IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

"Effect of Fibre Volume Fraction and Fibre Reinforcement Orientation (Various Layup Sequence) on Mechanical Properties of Epoxy-based E-Glass fibre Reinforced Bio-material Used as Implant (Human Bone -Tibia) - Finite Element Method."

[1] Dr.Shivanand N. Pujar, [2]Dr. K.R. Dinesh, [3]Dr. Jagdish S.P, [4]Dr. Shiva Kumar Gouda
[1]Senior Scale Lecturer, [2]Professor and Principal, [3]Associate Professor, [4]Associate Professor
[1,2,3,4]Department of Mechanical Engineering
[1]Senior Scale Lecturer, GPT Hubli, Karnataka, India, [2]Government Engineering College, KR Pete, India,
[3]R.Y.M Engineering College, Bellary, Karnataka, India, [4]Deputy Dean CIII, SDMCE-Dharwad, Karnataka, India.

The present research is aimed to evaluating the effect of fibre Volume fraction and fibre orientation on mechanical properties of Epoxy based E-Glass Reinforced Bio Composites using Finite Element Method, particularly using ANSYS 14.5 Software. An effort is made to understand the behaviour of E-glass reinforced Bio-Composites by considering the fibre Volume Fraction as 10%, 20%, 30% and 40%. We aimed to set up a mathematical, computer-based model using finite element analysis simulation software, ie ANSYS 14.5. The plate or shell type element were used to prepare the specimen according to ASTM principles D-3039/D-3039M-00. Tensile and Buckling tests were simulated to evaluate mechanical properties such as tensile and Buckling strength, yield strength, from this study the fibre Volume Fraction of 10% will be the promising the component for highest Ultimate strength. Then keeping constant 10% fibre Volume Fraction, then by evaluating the different orientation for Various Layup Sequences such as [0/0/0/0], [90/90/90/90], [0/90/90/0], [0/90/0/90], [90/0/0/90], [30/-30/30/-30], [45/-45/45/-45], [60/-60/60/-60] for Ultimate Tensile and Buckling Strength, the fibre Layup Sequences [0/90/0/90] shows the very near by value of Ultimate Tensile Strength that is approximately equal to Human Tibia Bone.

Keywords: E-Glass fibre reinforced, Epoxy, Tibia Human Bone, Fibre Volume Fraction, Fibre Layup Sequences.

1.INTRODUCTION

The Benchmark is a standard test problem with known target solution in the form of formulae, graphs or tables obtained using analytical, experimental and numerical methods used to validate Finite Element Models developed using ANSYS software were reported in Appendix -1,

[Appendix -1 Here we used the Experimental Results Reported by ***Hayder A. Rasheed, Rund Al-Masri, BacimAlali, Closed form stability solution of simply supported anisotropic laminated composite plates under axial compression compared with experiments, Engineering Structures 151 (2017) 327–336,*** and simulation were done to reproduce the same results, hence the deck preparation material modelling and assigning the ply thickness and orientation of ply are correct]

Hence, we have followed the same procedure in ANSYS to preparate the FE- Model for the current Study. Now a days the population strength increases day by day along with that the utilization of vehicles also more due to that the accidents are also increases. In the medical field the engineers are searching for the new material with research work and also it is a challenging job to the research scholars/scientists to invent new material to society. **[1] by *Hayder A. Rasheed, Rund Al-Masri, BacimAlali,*** *Closed form stability solution of simply supported anisotropic laminated composite plates under axial compression compared with experiments, Engineering Structures 151 (2017) 327–336,* They have created Benchmark problem for standard test problem with known target solution in the form of formulae, graphs or tables obtained using analytical, experimental and numerical methods used to validate Finite Element Models developed using ANSYS, **[2] Andrew Kemper, Craig McNally, Eric Kennedy, Sarah Manoogian**: The purpose of this study was to develop the tensile and compressive material properties for human tibia cortical bone coupons when subjected to different loading and calculated the Tibia Index. The Tibia Index is an injury tolerance criterion for combined bending and axial loading experienced at the midshaft of the leg. This research paper reports the Compressive, tensile and bending Strength of the Tibia Bone. Has noted the mechanical property of Human Tibia Bone in the age group of 20to 65 years and reported that , Ultimate Tensile Strength may vary from141 MPa to 192MPa, Ultimate Compressive strength may be 151 MPa to 183 MPa. **[3] Mr. Amit Patil** : The tibia is a long hollow leg bone, which has an expanded metaphysic and an epiphysis at both ends of a thick walled tabular diaphysis. The proximal end of the human tibia displays very special characteristics, The tibia 3 D CAD model is required for FEA analysis, it was obtained by using industrial 3D scanner. The obtained results are most useful for providing information to doctor for the risk of possible fractures location in human tibia bone.

METHOD AND METHODOLOGY

Composite materials.

A composite is defined as a material that consists of two or more phases combined at a macroscopic level and are not chemically soluble in each other. One phase is called reinforcement and other in which it is embedded is called the matrix. The reinforcement may be in the form of continuous fibers, woven fabric, short length fibers, particles or flakes. The matrix is generally continuous. In a unidirectional lamina all fibers are oriented in the same direction. Cross ply laminate has the layers of unidirectional lamina oriented at right angles to each other.

2.1 Finite Element Method

The finite element method is a numerical analysis technique used by engineers, scientists and mathematicians to obtain solutions to the differential equations that describes or approximately describes a wide variety of physical (and non-physical) problems. Physical problems range in diversity from solid, fluid and solid mechanics, to electromagnetism or dynamics. Hence, Finite Element Analysis (FEA) is being considered a part of the design process spanning across industries or domains, be it automotive, aerospace, medical, civil, electrical, etc.

The underlying premise of the method states that a complicated domain can be sub-divided into a series of smaller regions in which the differential equations are approximately solved. By assembling the set of equations for each region, the behaviour over the entire problem domain is determined.

Each region is referred to as an element and the process of subdividing a domain into a finite number of elements is referred to as discretization. Elements are connected at specific points, called nodes, and the assembly requires that the solution be continuous along common boundaries of adjacent elements.

2.2 Basic steps in finite element analysis

2.2.1 Discretization

The first step is discretization of a given domain using finite elements. The domain can be a solid, a liquid, a gas, or their combinations. A library of finite elements of different types, shapes and orders is available for this purpose. Each element has a finite number of nodes and each node a finite number of degrees of freedom, which are the fundamental unknowns. The elements are interconnected at their nodes only and the finite element mesh is generated using a pre-processor. Commercial pre-processor has capability for automated mesh generation and adaptive mesh refinement.

2.2.2 Selection of interpolation functions

Once an element shape has been chosen, the analyst must determine how the variation of the field variables across the element domain is to be represented or approximated. In most cases, a polynomial interpolation function is used. The number of nodes assigned to an element dictates the order of the interpolation function, which can be used. Interpolation functions are also referred to as shape functions or approximating functions. If the analyst is using an existing finite element package, then most likely the choice of interpolation function is implicit in the choice of element type.

2.2.3 Derivation of element equations

Once the interpolation functions have been chosen, the field variables in the domain of the element are approximated in terms of discrete values at the nodes. Consequently, a system of equations is formed which express the element properties in terms of quantities at the nodes.

2.2.4 Assembly of element equations

The assembly procedure combines each element approximation of the field variable (as defined in the previous steps) to form a piecewise approximation of the behaviour over the entire solution domain. Assembly is accomplished using the following basic role of compatibility. The value of the field variable at a node must be the same for each element that shares that node. This step is handled automatically by finite element package.

2.2.5 Application of the boundary conditions

The global system of equations created in the previous steps cannot be solved, pending application of the boundary conditions. Mathematically, before applying the boundary conditions, the system of equations is indeterminate and does not have a unique solution. In the same way that a structure must be physically fixed to the ground to prevent it from moving when a force is applied, anode must also be fixed to the ground.

In finite element terminology, a node has prescribed boundary condition if the value of the field variable is already known. For example, a prescribed nodal displacement of zero is assigned to fixed supports of a structure. In a thermal analysis, temperature is prescribed at one or more nodes. Usually these nodes lie on the boundaries of the finite element mesh.

2.2.6 Solution of the system of equations

Once the boundary conditions have been applied to the assembled matrix of equations, standard numerical techniques can be used to solve for the unknown field variable at each node. If the system of equations is linear, a Gaussian elimination or Cholesky decomposition algorithm can be used.

In structural analyses, the matrix equations generated by the finite element process are often sparsely populated and symmetrical. Many solution methods make use of these properties to provide fast and efficient computation algorithms, which are now implemented in all finite element packages.

In dynamic analysis, the matrix equation can be reduced to a sub-set by taking advantages of the fact that the lower natural frequencies are governed mainly by the structural parts that have the highest mass to stiffness ratio.

If the set of equations is non-linear, then the solution procedure is more difficult to obtain. The choice of the solution method is very much dependent on the size of the problem as well as the type of analysis.

2.2.7 Post-processing

The result obtained are observed and verified in the form of plots, graphs and lists. Keeping in mind the assumptions and the boundary conditions the user can refine the model and follow the above procedure until he is thoroughly satisfied.

2.3 Finite Elements

The following element types are available in ANSYS to model layered composite materials: SHELL181, SHELL281, SOLID185 Layered Solid, and SOLID186 Layered Solid.

SHELL181 is suitable for analysing thin to moderately-thick shell structures. It is a four-node quadrilateral element with six degrees of freedom at each node: translations in the x, y, and z directions, and rotations about the x, y, and z-axes. (If the membrane

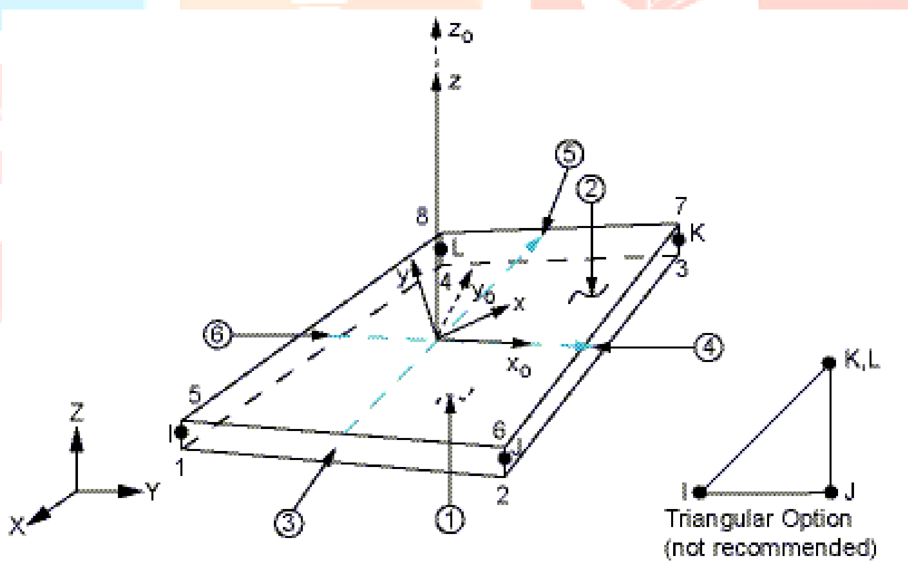


Figure 2.1 SHELL181 Geometry

option is used, the element has translational degrees of freedom only). The degenerate triangular option should only be used as filler elements in mesh generation.

SHELL181 is well-suited for linear, large rotation, and/or large strain nonlinear applications. Change in shell thickness is accounted for in nonlinear analyses. In the element domain, both full and reduced integration schemes are supported. SHELL181 accounts for follower (load stiffness) effects of distributed pressures.

SHELL181 may be used for layered applications for modeling composite shells or sandwich construction. The accuracy in modeling composite shells is governed by the first-order shear-deformation theory (usually referred to as Mindlin-Reissner shell theory).

The element formulation is based on logarithmic strain and true stress measures. The element kinematics allow for finite membrane strains (stretching). However, the curvature changes within a time increment are assumed to be small.

2.5 Buckling analysis using ANSYS

Buckling analysis is a technique used to determine buckling loads (critical loads at which a structure becomes unstable) and buckled mode shapes (the characteristic shape associated with a structure's buckled response).

Eigenvalue buckling analysis predicts the theoretical buckling strength (the bifurcation point) of an ideal linear elastic structure. (Figure 2.5.1) This method corresponds to the textbook approach to elastic buckling analysis: for instance, an eigenvalue buckling analysis of a column will match the classical Euler solution. However, imperfections and nonlinearities prevent most real-world structures from

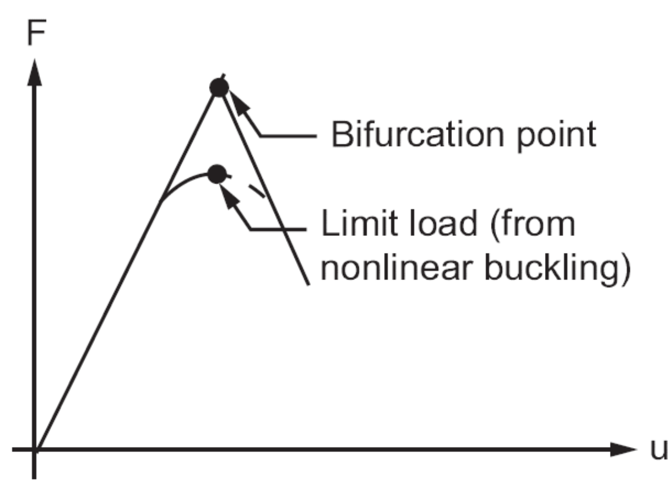


Figure 2.2 Linear (eigenvalue) buckling curve

achieving their theoretical elastic buckling strength. Thus, eigenvalue buckling analysis often yields unconservative results, and should generally not be used in actual day-to-day engineering analyses.

To calculate the stiffness matrix, Prestress effects (PSTRES) were used in the eigenvalue buckling analysis. Unit loads are usually sufficient (that is, actual load values need not be specified). The eigenvalues calculated by the buckling analysis represent buckling load factors. The methods available for buckling analysis are Block Lanczos and Subspace Iteration.

Tensile test of composite plate shells corresponding to volume fraction of E-glass/epoxy.

a) Geometric properties

Geometric/Finite Element modelof a plate shell configuration according to ASTM principles D-3039/D-3039M-00is shown in figure 2. Specimens were 250 mm in length, 25 mm wide and 2.5 mm (i.e. 4 plies) thick. The overall gauge-length (i.e. region between grips) was 150 mm. End tabbing was identical to that used for the unidirectional specimens with square end tabs.

b) Laminate details

The lamination angle is measured with respect to the x-axis (i.e. 0^0fibers run parallel to the x-axis and 90^0fibers run parallel to the z-axis). Accordingly, the angle is rotated about the y-axis.as shown in Figure -1.

No. of plies = 4

Ply thickness = 0.625 mm

Layup = [0/90/0/90]

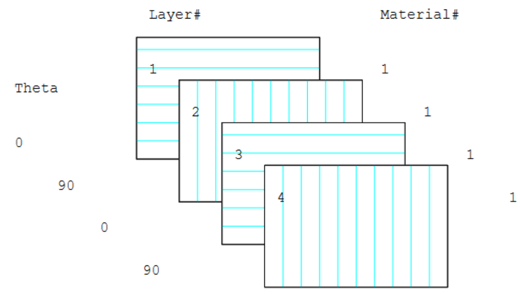


Figure 1. Ply Orientation .

c) Material details

Table -1. Material properties

Property	**E-glass /epoxy**			
Fiber volume fraction, V_f	0.1	0.2	0.3	0.4
Matrix volume fraction, V_m	0.9	0.8	0.7	0.6
Young's Modulus in the 1-direction, E_1 [GPa]	10.36	17.32	24.28	31.24
Young's Modulus in the 2-direction, E_2[GPa]	3.75	4.2	4.76	5.44
Young's Modulus in the 3-direction, E_3 [GPa]	3.75	4.2	4.76	5.44
Shear Modulus in 1-2 direction, G_{12} [GPa]	1.6	1.74	1.91	2.1
Shear Modulus in 2-3 direction, G_{23} [GPa]	1.28	1.39	1.52	1.68
Shear Modulus in 3-1 direction, G_{31} [GPa]	1.28	1.39	1.52	1.68
Poisson's ratio in 1-2 direction, ν_{12}	0.33	0.32	0.3	0.29
Poisson's ratio in 2-3 direction, ν_{23}	0.33	0.32	0.3	0.29
Poisson's ratio in 3-1 direction, ν_{31}	0.33	0.32	0.3	0.29

d) Element details

Shell 181, 4-noded quadrilateral shell element, with four nodes and six degrees of freedom per node.

e) Mesh details

The mesh was progressively refined to ensure convergence.

Type of mesh: Mapped mesh

Number of elements: 962

Number of nodes: 1050

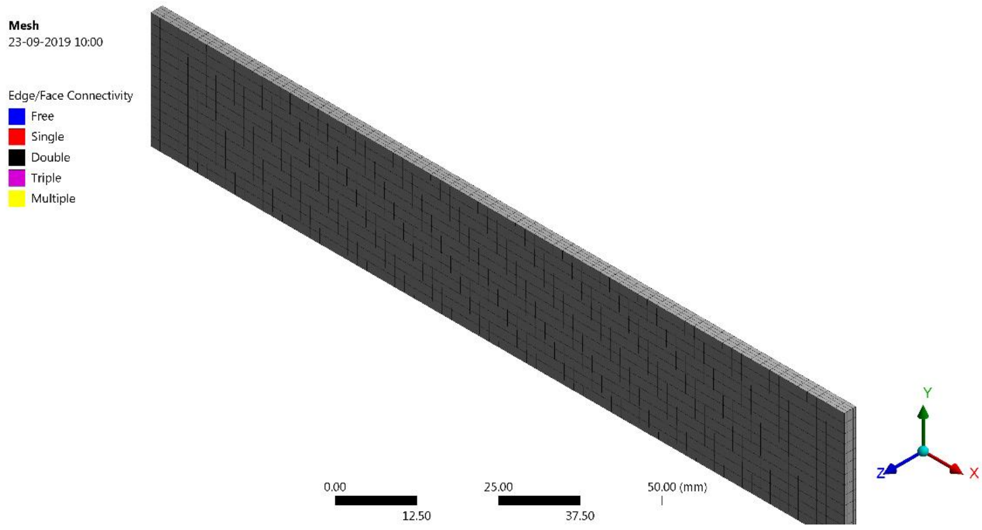


Figure 2. Finite element model of a plate shell

f) Boundary condition

A plate shell with one end clamped and the othersubjected to external force, P = 23400 N, as shown in figure 3.

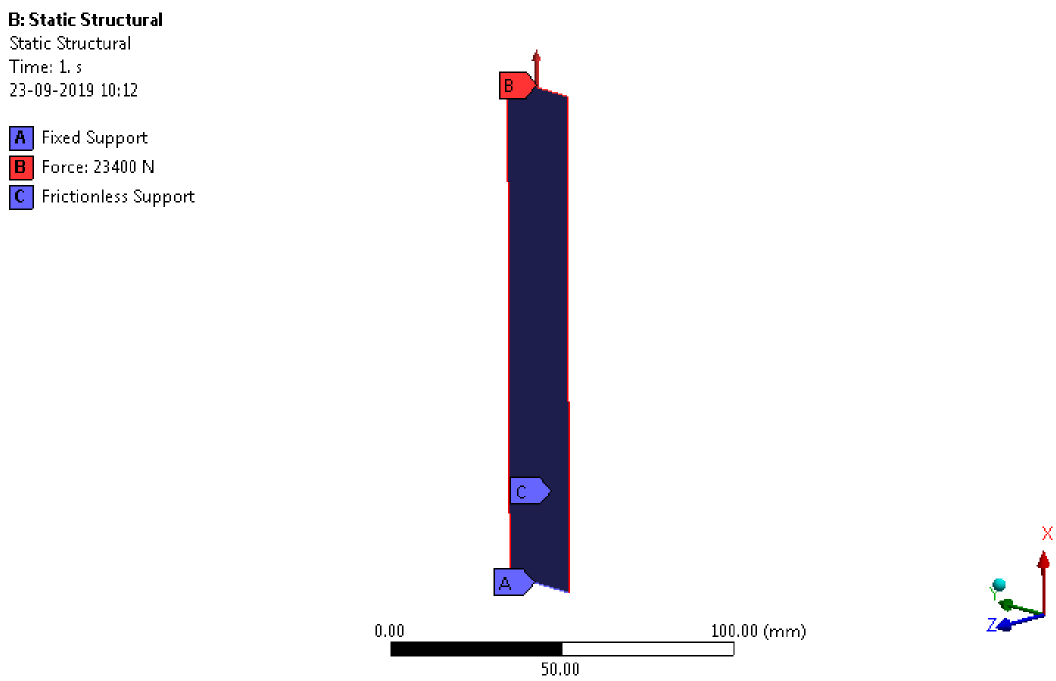


Figure 3. Finite element model of a plate shell subjected to external force, P (N).

Results presentation and discussion

Table -2. Tensile test for various volume fraction of E-glass/Epoxy Composite Plate.

Fiber volume fraction, V_f	Matrix volume fraction, V_m	Deformation [mm]	Stress [N/mm^2]	Stiffness [N/mm]
0.1	0.9	6.2	169.7	27.3
0.2	0.8	4.1	133.5	32.5
0.3	0.7	3	123.8	41.2
0.4	0.6	2.4	119	49.5

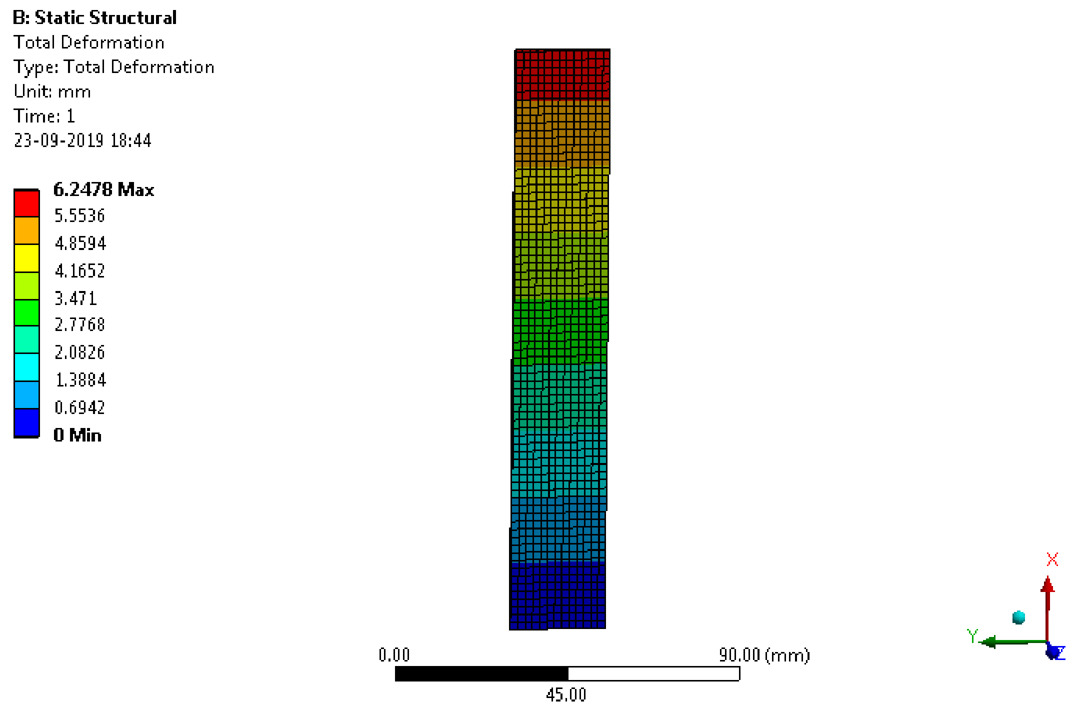


Figure.4. Total deflection of E-glass/Epoxy.

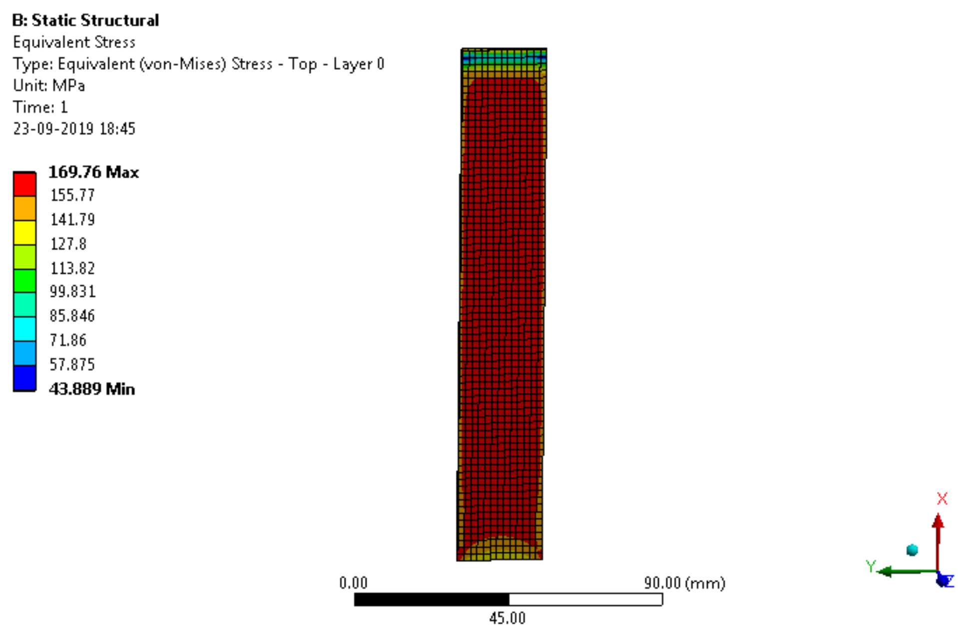


Figure-5 Von-misses stress of E-glass/Epoxy.

Conclusion from the Volume Fraction Analysis.

From the obtained results for different Fibre Volume Fraction ie for 10%,20%,30% and 40% , The Ultimate Tensile Strength obtained is 169.7 N/mm2 , Which is equal to the Tibia bone tensile Strength hence for the further research considered the 10% fiber Volume fraction. Then keeping constant Volume fraction then varying the fiber orienation for different Layup Sequences, the folowing simulation are carried out.

Tensile test of composite plate shells corresponding to various layup sequences.

Finite Element Model

Laminate details

The lamination angle is measured with respect to the x-axis (i.e. 0^0fibers run parallel to the x-axis and 90^0fibers run parallel to the z-axis). Accordingly, the angle is rotated about the y-axis. No. of plies = 4 , Ply thickness = 0.625 mm

Layup = [0/0/0/0], [90/90/90/90], [0/90/90/0], [0/90/0/90], [90/0/0/90], [30/-30/30/-30], [45/-45/45/-45], [60/-60/60/-60].

Material details

Table -3. Material properties

Property	E-glass /epoxy
Fiber volume fraction, V_f	0.1
Matrix volume fraction, V_m	0.9
Young's Modulus in the 1-direction, E_1 [GPa]	10.36
Young's Modulus in the 2-direction, E_2[GPa]	3.75
Young's Modulus in the 3-direction, E_3 [GPa]	3.75
Shear Modulus in 1-2 direction, G_{12} [GPa]	1.6
Shear Modulus in 2-3 direction, G_{23} [GPa]	1.28
Shear Modulus in 3-1 direction, G_{31} [GPa]	1.28
Poisson's ratio in 1-2 direction, ν_{12}	0.33
Poisson's ratio in 2-3 direction, ν_{23}	0.33
Poisson's ratio in 3-1 direction, ν_{31}	0.33

Results presentation and discussion

Table -4. Tensile load for various layup sequences of E-glass/Epoxy Composite Plate.

Layup	Deformation [mm]	Stress [N/mm^2]	Stiffness [N/mm]
[0/0/0/0]	4.3	340	79
[90/90/90/90]	11.8	312	26.4
[0/90/90/0]	6.2	472	76
[0/90/0/90]	**6.2**	**169**	**27.2**
[90/0/0/90]	6.3	260	41.2
[30/-30/30/-30]	6.6	617	93.4
[45/-45/45/-45]	9.5	648	68.2
[60/-60/60/-60]	11.4	496	43.5

Conclusion from the Various Layup Sequences Analysis.

From the obtained results for different layup sequences ie for [0/0/0/0], [90/90/90/90], [0/90/90/0], [0/90/0/90], [90/0/0/90], [30/-30/30/-30], [45/-45/45/-45], [60/-60/60/-60]. The Ultimate Tensile Strength obtained is 169.7 N/mm2 , Which is equal to the Tibia bone tensile Strength hence for the further research considered the [0/90/90/0] fiber Reinforcement Orienation.

Then keeping constant Volume fraction[10%] and the fiber orienation for different Layup Sequences [0/90/90/0], then Buckling Behaviour of E-glass Reinforced Bio-compostes are analysed and reported here.

Buckling behaviour of composite plate shells corresponding to various layup sequences

Figure -6 shows a plate shell made of laminated composite material, with one end clamped and the other subjected to external force. The Geometric parameters used in computation are: Length L of 155 mm, width of 25 mm, thickness t of 3.17 mm. The material properties are listed in Table-4.The buckling behaviour characteristics are determined for each case while changing the fibre orientation angle in the laminate plate.

Finite Element Model

Geometric/Finite Element modelof a plate shell configuration according to ASTM principles D-3410/D-3410M-03is shown in figure-6. Specimens were 155 mm in length, 25 mm wide and 3.17 mm (i.e. 4 plies) thick. The overall gauge-length (i.e. region between grips) was 25 mm. End tabbing was identical to that used for the unidirectional specimens with square end tabs.

Laminate details

No. of plies = 4, Ply thickness = 0.7925 mm

Layup = [0/0/0/0], [90/90/90/90], [0/90/90/0], [0/90/0/90], [90/0/0/90], [30/-30/30/-30], [45/-45/45/-45], [60/-60/60/-60]

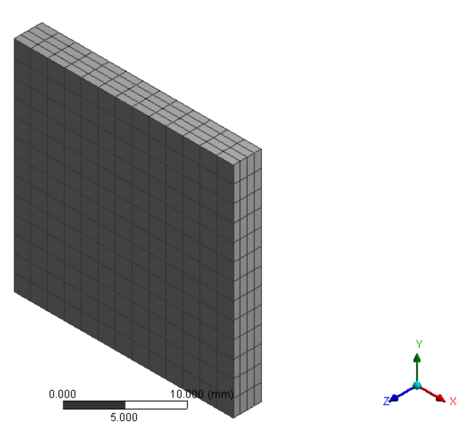


Figure -6. FE- model of composite plate shell

Material details[x],[x]

Table - 4 Material properties

Property	E-glass /epoxy
Fiber volume fraction, V_f	0.1
Matrix volume fraction, V_m	0.9
Young's Modulus in the 1-direction, E_1 [GPa]	10.36
Young's Modulus in the 2-direction, E_2[GPa]	3.75
Young's Modulus in the 3-direction, E_3 [GPa]	3.75
Shear Modulus in 1-2 direction, G_{12} [GPa]	1.6
Shear Modulus in 2-3 direction, G_{23} [GPa]	1.28
Shear Modulus in 3-1 direction, G_{31} [GPa]	1.28
Poisson's ratio in 1-2 direction, ν_{12}	0.33
Poisson's ratio in 2-3 direction, ν_{23}	0.33
Poisson's ratio in 3-1 direction, ν_{31}	0.33

g) Element details

Shell 181, 4-noded quadrilateral shell element, with four nodes and six degrees of freedom per node

h) Mesh details

The mesh was progressively refined to ensure convergence.

Type of mesh: Mapped mesh

Number of elements: 169

Number of nodes: 196

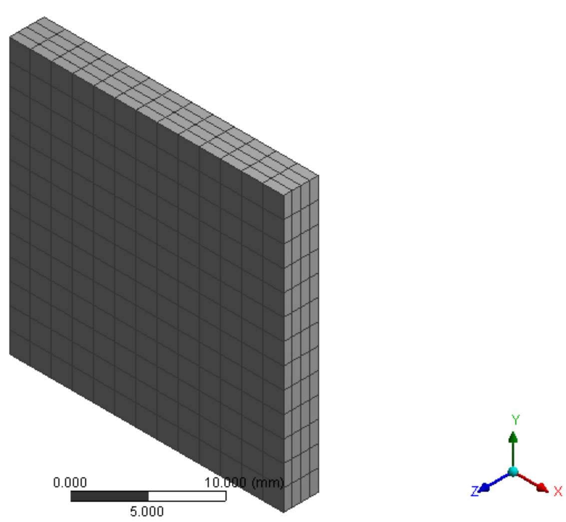


Figure -7 Finite element model of a plate shell

i) Boundary condition

A plate shell with one end clamped and the othersubjected to external force, P= 1 N, as shown in figure x.x.

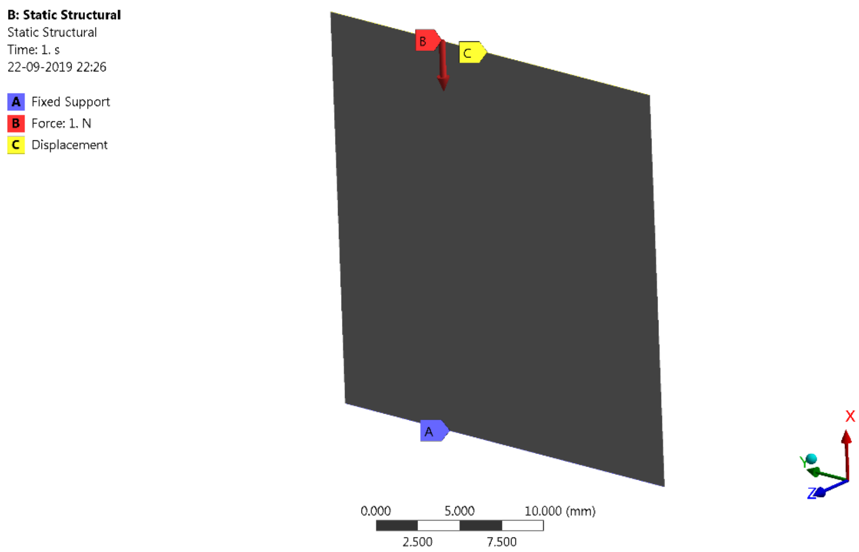


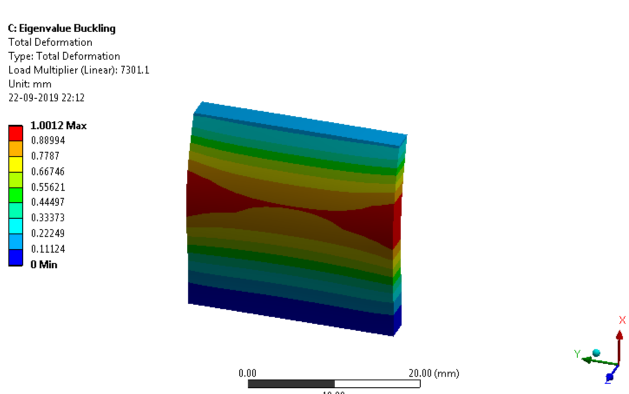
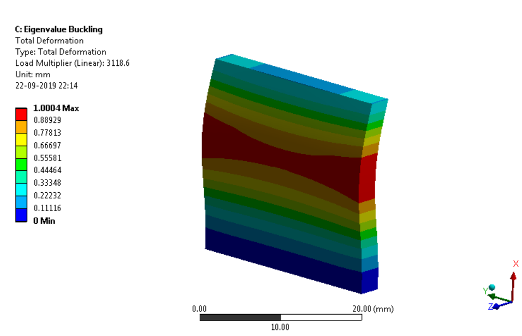
Figure -8 Finite element model of a plate shell subjected to external force, P = 1 N.

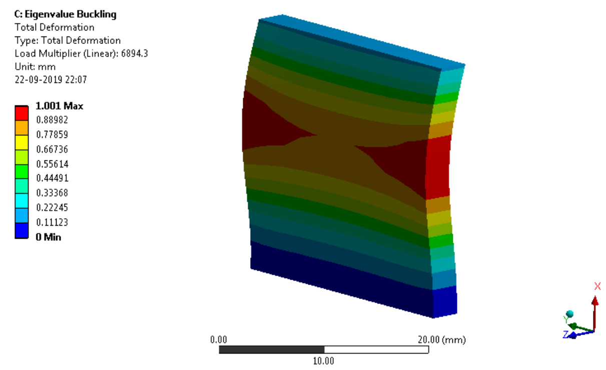
Results presentation and discussion

Table -5 Buckling load for various layup sequences of E-glass/Epoxy Composite Plate.

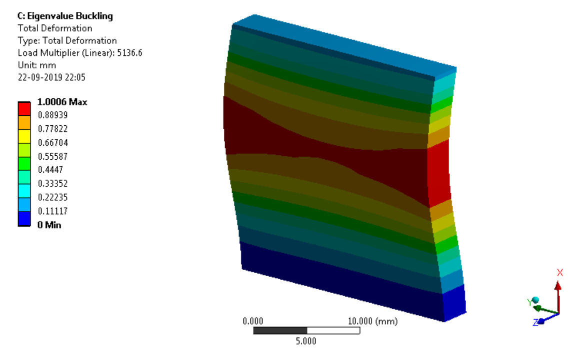
Layup	Buckling Load [N]
[0/0/0/0]	7301
[90/90/90/90]	3118
[0/90/90/0]	6894
[0/90/0/90]	5136
[90/0/0/90]	3686
[30/-30/30/-30]	5304
[45/-45/45/-45]	4137
[60/-60/60/-60]	3411

Table -6 Mode shapes of buckling of E-glass/ epoxy composite shells corresponding to various layup sequences

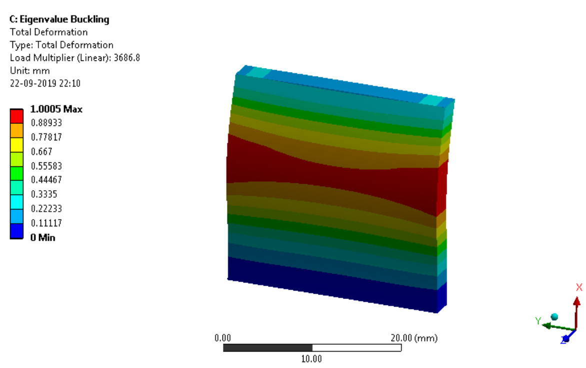
 **N = 1, [0/0/0/0], P=7301 N**	 **N = 1, [90/90/90/90], P=3118 N**



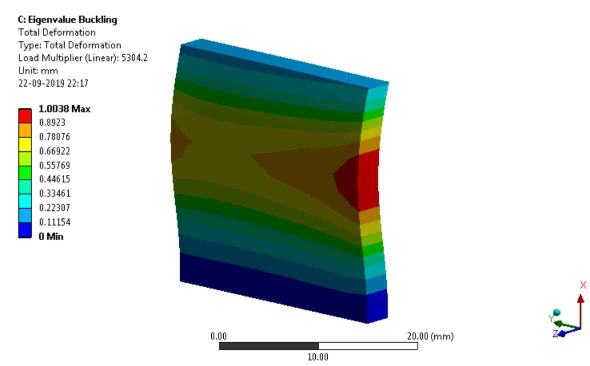
N = 1, [0/90/90/0], P=6894 N



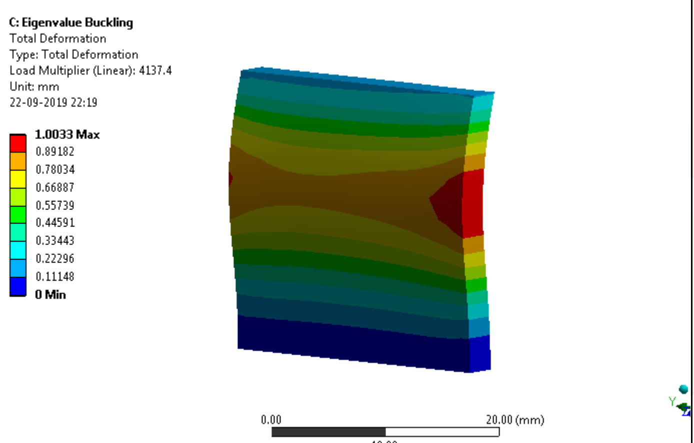
N = 1, [0/90/0/90], P=5136 N



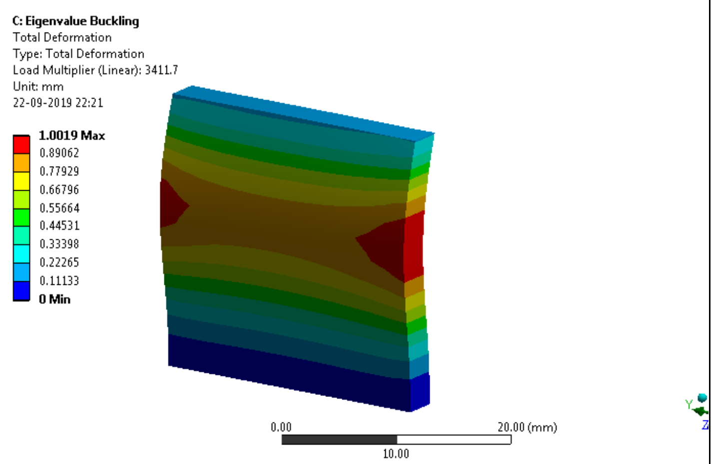
N = 1, [90/0/0/90], P=3686 N



N = 1, [30/-30/30/-30], P=5304 N



N = 1, [45/-45/45/-45], P=4137 N



N = 1, [60/-60/60/-60], P=3411 N

Conclusion Buckling Analysis.

Computed critical buckling pressures (kN/m^2) when various layup sequences of [0/0/0/0], [90/90/90/90], [0/90/90/0], [0/90/0/90], [90/0/0/90], [30/-30/30/-30], [45/-45/45/-45], [60/-60/60/-60] are given in Table-5.for E glass /epoxy.

Computed values of critical buckling loads corresponding to various layup sequences are plotted against in Table-5forE glass /epoxy composite shells. It is noted that the critical buckling load of [0/0/0/0] are significantly higher than [0/90/90/0], [90/90/90/90], [0/90/0/90], [90/0/0/90], [30/-30/30/-30], [45/-45/45/-45], [60/-60/60/-60]. The ply orientation angle θ has a significant effect on the buckling behavior.Mode shapes of buckling of high modulus E glass / epoxy composite shells corresponding to various layup sequences are shown in Table 6.

Appendix-1

Benchmark is a standard test problem with known target solution in the form of formulae, graphs or tables obtained using analytical, experimental and numerical methods used to validate Finite Element Models developed using ANSYS software.

According to Hayder A. Rasheed, Rund Al-Masri, BacimAlali, *Closed form stability solution of simply supported anisotropic laminated composite plates under axial compression compared with experiments*, Engineering Structures 151 (2017) 327–336, validated the Experimental results with Analytical values.

a) The FE-Model developed according the Geometrical property (Table -1) and material property in Table -2.

b) Boundary Condition of the plate shell with one end free and the other clamped as shown in Figure -2. The external load, P = 1 N was specified at the outer edge of the plate shell. [1]

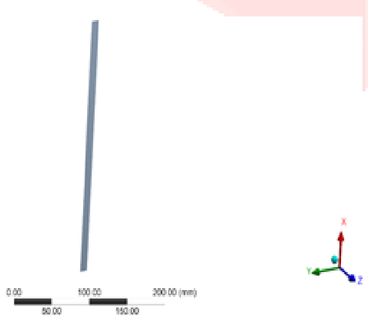
c) Meshing details

The mesh was progressively refined to ensure convergence. Number of elements: 735

Number of nodes: 888 showed in Figure 3

Table- 1 Geometric properties

	Thickness, mm	Width, mm	Length, mm
1_1	5.7997	10.4733	295
1_2	5.7277	8.145	292.5
2_1	6.3458	10.6342	294
2_2	6.1976	10.5537	294.5



FE- model of plate shell.

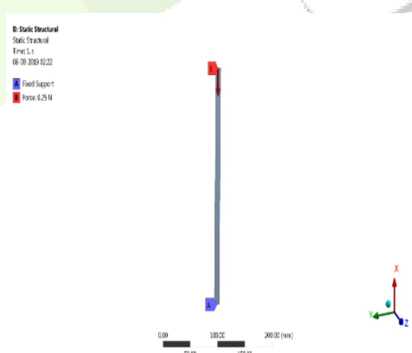
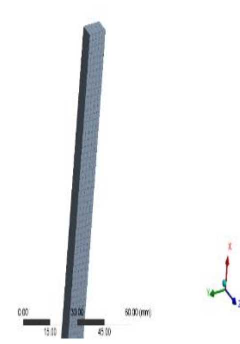


Plate shell subjected to external lode, P = 1 N



Finite element model of a plate shell.

Material details

Table2 Material properties of Composite properties of E-glass/epoxy [1]

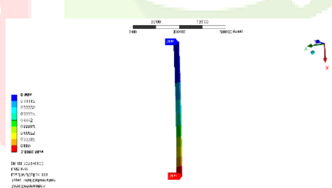
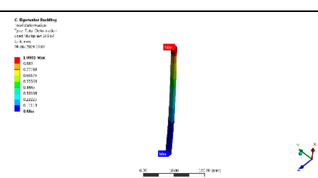
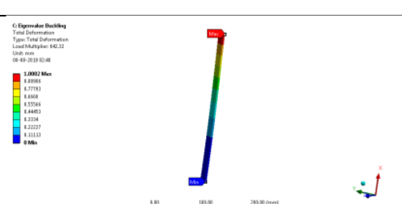
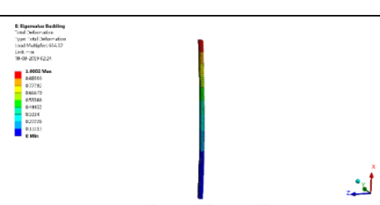
	V_f	V_m	E1, MPa	E2, E3, MPa	G12=G23=G31, MPa	v12=v23=v31
1_1	0.5344	0.4657	40546.569	11336.196	4175.05	0.322
1_2	0.5411	0.459	41005.765	11514.961	4241.67	0.3214
2_1	0.4884	0.5117	37401.468	10214.439	3757.51	0.3261
2_2	0.5	0.5	38200	10483.413	3857.55	0.325

Results & Discussion:

Table 3 Comparison of experimental results with closed form and numerical results for E-glass , epoxy composite plate.

Ply-orientation		P_{cr}Experimental, N[1]	P_{cr}Numerical, N[1]	P_{cr}Numerical, N(ANSYS)
30/_30/0/90	1_1	300	337.004	338
	1_2	242.1875	264.665	269.64
30/_30/90/0	2_1	553.125	647.82	642.32
	2_2	562.5	610.518	614.12

Table 4 Critical buckling load

Mode shapes	**Buckling Load, N**
	338
	269.64
	642.32
	614.12

From the validation of experimental results with closed form and numerical results for E glass/epoxy composite plate along with ANSYS model developed in this research work are compared and found that the results are matching with maximum 10% Error. From this we can say the developed geometrical and FE model and the material property assigned to the model are correct. So, we can use the same set of Geometrical and FE model for the further research.

REFERENCES

1. Hayder A. Rasheed, Rund Al-Masri, BacimAlali, *Closed form stability solution of simply supported anisotropic laminated composite plates under axial compression compared with experiments*, Engineering Structures 151 (2017) 327–336
2. **Andrew Kemper, Craig McNally, Eric Kennedy, Sarah Manoogian**. “The Material Properties of Human TIBIA Cortical Bone in Tension and Compression: Implication for The TIBIA INDEX” Center for Injury Biomechanics.United States Paper Number 07-0470 Kemper1.
3. **Mr. Amit Patil, Mr.V. R.Gambhire,Mr.P.J.Patil** “Experimental and Ansys analysis of stresses in real preserved intact human tibia bone under static body load to enhance the design of implant for orthopedic surgery.” International Journal of Advance Engineering and Research Development Volume 5, Issue 07, July -2018.
4. **“Analysis of mechanical properties of carbon fiber polymer Composite materials used as orthopaedic implants”.** ***International* OPEN ACCESS *Journal Of Modern Engineering Research (IJMER*|** IJMER | ISSN: 2249–6645 | www.ijmer.com | Vol. 5 | Iss. 8 | August 2015 |.
5. R.A..Latour, J. Black, "Development of FRP composite structure biomaterials:Ultimate strength of the fiber/matrix interfacial bond in in vivo simulated environments," J. Biomed. Mater .Res. , 26, 593,-606(1992).
6. M. Zimmerman, J.R. Parsons, H. Alexander, " The design and analysis of a laminated partially degradable composite bone plate for fracture fixation," J. Biomed. Mater. Res.: Appl. Biomater., 21(A3), 345-361 (1987).
7. FINITE ELEMENT MODEL OF THE HUMAN LOWER EXTREMITY SKELETON SYSTEM IN A LATERAL IMPACT.J.K. Yang, A. Wittek, and J. Kajzer Dept of lnjury Prevention, Chalmers University of Technology Gothenburg, Sweden.
8. STEVEN A. GOLDSTEIN, DOUGLAS L. WILSON, DAVID A. SONSTEGARD and LARRY S. MATTHEW .The Mechanical Properties of Human TIBIAL Trabecular Bone as a Function of Metaphyseal location. .. Biomechanm Vol 16. No I2 pp 965-969. 1983.
9. S.N.Khan Å , R. M. Warkhedkar and A.K.Shyam. “Human Bone strength Evaluation through different Mechanical Tests” International Journal of Current Engineering and Technology E-ISSN 2277 – 4106, P-ISSN 2347 – 51.