International Journal of Engineering Science Invention
ISSN (Online): 2319 – 6734, ISSN (Print): 2319 – 6726
www.ijesi.org ||Volume 4 Issue 12|| December 2015 || PP.26-33

Enhancing dental implant model by evaluation of three dimensional finite element analysis.

Nilesh Parkhe[1], Umesh Hambire [2], Dr. Chaitali Hambire [3],
Dr. Siddharth Gosavi[4]

[1,2] Department of Mechanical Engineering, Government College of Engineering, Aurangabad,M.H., India
[3]Department of Pediatric Dentistry, SMBT Dental College, Sangamner, Ahmednagar, M.H., India
[4] Professor, Department of Prosthodontics, Krishna dental College, Libyan arab Jamahiriya, karad, M.H., India

ABSTRACT*: The FEM is numerical approximation to solve partial differential equation (PDE) and integral equations that are formulated to describe physics of complex structure, permitting the numerical analysis of complex structures based on their material properties. The application of finite element ranging from biomedical engineering. Biomechanics is fundamental to any dental implant design. Following functional load stress and strains are created inside the biological structures. 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 1. Implant- A: Tapered cylindrical implant with alternate thread angle [30°, 60°, 30°, and 60°] & 2. Implant- B: Tapered cylindrical implant with alternate thread angle & height [30° & 0.5mm, 60°& 0.3mm, 30°& 0.5mm, 60°& 0.3mm]. 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- *ANSYS, dental implant, load, Static Structural Analysis, Thread shape of implant.*

I. INTRODUCTION

Thread shape & geometry is an important intention in biomechanical optimization of dental implants. Threads are used to maximize preliminary contact, improve initial steadiness, enlarge implant outside area and favor dissipation of interfacial stress. It is required to evaluate the thread design of dental implant to improve further clinical success.

Many different methods have been used to study the stress/strains in bone and dental implants. Photoelasticity provides high qualitative information pertaining to the overall position of stresses but only partial quantitative information. Strain-gauge measurements give accurate data concerning strains only at the specific location of the gauge. Finite element analysis (FEA) is able to provide complete quantitative data at any location within mathematical model. Thus FEA has become an important analytical tool in the evaluation of implant systems in dentistry.

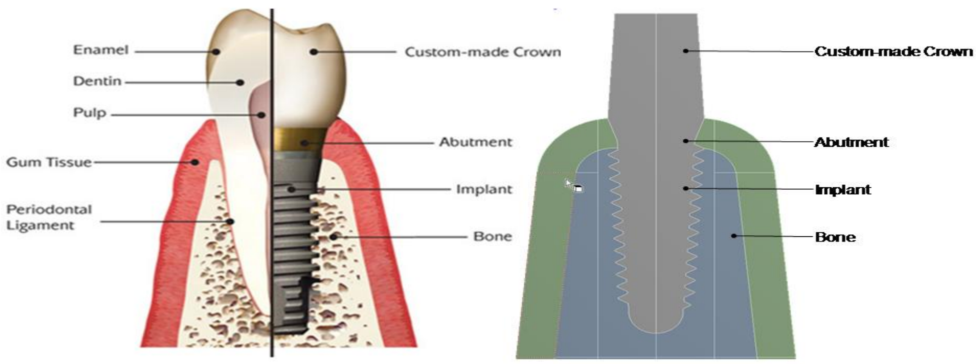

Fig.: Similarities of real Model with CAD Model of Dental Implant

II. ACCURACY OF FINITE ELEMENT MODELS

Accuracy of FEA is determined by comparing its result to the experimental tests it is quite difficult to made exact set up of 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 & 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 & 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 & loading conditions.

Differant CAE softwares 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,

III. ANSYS ADVANTAGES

1. Unequalled Depth

The ANSYS commitment is to provide unequalled technical depth in any simulation domain. Whether it's structural analysis, fluids, thermal, electromagnetic, meshing, or process & data management we have the level of functionality appropriate for your requirements. Through both significant R&D investment and key acquisitions, the richness of our technical offering has flourished. We offer consistent technology solutions, scalable from the casual user to the experienced analyst, and seamless in their connectivity. In addition, we have world class expertise for all of these domains, available to help you implement your ANSYS technology successfully.

2. Unparalleled Breadth

Unlike other engineering simulation companies, who may possess competence in one, or maybe two, fields, ANSYS can provide this richness of functionality across a broad range of disciplines, whether it be explicit, structural, fluids, thermal, or elecromagnetics. All of these domains are supported by a complete set of analysis types and wrapped by a unified set of meshing tools. Together, these domains form the cornerstones of the ANSYS portfolio for Simulation Driven Product Development, and constitute a complete portfolio of unparalleled breadth in the industry.

3. Comprehensive Multiphysics

A strong foundation for multiphysics sets ANSYS apart from other engineering simulation companies. Our technical depth and breadth, in conjunction with the scalability of our product portfolio, allows us to truly couple multiple physics in a single simulation. Technical depth in all fields is essential to understand the complex interactions of different physics. The portfolio breadth eliminates the need for clunky interfaces between disparate applications. The ANSYS capability in multiphysics is unique in the industry; flexible, robust and architected in ANSYS Workbench to enable you to solve the most complex coupled physics analyses in a unified environment.

4. Engineered Scalability

Scalability is a critical consideration when considering software for both current and long term objectives. At ANSYS engineered scalability means flexibility you need has been designed for your particular needs. ANSYS provides you with the ability to apply the technology at a level that is appropriate for the size of the problem, execute it on a full range of computing resources, based on what's appropriate and available, and finally the ability to deploy the technology within your company's user community. The result is efficient usage and optimum return on your investment, whether you have a single user or an enterprise-wide commitment to Simulation Driven Product Development. As your requirements grow and the level of sophistication and maturity evolves, the technology from ANSYS also will scale up accordingly.

5. Adaptive Architecture

Adaptive software architectures are mandatory for today's world of engineering design and development where a multiplicity of different CAD, PLM, in-house codes and other point solutions typically comprise the overall design and development process. A software environment is needed which anticipates these needs and gives you the tools and system services for customization as well as interoperability with other players. Such adaptability is a mandatory requirement and characteristic of the ANSYS simulation architecture, enabling your organization to apply the software in a manner which fits with your philosophy, environment and processes. ANSYS Workbench can be the backbone of your simulation strategy, or peer-to-peer with other software environments, or ANSYS technology can be a plug-in to your CAE supplier of choice. The ANSYS commitment to Simulation Driven Product Development is the same in any case.

6. Coefficient friction

It is dimensionless scaler quantity & depends on martial used

$$\mu = \frac{\text{Friction force between two bodies}}{\text{force pressing them together}}$$

1. µ 0.3- Smooth metal surface and bone.
2. µ 0.45 - Rough metal surface and bone.
3. µ 1 - Excessive rough metal surface and bone.

For analysis we used frictional contacts between surface to surface. Frictional contact is nonlinearbehavior for this contact we used contact 174 & target 170 elements in ANSYS. Model was modelled by using nonlinear frictional contact because to get initial stability for the immediate loading this contact also allowed minor displacement between implant & bone.

IV. STUDY CRITERIA

Our main aim is to study the bone stress by varying implant design. So for easier comparison we keep simplify geometry on inner and outer bone. Modelling a complete mandible is quite difficult so we use selected segment of mandible which is much easier. For our study we take cut section from mandible as shown below.

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 outer bone of thicknesses of 2mm which represents a cortical bone.

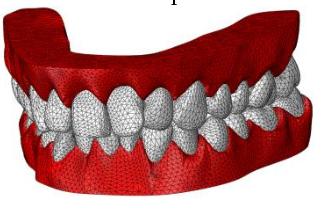

Fig.: FEA Mesh model of teeth & mandible

V. 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 error is below some accepted tolerance level that will vary from project to project based on our design and analysis goals.

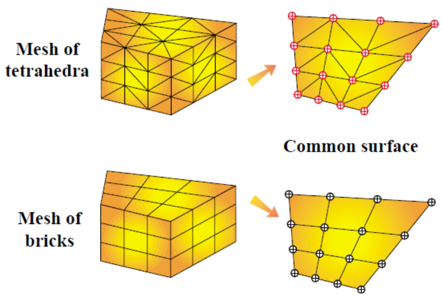

Fig.: Example of a 3-D non-matching mesh. Top portion discretized with tetrahedral, lower portion with brick.

We will need to track a characteristic output parameter from our simulation as we vary the mesh size and determine at which mesh size the parameter has "converged" on the correct value. Note that "converged" is used in quotation marks because the convergence criteria will depend on our design and analysis goals.

In general, convergence is a coming together of two or more distinct entities or phenomena. Convergence is increasingly prevalent in the CAE world; in this context the term refers to the combination of two or more different simulations in single loading conditions by varying element size.

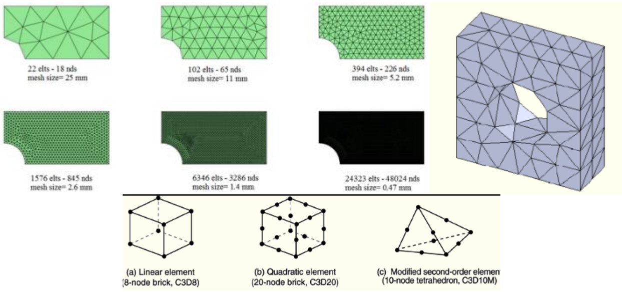

Fig.: similar object with different mesh size

VI. BOUNDARY CONDITIONS

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 & directions. We can consider infinite changes but to limit our study we go only below boundary conditions.

1. Base of FEA model is fixed

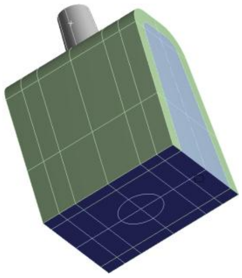
Fig: Boundary conditions applied on Implant assembly

2. Sides of FEA model is frictionless supports

Implant and the bones are connected to each other by Frictional contacts with coefficient of friction 0.3 as shown below

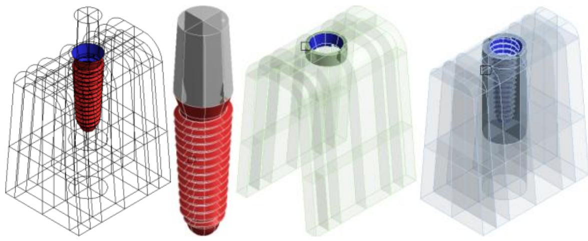
Fig: Frictional contacts created in Implant assembly

Then inner bone & outer bone in connected by each other by bonded contacts as shown below. Contacted surface is shown in red & blue colour.

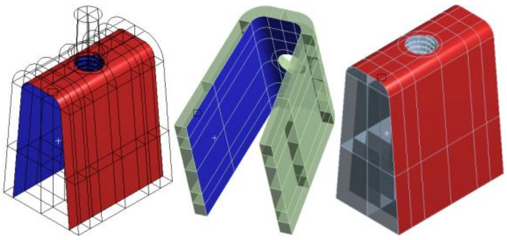
Fig: Bonded contacts created in Implant assembly

3. Loading was applied on the top of abatement on horizontal surface of implant assembly

Model were constraint in all directions on the mesial & 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

1.) Total assembly is fixed at bottom and Axial downward (Compresive-100 N)[5] is applied on implant as shown in below figure

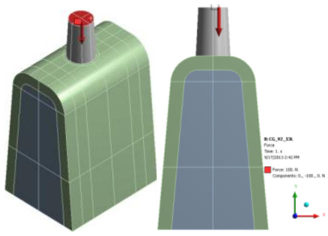

Fig: Vertical loading conditions applied on Implant assembly

2.) Angular force of 100N from buccal (cheek) to lingual (tongue) [5] side as shown below

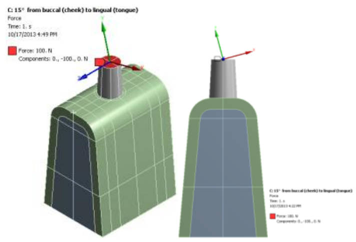

Fig: Angular forces applied on Implant assembly

3. Axial upward (Tension- 50N).

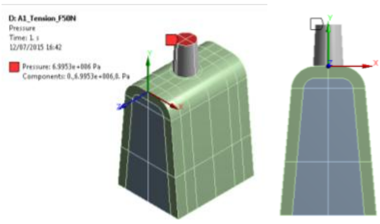

Fig: Axial upward load applied on Implant assembly

4. Bending (20 N cheek to tongue).

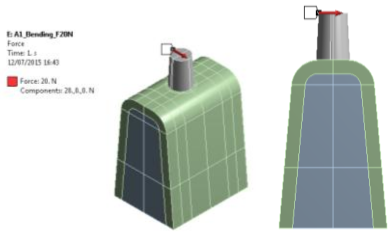

Fig: Bending load applied on Implant assembly

VII. RESULT

After applying all boundary condition & material properties FEA model is solved with the help of ANSYS 13.0. We get the results of FEA analysis after completion of solving procedure in ANSYS are as follows

1. Compression of 100N

   a. Equivalent Stress [Mpa]

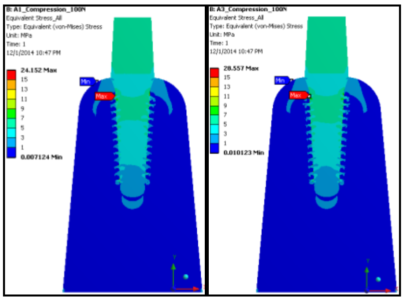

Fig.: Equivalent (Von-Mises) Stress of Assembly

b. Total Deformation [mm]

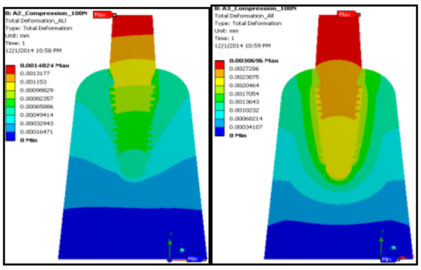

Fig: Total deformation of Assembly

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

a. Equivalent Stress [Mpa]

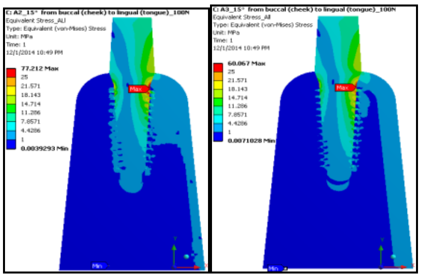

Fig.: Equivalent (Von-Mises) Stress

b. Total Deformation [mm]

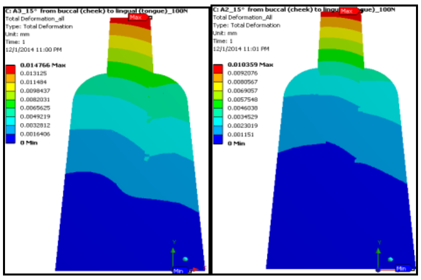

Fig: Total deformation of Assembly

3. Tension of 50N

a. Equivalent Stress [Mpa]

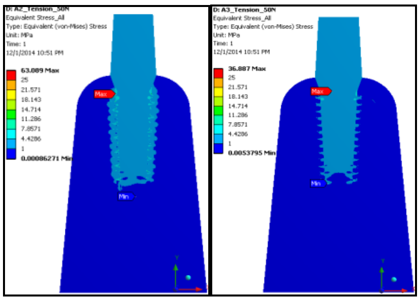

Fig.: Equivalent (Von-Mises) Stress

b. Total Deformation [mm]

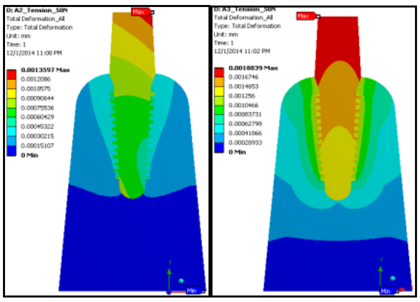

Fig: Total deformation of Assembly

4. Bending of 20N

a. Equivalent Stress [Mpa]

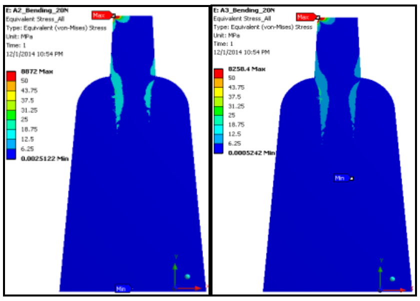

Fig.: Equivalent (Von-Mises) Stress

b. Total Deformation [mm]

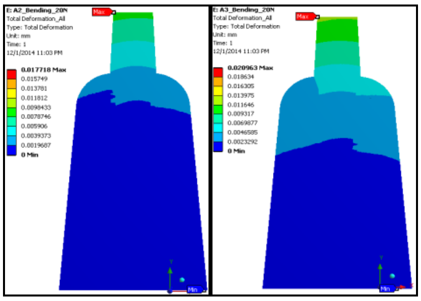

Fig: Total deformation of Assembly

Implant	Compression 100N	15 Degree 100 N	Tension 50 N	Bending 20 N
Implant A	24.152	77.212	63.089	8872
Implant B	28.557	60.067	36.887	8258.4

Table: comparison of Equivalent (Von-Mises) Stress

Implant	Compression 100N	15 Degree 100 N	Tension 50 N	Bending 20 N
Implant A	0.0014	0.01035	0.0013	0.0177
Implant B	0.003	0.01476	0.0018	0.0209

Table: Comparison of Total deformation

VIII. VALIDATION

After getting the FEA result, we represented all results in graphical form as shown below

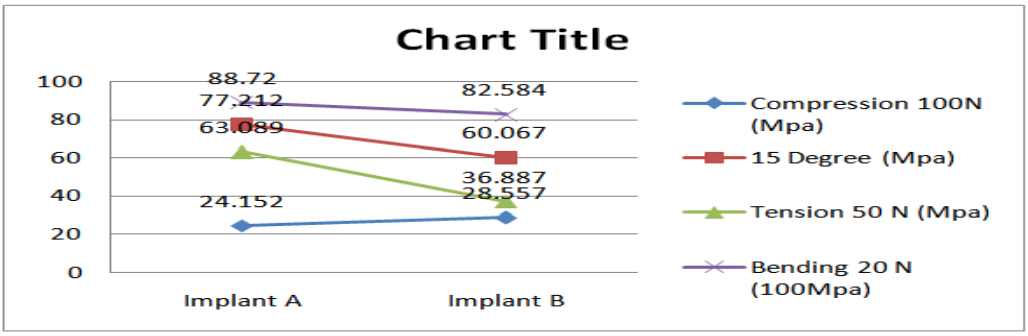

Fig: Graphical comparison of Equivalent tresses

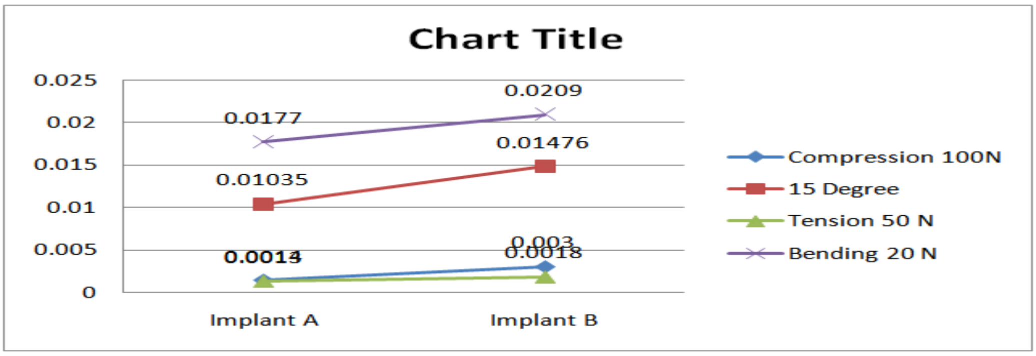

Fig: Graphical comparison of Total deformation in µmm.

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

IX. CONCLUSION

The aim of this study was to find the pure effect upon the variations of the thread shapes. For this reason it was assumed that all the parameters of the models were identical except the thread shape. This makes it possible to make a comparison between threads of different shape. After comparing results of both implant we found that Implant X: Tapered cylindrical implant with constant thread angle [60°] 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 resorption.

X. ACKNOWLEDGMENT

I would like to take this opportunity to convey my sincere appreciation for the the paper on the topic “Application of finite element analysis to investigate the effect of thread design on stress distribution in mandible regulated by implant parameters in dentistry” which is direct result of suggestion & direction from Prof. R.B. Patil, Department of Mechanical Engineering .It is my proud privilege to express my sincere thanks for encouragement and motivation provided by Prof. Dr. M. S. Kadam, (HOD Mechanical Department), Dr. S. D. Deshmukh (Principal, JNEC) and Dr. Sidharth Gosavi (Krishna dental school.

My sincere thanks to all my friends who have helped me directly or indirectly in the course of successful completion of this paper.

REFERENCES

[1] Y. Akagawa, Y. Sato*, E. R. Teixeira, N. Shindoi & M. Wadamoto ; "A mimic osseointegrated implant model for Three-dimensional finite element analysis"
[2] Gefen; "Optimizing the biomechanical compatibility of orthopedic screws for bone fracture fixation"
[3] Ming-Lun Hsu and Chih-Ling Chang; " Application of finite element analysis in dentistry"
[4] Heng-Li Huang, Jui-Ting Hsu, Lih-Jyh Fuh, Ming-Gene Tu, Ching-Chang Ko , en-Wen Shen; "Bone stress and interfacial sliding analysis of implant designs on an immediately loaded maxillary implant: A non-linear finite element study"
[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"
[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"
[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"
[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"
[9] H.-J. Chun, S.-Y. Cheong, J.-H. Han, S.-J. Heo, J.-P. Chung, I.-C. Rhyu,Y.-C. Choi, H.-K. Baik, Y. Ku & M.-H. Kim; "Evaluation of design parameters of osseointegrated dental implants using finite element analysis"
[10] José Henrique Rubo, Edson Antonio Capello Souza; "Finite-Element Analysis of Stress on Dental Implant Prosthesis"
[11] Kıvanç Akça, Murat C. Çehreli, Haldun I·plikçiogˇlu; " A Comparison of Three- Dimensional Finite Element Stress Analysis with In Vitro Strain Gauge Measurements on Dental Implants"
[12] Dr. Uri Arny, Ilan Weissberg & Oved Gihon; "Hybrid Dual Thread Screw Implant - Analytical and Experimental Research"
[13] H.G.Hanumantharaju, Dr.H.K.Shivanand; "Static Analysis Of Bi-Polar Femur Bone Implant Using Fea"
[14] Linish Vidyasagar, Peteris Apse; "Dental Implant Design and Biological Effects on Bone-Implant Interface"
[15] Ching-Chang Ko, Eduardo Passos Rocha and Matt Larson; "Past, Present and Future of Finite Element Analysis in Dentistry"
[16] Mohamed I. El-Anwar, Mohamed M. El-Zawahry and Mohamed El-Mofty; "Load Transfer on Dental Implants and Surrounding Bones"
[17] Linish Vidyasagar, Peteris Apse; "Dental Implant Design and Biological Effects on Bone-Implant Interface"
[18] Ester Orsini, Alessandra Trirè, Marilisa Quaranta, Beatrice Bacchelli, Dèsirèe Martini, Alessandro Ruggeri; "Evaluation of thread pitch as a design key factor in dental implant osseointegration"
[19] A dentist, from India, asks; "Implant Thread Design: Influence on Osseointegration"
[20] Ekachai Chaichanasiri , Pruettha Nanakorn, Wichit Tharanon , Jos Vander Sloten ; "Finite Element Analysis of Bone around a Dental Implant Supporting a Crown with a Premature Contact"
[21] Ester Orsini, Alessandra Trirè, Marilisa Quaranta, Beatrice Bacchelli, Dèsirèe Martini, Alessandro Ruggeri; "Stress Distribution in Mandible Regulated by Bone and Dental Implant Parameters: Part I - Methodology"