ISSN: 2321-8169
131 - 135

Convergence Study for Finite Element Analysis on Bone used for Dental Implant

Mrs. Archana N. Mahajan,
Civil Engineering Department,
Vishwatmak Om Gurudev College of Engineering,
Aghai, Thane, Maharashtra, India.
(e-mail: anm_sejal@yahoo.com).

Dr Ms Kshitija N. Kadamis
Applied Mechanics Department,
Government College of Engineering,
Amravati, Maharashtra, India.
(email: kadamkshitija7@ gmail.com)

Abstract—The stress analysis of bone structure used for dental implant can enhance the success rate of dental implant. The complex geometry of bone and loading on it is modeled using FEM. The paper reports on a theoretical investigation of the convergence properties of several finite element approximations in current use and assesses the magnitude of the principal errors resulting from their use for bone which is used for dental implant. Convergence analyses of finite element models of mandible jaw bone using different element size, meshing control has been conducted. The aim is to achieve minimum boundary loss with correct size of mesh and shape of the element. The results of convergence study have implemented on bone to find accurate results using Ansys software.

Keywords: Dental implant, Mandible jaw bone, Solid 185 element, Hypermesh, Ansys.

*****

I. INTRODUCTION

The ultimate purpose of finite element analysis is to create mathematically the behavior of an actual engineering system.FEM has become one of the most successful engineering computational methods and most useful analysis tool since the 1960s[1,2]. It is showing overwhelming capability and versatility in its application in dentistry[3].

Figure.1: Dental Implant with Bone

The utilization of dental insert set into the bone serves as a stay for the prosthetic gadget as appeared in Figure1 and as one of the better preventive upkeep strategies in dentistry.

SOLID185 is utilized for 3-D demonstrating of strong structures. The model era is the procedure of characterizing the geometric arrangement of the models hubs and components. The FEM utilizing polynomial shape capacities won't deliver the careful answer for issue, however we can create closest to the genuine arrangement and that can be satisfactory. The protection is given by the merging property of the FEM, which expresses that the FEM arrangement will meet to the precise arrangement that is persistent at self-assertive exactness when the component size turns out to be interminably little, and the length of the complete straight polynomial premise is incorporated into the premise to frame the FEM shape capacities. In FEA, discretization of an auxiliary model is another name for cross section era. The vast majority of the business FEA programs have the ability of naturally producing FEA network. Client needs to give the component sort, mechanical properties, limitations and burdens. The hypothetical foundation for this meeting highlight of the FEM is because of the ceaseless certainty the capacity can simply be approximated by a first request polynomial with a second request of refinement mistake. Joining studies are completed by examination where we streamline the lattice to touch base at the sought results[4]. FE arrangement shows meeting if the discretization bungle →0 as the cross section is mama interminably fine[5]. (i.e., component size →0) Researchers have utilized distinctive union criteria[6]. These are delegated takes after:

1. In anxiety union cross section is refined until the rate variety in anxiety is under 1, 5, 10 or any given quality chose by the client. When all is said in done the hassles will meet more gradually than the uprooting, so it is not adequate to analyze the dislodging convergence[7].

2. Union in vitality and removal u definite uprooting answer for an issue that makes the potential vitality of the framework a base comparing anxiety, Exact strain vitality of the b

$$\boxed{U = \frac{1}{2}\int_V \underline{\sigma}^T \ \underline{\varepsilon} \ dV}$$

FE solution (‘h’ refers to the element size) corresponding stress and strain approximate strain energy of the body.

$$\boxed{U_h = \frac{1}{2}\int_V \underline{\sigma}_h{}^T \ \underline{\varepsilon}_h \ dV}$$

3. Deflection Convergence is similar to the above convergences, except, node deflection values are used for the convergence criterion.The criteria used for monotonic convergence is completeness and compatibility.

$$\|\underline{u}-\underline{u}_h\|_0 \equiv \sqrt{\int_V \left[(u-u_h)^2+(v-v_h)^2\right] dV} \to 0 \; as \;\; h\to 0$$

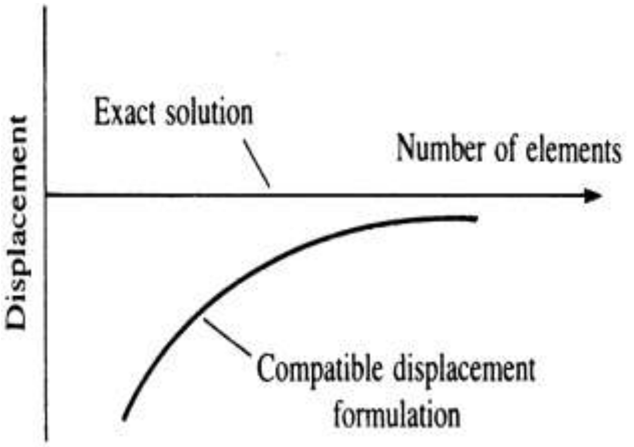


Figure.2: Convergence of a Finite Element Solution

Important property of finite element solution is the conditions of monotonic convergence are satisfied (compatibility and completeness) the finite element strain energy always underestimates the strain energy of the actual structure.The objective of this paper is to study the convergence of finite element solutionto 3 dimensional of mandible bone with free mesh as per shows Figure2.

II. METHODOLOGY

A CBCT sweep of the mandible bone has been taken to get Digital Imaging and Communications in Medicines (DICOM) design picture as appeared in Figure3. The prerequisite of real shape and size of bone from DICOM pictures used to deliver by Mimics11.11 programming. (Intuitive Medical Image Control System, Materialize Inc., Leuven, Belgium) The picture with network design must be changed over into surfaces by hypermesh to join all matrix hubs and structures isocontour with the assistance of division. The Figure4 indicates automated representation of the bone geometry which has made surface of bone. The bone have sporadic geometry so it's extremely hard to make surface. The NX (UG) programming has been utilized to deliver 3D model appeared in Figure5. The surface smoothing is not performed on the bone surface in order to increment in arrangement accuracy[8].

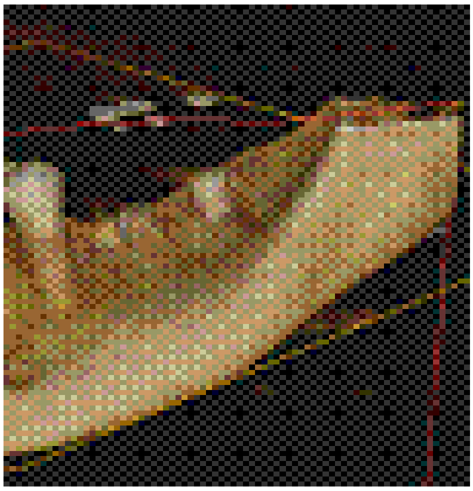


Figure.3: DICOM Image of Mandible Bone

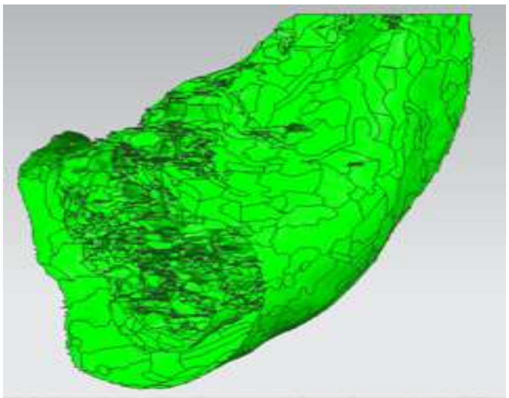


Figure.4: Surfaces created in Hypermesh

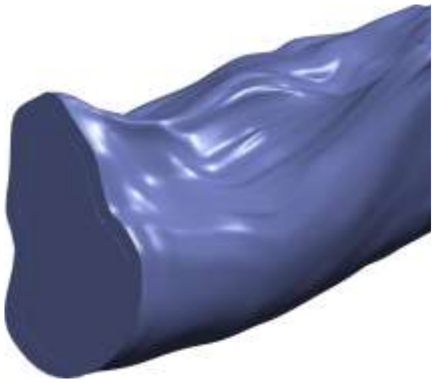


Figure.5: 3 D Model

A free mesh has placed to bone model as shown in Figure 6, because no restrictions in terms of element shapes and has no specified pattern applied to it.

ISSN: 2321-8169
131 - 135

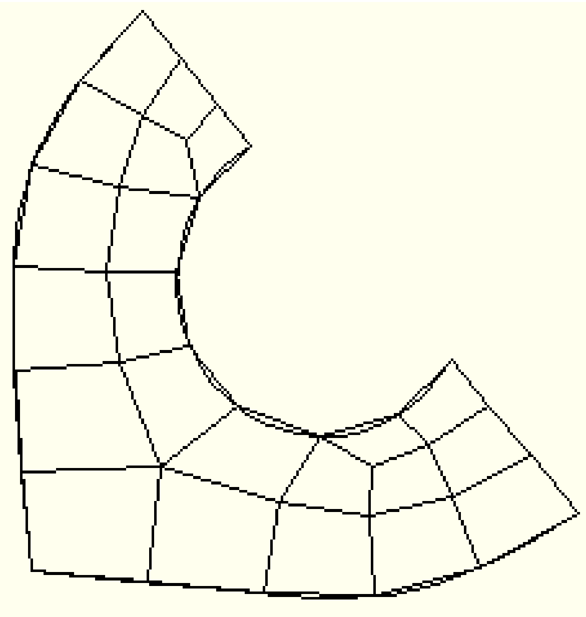


Figure.6: Free Pattern of Mesh

The system for creating a lattice of hubs and components comprises of steps are as per the following:

1. Set of the component traits

Before creating lattice of hubs and components, characterize the component name can be looked over the Ansys component library comprises of more than 100 diverse component definitions or sort. The component has versatility, hyperelasticity, stress solidifying, creep, vast avoidance, and substantial strain capacities. It likewise has blended definition capacity for mimicking disfigurements of almost incompressible elastoplastic materials, and completely incompressible hyperelasticmaterials[9].

Genuine consistent sets, for example, thickness or cross sectional region required for the count of the component lattice yet which can't be resolved from the hub areas, or material properties are data as genuine steady set. Different materials properties are utilized for every component sort. Average material properties incorporate Young's modulus, thickness and so forth. The component coordinate framework is utilized for orthotropic material, information bearings, connected weight headings, and under a few circumstances, stress yield bearings. Anisotropic versatile ability is accessible with the strong 185 auxiliary component. The hex component has eight hubs, each having three DOFs, interpretation in the nodal x, y, z bearings. The tetrahedral component has four hubs, each having three DOFs in the nodal x, y, z headings, making the aggregate DOFs in a tetrahedron component twelve. The hex and tetrahedral component appears in Figure7.

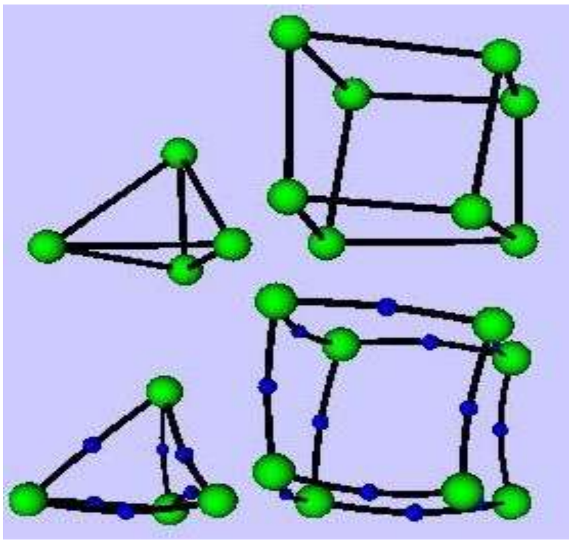


Figure.7: Hex and Tetrahedral Element

2. Set mesh controls

Network controls permit us to build up such components as the component shape, midside hub situation, and component size to be utilized as a part of cross section strong model. This stride is a standout amongst the most critical of whole examination for the choices of model advancement will significantly influence the exactness and economy of investigation.

3. Create the cross section

The hypermesh gives an advantageous way to a large portion of the most widely recognized cross section controls and also to the most much of the time performed coinciding operations.

The hex or tetramesh highlight can be gotten to by from the lattice menu, point to 3-D, and snap Hex or Tetra network. On the 3D page, tap the hex or tetramesh sub-board. The hex or tetramesh board permits client to fill an encased volume with first or second request hex or tetrahedral components. An area is viewed as encased in the event that it is totally limited by a shell lattice where every component has material on one side and open space on the other. In Figure8 demonstrates that use of limit conditions on bone edge load 110 N, insertion torque is 300Nmm and altered backing. The Figure 9 and 10 demonstrates component fitting sort is hex and tetrahedral work separately. Higher request components are helpful if the limit of the geometry is bend in nature[10].

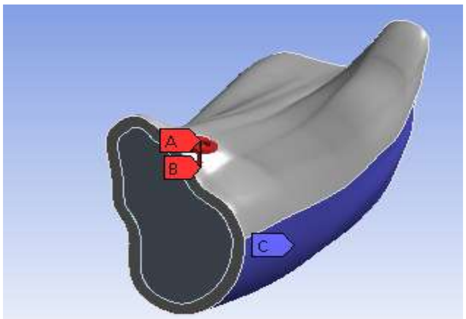


Figure.8: Boundary Condition A:110 N, B:300N.mm,C:Fixed Support

ISSN: 2321-8169
131 - 135

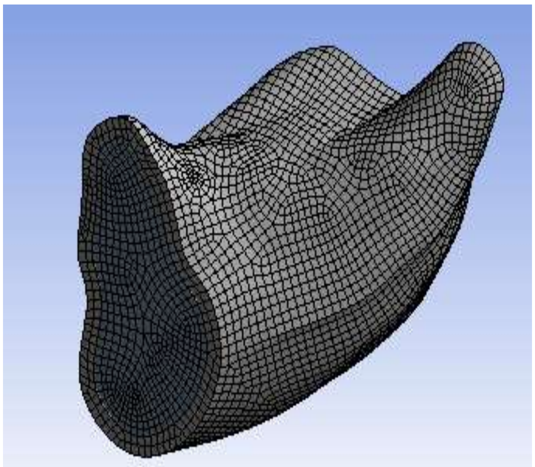


Figure.9: Hex Meshing on Bone

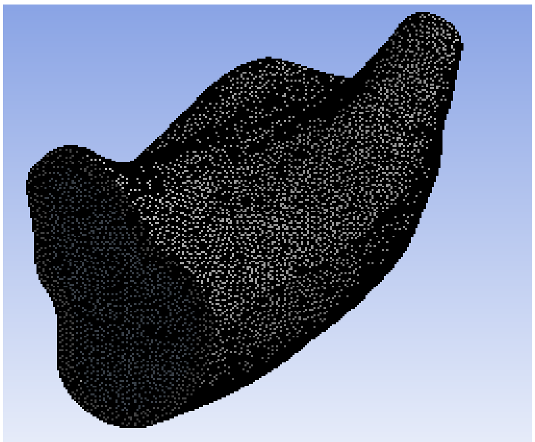
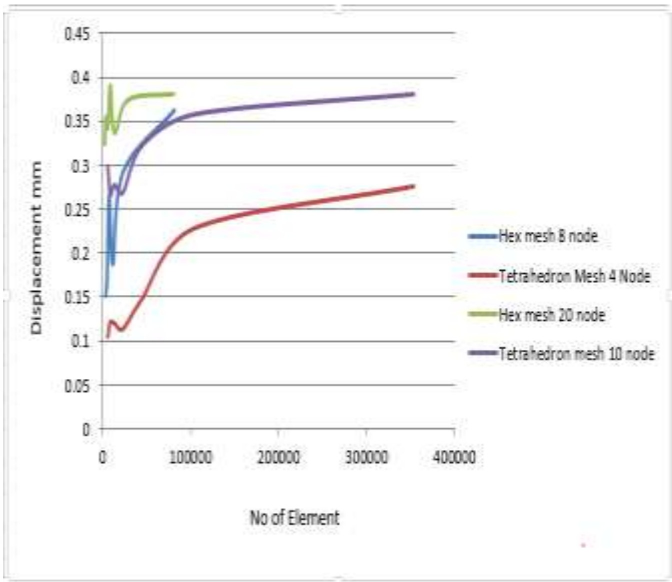
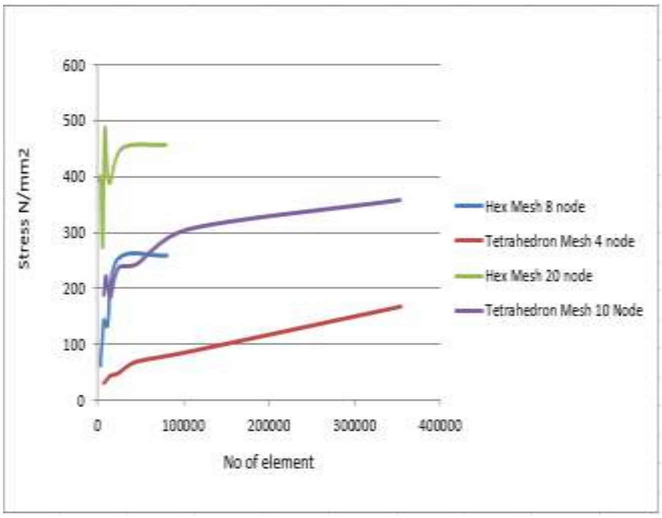


Figure.10: Tetrahedron Meshing on Bone



Graph No.1: Convergence of Displacement with Number of Elements



Graph No.2: Convergence of Stress with Number of Elements

In the convergence study Fem gives number of results from that have to find desired solution of any model. As per Graph No.1 shows that hex mesh 20 noded and tetrahedron mesh 10nodedcurves nearer to each other and given constant compatible displacement formulation with no of element. Hex mesh 8noded curve shows that number of element increases the curve will be going to intersect to other convergence curves of 10 noded tetrahedron and 20 noded hex mesh. Therefore 8nodedhex element is discarded. The tetrahedron 4 noded mesh curve is away from the other curves that indicated this element is not suitable for irregular boundaries of bone model. The most suitable element from displacement convergence is hex 20nodedmesh and may be giving the desired results.

In the stress convergence Graph No.2 shows that tetrahedron mesh 4 noded and 10noded curves parallel to each other and also hex mesh 8 noded and 20noded parallel to each other. The curves of 20 noded hex mesh and 10noded tetrahedron shows the stress convergence with discretization error is less. The hex 20noded mesh is most suitable from stress convergence with lesser no of elements.

III. CONCLUSIONS

This convergent study is very essential since true solution of this complicated analysis is not available. so convergence study becomes crucial. convergence study of displacement reveals that thetetrahedran element with 4 or 10 nodes show slow convergence while hex element with 8 or 20 nodes show fast convergence. Higher order elements i.e. 10 noded tetrahedron and 20noded hex elements are showing asymptotic nature where as lower order elements have resulted in bad convergence. The complex curvatures of bone boundaries need to be modeled using higher order element. Stress being secondary unknown in analysis also show similar trend in convergence study. The convergence study of stress also show suitability of higher order elements for this analysis. amongst presented higher order elements, 20noded hex element is recommended here for since it resulted fast convergence. This study gives an impetus to application of FEM in the field of Bioengineering.

REFERENCES

[1] Ergatoudis, I., Irons, B.M., and Zienkiewicz, O.C., 1968, "Curved, isoparametric, quadrilateral elements for finite element analysis," International Journal of Solids and Structures, **4**(1), 31-42.

[2] Rzemieniecki, J.S., 1969, "Theory of matrix structural analysis. Journal of Sound and Vibration,"10(2), 358-359.

[3] Watanabe, F., Hata, Y., Komatsu, S., Ramos, T.C., and Fukuda, H., 2003,"Finite element analysis of the influence of implant inclination, loading position, and load direction on stress distribution," Odontology, 9(1),31-36.

[4] Joseph E Walz, Robert E, Fulton and Nancy Jane Cyrus,"Accuracy and Convergence of Finite Element Approximation,"Nasa Langley Research Centre, Langley Station, Hampton, Va.

[5] Prof.SuvranuDe," Introduction to Finite Elements-Convergence of analysis results," ppt.

[6] A.S. Meghre,Ms K. N. Kadam," Finite Element Method in Structural Analysis," Khanna Publishers.

[7] FEM Convergence Testing, Copyright @2001, University of Alberta.

[8] Archana N Mahajan, DrMs K N Kadam," Stresses on Single Dental Implant Cement and Screw Retained Crown by 3 D Finite Element Analysis,"60thCongress of the Indian Society ofTheoretical and Applied Mechanics. (An International Conference)

[9] Solid 187 Element Description Inside Mines, inside.mines.edu.

[10] Ansys, Inc, Southpointe,225, Technology Drive, Canonsburg, PA15317, ansysinfo@ansys.com, http://www.ansys.com.

Author Profile

Archana N. Mahajan received the B.E. degree in Civil Engineering from KarmveerKakasahebWagh College of Engg. Nasik, in 1997 and M.E. degree in Structural Engineering from Amrutvahini College of Engg. Sangamner, in 2012 and currently doing Ph. D. from Amravati University.

Dr. Kshitija N. Kadam has a qualification of Ph. D. and currently working in Government College of Engg, Amravati. She has an experience of 20 years in the field of teaching & research. To her credit she has 45 research papers and **'The John C Gammon Prize'** to one of her research paper from Indian Institution of Engineers (India).