).

Fig.6. Axial upward load applied on Implant assembly

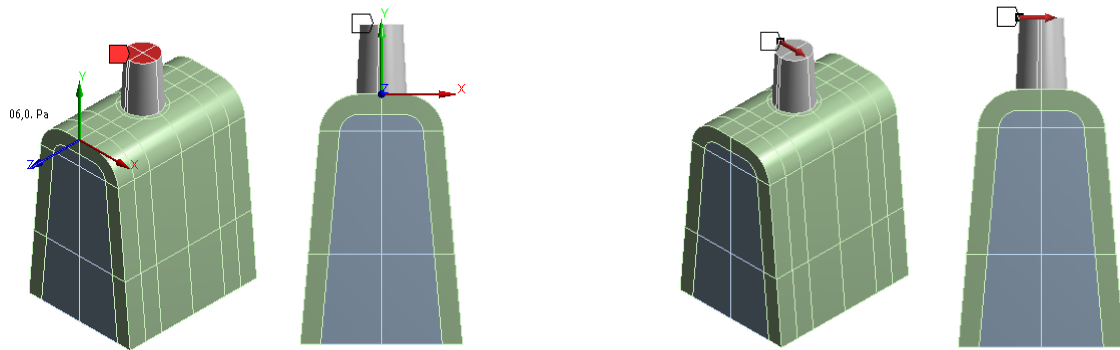
4. Bending (20 N cheek to tongue).

Fig.7. Bending load applied on Implant assembly

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 11, November 2015



Analysis was performed for each loading conditions separately with the help of ANSYS 13.0 Workbench software program. Von-mises stresses (Equivalent stresses) distribution and Total Deformation were used to display the stress in the cortical and cancellous bone.

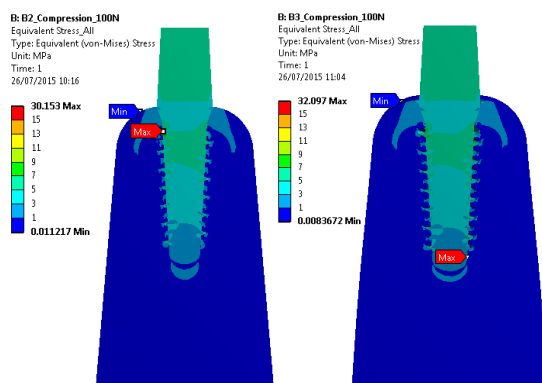
## IX. STATISTICAL ANALYSIS

After applying all boundary condition, all material properties & Loading conditions simulation is carried out. After completing simulation with the help of ANSYS 13.0. We have extracted results of FEA analysis and they are as follows

### 1. Compression of 100N

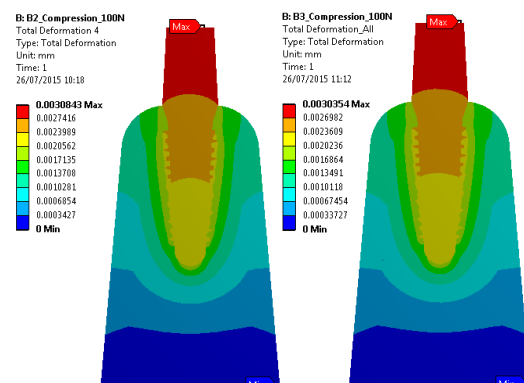
#### a. Equivalent Stress [Mpa]

Fig.8. Equivalent (Von-Mises) Stress of Assembly



#### b. Total Deformation [mm]

Fig.9. Total deformation of Assembly



### 2. Compression of 100N at 15° from cheek to tongue

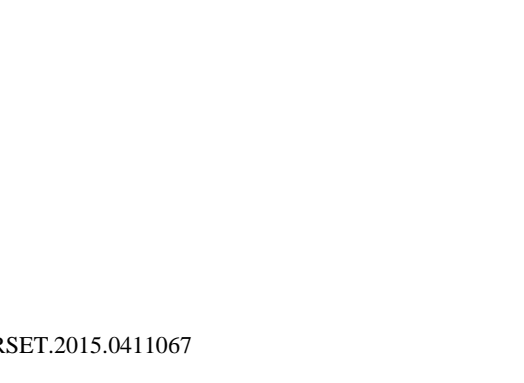
#### a. Equivalent Stress [Mpa]

Fig. 10. Equivalent (Von-Mises) Stress of Assembly



#### b. Total Deformation [mm]

Fig.11. Total deformation of Assembly

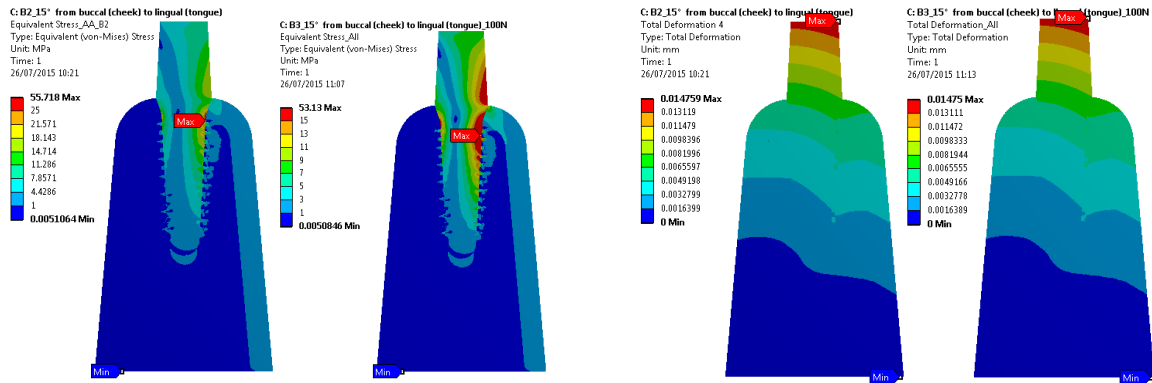




# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

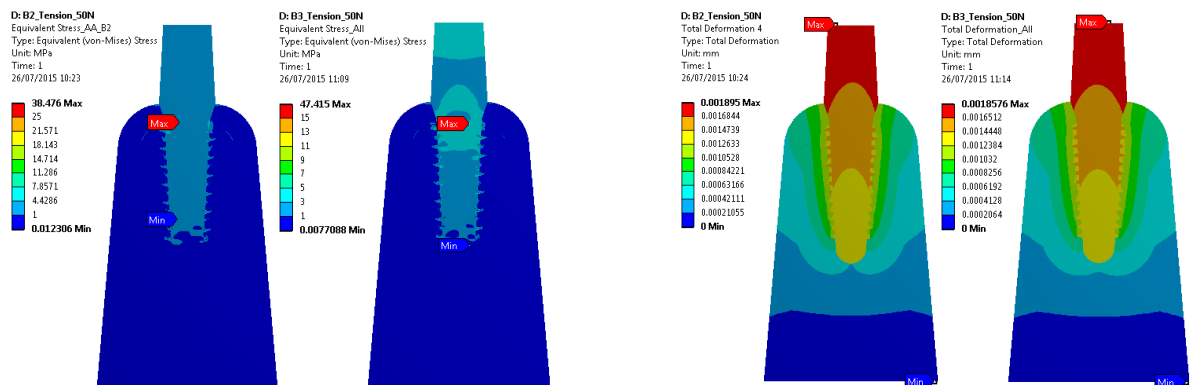
Vol. 4, Issue 11, November 2015



## 3. Tension of 50N

a. Equivalent Stress [Mpa]  
Fig.12. Equivalent (Von-Mises) Stress of Assembly

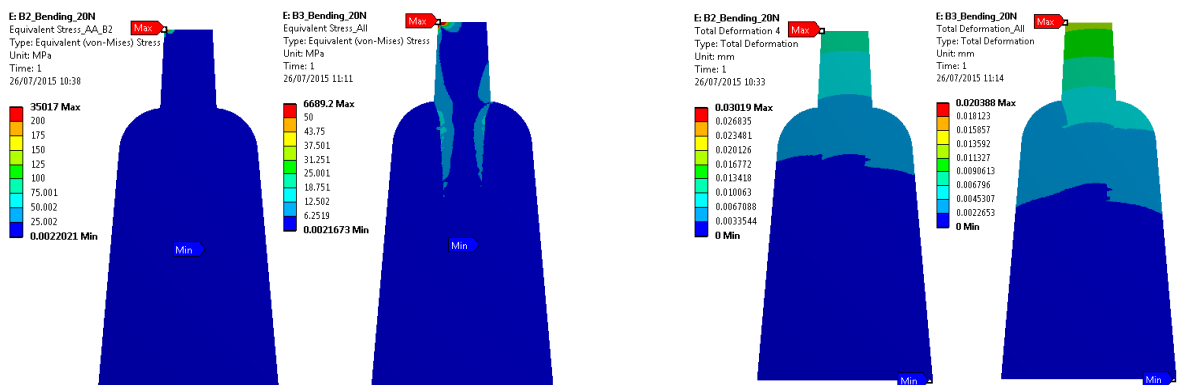
b. Total Deformation [mm]  
Fig.13. Total deformation of Assembly



## 4. Bending of 20N

a. Equivalent Stress [Mpa]  
Fig. 14. Equivalent (Von-Mises) Stress of Assembly

b. Total Deformation [mm]  
Fig.15. Total deformation of Assembly



# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 11, November 2015

Table 1.Comparison of Equivalent (Von-Mises) Stress

Implant	Compression 100N	15 Degree 100 N	Tension 50 N	Bending 20 N
Implant A	30.153	55.718	38.476	35017
Implant B	32.097	53.13	47.415	6689.2

Table 2. Comparison of Total deformation

Implant	Compression 100N	15 Degree 100 N	Tension 50 N	Bending 20 N
Implant A	0.003	0.0147	0.0018	0.03019
Implant B	0.003	0.0147	0.0018	0.020388

## X. VALIDATION

As tabulate above the results extracted from FEA we try to summarize data set in visual form to clarify trends better than do tables. As we know bar graph show each data category in a frequency distribution it is very easy to validate data

Fig. 16. Graphical comparison of Equivalent tresses

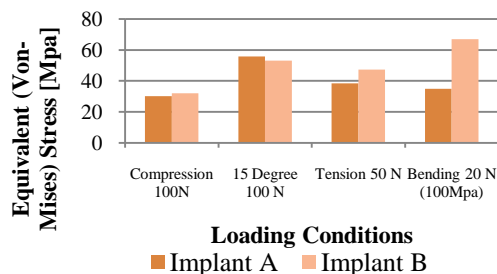
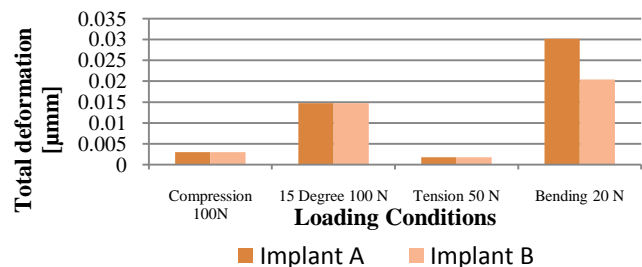


Fig.17. Graphical comparison of Total deformation in  $\mu\text{mm}$ .



After comparing the result in terms of stress and deformation we found that **Implant A** is optimum than Implant B.

## XI. CONCLUSION

The aim of this study was to find the effect of stress distribution and deformation in implant assembly upon the variations of the thread shapes. For this reason it was assumed that all the parameters of the models were identical except the Implant thread shape. This makes it possible to make an evaluation among threads of different shape. After comparing results of both implants we found that **implant A**: Tapered cylindrical implant with alternate thread shape of triangular and Trapezoidal found optimum.

It has been reported that even loads below the ultimate bone stress can cause bone failure, as in the case of fatigue failures, in which the micro damage of bone can no longer be repaired. The accumulated micro damage might result in bone restoration.

## REFERENCES

- [1] Y. Akagawa, Y. Sato, E. R. Teixeira, N. Shindoi and M. Wadamoto, "A mimic osseointegrated implant model for three-dimensional finite element analysis", J Oral Rehabil. 2003 Jan; 30(1):41-5. Vol. 28 No. 2, 324-330.
- [2] Gefen A., "Optimizing the biomechanical compatibility of orthopedic screws for bone fracture fixation", Med Eng Phys. 2002 Jun; 24(5):337-47.
- [3] Ming-Lun Hsu and Chih-Ling Chang, "Application of finite element analysis in dentistry", J Prosthet Dent. 2001 Jun; 85(6):585-98.



## International Journal of Innovative Research in Science, Engineering and Technology

*(An ISO 3297: 2007 Certified Organization)*

**Vol. 4, Issue 11, November 2015**

- [4] Heng-Li Huang, Jui-Ting Hsu, Lih-Jyh Fuh, Ming-Gen Tu, Ching-Chang Ko, Wen-Shen; "Bone stress and interfacial sliding analysis of implant designs on an immediately loaded maxillary implant: A non-linear finite element study", J Dent. 2008 Jun; 36(6):409-17. doi: 10.1016/j.jdent.2008.02.015. Epub 2008 Apr 1.
- [5] 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" ISSN1 746-7233, England, UKWorld Journal of Modelling and Simulation Vol. 3 (2007) No. 4, pp.310 – 314.
- [6] M Karl, W Winter, AJ Dickinson, MG Wichmann, SM Heckmann; "Different bone loading patterns due to fixation of three-unit and five-unit implant prostheses", Aust Dent J. 2007 Mar;52(1):47-54.
- [7] Liang Kong, Baolin Liu, Dehua Li, Yingliang Song, Aijun Zhang, Faning Dang, Xinqiang Qin, Jin Yang; "Comparative study of 12 thread shapes of dental implant designs: a three-dimensional finite element analysis", ISSN 1 746-7233, England, UK World Journal of Modelling and Simulation Vol. 2 (2006) No. 2, pp. 134-140.
- [8] Osama Abu-Hammad, Ameen Khraisat, Najla Dar-Odeh, Mohammed El-Maaytah; "Effect of Dental Implant Cross-Sectional Design on Cortical Bone Structure Using Finite Element Analysis", Clin Implant Dent Relat Res. 2007 Dec;9(4):217-21.