International Journal of Modern Engineering Research (IJMER)
www.ijmer.com Vol.2, Issue.4, July-Aug. 2012 pp-1576-1587 ISSN: 2249-6645

Finite Element Analysis of Thin Walled-Shell Structures by ANSYS and LS-DYNA

T. Subramani[1] Athulya Sugathan[2]

[1]*Professor & Dean, Department of Civil Engineering, VMKV Engg. College, Vinayaka Missions University, Salem, India*
[2]*PG Student of Structural Engineering, Department of Civil Engineering, VMKV Engg. College, Vinayaka Missions University, Salem, India*

ABSTRACT: *Buckling is a critical phenomenon in structural failure. Buckling is the failure of structures under compression load. Also buckling strength of structures depends on many parameters like supports, linear materials, composite or nonlinear material etc. Also buckling behavior is influenced by thermal loads and imperfections. Analyzing all these conditions is difficult task. So few parameters are considered for the present work. Due to the advances in the Finite element techniques, analysis of these problems is possible which is difficult in earlier days. Formulae's are available based on experimental techniques for linear range and not available for nonlinear range. So these problems are solved by the advances in computer technology with Finite element techniques in the nonlinear domain. In the present work both cylindrical and elliptical members are considered for buckling strength. Initially both the members are created using Ansys top down approach. Scaling options are used to built the elliptical members. The structure is divided to ease map meshing. Initially one end constrained and other free condition is considered for analysis. The structure is fine meshed to get better results. Both Shell63 in elastic range and shell43 in plastic range are used for analysis. The elliptical results are compared with theoretical results to check Finite element validity. The results are very close and analysis is extended for circular members. The stresses and loads are very high with linear analysis. But the stresses and loads are considerably reduced with nonlinear analysis. The effect of thickness on buckling load and stresses are plotted. The buckling load is increasing with increase in thickness. The hinged boundary conditions shows higher buckling strength compared to the initial boundary conditions for both elliptical and cylindrical members. The problem executed in the time domain also indicates the stresses reaching to the yield point and converging towards the critical loads. All results are represented with necessary graphical and pictorial plots*

KEY WORDS: *Buckling, ANSYS, LS-DYNA, Linear and Non – Linear Analysis*

I. INTRODUCTION

When a structure (subjected usually to compression) undergoes visibly large displacements transverse to the load then it is said to buckle. Buckling may be demonstrated by pressing the opposite edges of a flat sheet of cardboard towards one another. For small loads the process is elastic since buckling displacements disappear when the load is removed. Local buckling of plates or shells is indicated by the growth of bulges, waves or ripples, and is commonly encountered in the component plates of thin structural members.

Buckling proceeds either in stable or unstable equilibrium state.

Stable: in which case displacements increase in a controlled fashion as loads are increased, ie, the structure's ability to sustain loads is maintained.

Unstable: in which case deformations increase instantaneously, the load carrying capacity nose- dives and the structure collapses catastrophically

Neutral equilibrium: is also a theoretical possibility during buckling - this is characterized by deformation increase without change in load. Buckling and bending are similar in that they both involve bending moments. In bending these moments are substantially independent of the resulting deflections, whereas in buckling the moments and deflections are mutually inter-dependent - so moments, deflections and stresses are not proportional to loads

If buckling deflections become too large then the structure fails - this is a geometric consideration, completely separated from any material strength consideration. If a component or part there for is prone to buckling then its design must satisfy both strength and buckling safety constraints - that is why Buckling is important.

1.1 TYPES OF BUCKLING OF STRUCTURES

1.1.1 Buckling of thin-walled structures

A thin-walled structure is made from a material whose thickness is much less than other structural dimensions. Into this category fall plate assemblies, common hot- and cold-formed structural sections, tubes and cylinders, and many bridge and aeroplane structures.

1.1.2 Plate and thin shell buckling

Local buckling of an edge-supported thin plate does not necessarily lead to total collapse as in the case of columns, since plates can generally withstand loads greater than critical. However the P-q curve illustrates plates' greatly reduced stiffness after buckling, so plates cannot be used in the post-buckling region unless the behavior in that region is known with confidence. It should be emphasized that the knee in the P-q curve is unrelated to any elastic- plastic yield transition; the systems being discussed are totally elastic. The knee is an effect of overall geometric rather than material instability

1.1.3 Torsional Buckling

Torsional buckling of columns can arise when a section under compression is very weak in torsion, and leads to the column rotating about the force axis. (Figure.1)

1.1.4 Flexural Torsional Buckling

More commonly, where the section does not possess two axes of symmetry as in the case of an angle section, this rotation is accompanied by bending and is known as flexural torsional buckling.

1.1.5 Lateral Buckling

Lateral buckling of beams is possible when a beam is stiff in the bending plane but weak in the transverse plane and in torsion, as is the I-beam of the sketch. It often happens that a system is prone to buckling in various modes. These usually interact to reduce the load capacity of the system compared to that under the buckling modes individually. An example of mode interaction is the thin box section which develops local buckles at an early stage of loading, as shown greatly exaggerated here. The behavior of the column is influenced by these local buckles, and gross column buckle will occur at a load much less than the ideal Euler load

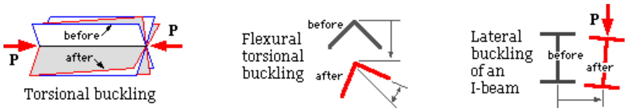


FIGURE.1 TORSIONAL BUCKLING

Stability of Equilibrium:

Buckling properties of a system are reflected in the shape of the equilibrium path, ie. the load- deflection curve. If a system is in equilibrium then its total potential energy, U, has a turning value since an infinitesimal disturbance, d□ , of the system from its equilibrium position does not change the potential energy (PE) - □ is any convenient characterizing displacement. Expressed mathematically, the equilibrium path is thus defined by:

U' ≡ dU/d□ = 0

The type of equilibrium which exists at any point on the path - stable, neutral or unstable - is also an important consideration; it may be deduced by taking the second derivative :-

U" = d2U/d□ 2 > 0 ; PE - minimum stable equilibrium
= 0 ; PE - zero slope neutral equilibrium

II. Finite Element Methods

2.1 INTROUDUCTION

Today, finite element method enjoys a position of predominance among the computational methods to occur in this century, within only a few decades this technique has evolved from one with initial application in analysis of aircraft structure as early as 1941 and in structural engineering to a widely utilized and richly varied computational approach for many scientific and technological areas. The stress analysis in the field of structural mechanics is invariable complex and for many of the engineering problems; it is extremely difficult and tedious to obtain analytical solutions. In this situation, most of the practical problems are solved by numerical methods, which provide approximate but acceptable solutions. With the advent of computers, one of the most powerful techniques that has emerged from the realm of engineering analysis is the finite element method and the method being general, can be used for the analysis of structures are solid of complex shapes and complicated boundary conditions.

The basic concept of finite element method is discritization of a structure into finite number of elements, connected at finite number of points called nodes. The material properties and the governing relationships are considered over these elements and expressed in terms of nodal displacement at nodes. An assembly process duly considering the loading and constraints results in a set of equations governing the structural response, which are established through the application of appropriate variation principle. Solutions of these equations give the response of the structure. Selecting proper elements and subdividing the structure with large number of finite elements or by taking higher order elements can increase the accuracy of solution obtained by finite element method. In modern design practice, with the advent of large and fast modern digital computers and advancement in numerical techniques; solutions to various static and dynamic problems has become fast and efficient.

2.2 TYPES OF ELEMENTS

Few Important FEM elements are as follows.

TRUSS: Slender element (length>>area) which supports only tension or compression along its length; essentially a 1D spring.

1. BEAM: Slender element whose length is much greater that its transverse dimension which supports lateral loads, which cause flexural bending.
2. TORSION: Same as truss but supports torsion.
3. 2D SOLID: Element whose geometry definition lies in a plane and applied loads also lie in the same plane. Plane stress occurs for structures with small thickness compared with its in plane dimension - stress components associated with the out of plane coordinate are zero. Plane strain occurs for structures where the thickness becomes large compared to its in plane dimension - strain component associated with the out of plane coordinate are zero.
4. PLATES: Element whose geometry lies in the plane with loads acting out of the plane which cause flexural bending and with both in plane dimensions large in comparison to its thickness - two dimensional state of stress exists similar to plane stress except that there is a variation of tension to compression through the thickness.
5. SHELLS: Element similar in character to a plate but typically used on curved surface and supports both in

plane and out of planeloads – numerous formulations exist.

6. 3D SOLID: Element classification that covers all elements - element obeys the strain displacement and stress strain relationships.

2.3 MERITS OF FINITE ELEMENT METHOD

The systematic generality of finite element procedure makes it a powerful and versatile tool for a wide range of problems. Thus, flexible, general-purpose computer programs can be developed and can be applied to various problems with little or no modification. FEM can be easily interpreted in physical terms. As well it has a strong mathematical base. Hence, finite element method can be easily applied to any problem with a proper knowledge of the physical system under consideration and can be solved to a greater accuracy by the application of proper mathematical tool. Non-homogenous continuum can also be dealt with by merely assigning different properties to different elements. It is even possible to vary the properties within an element according to the polynomial applied. Finite element method accommodates complex geometry with ease and is capable of handling non-linear and time dependent system also. In finite element method, since boundary conditions are introduced in the assembled equations, it requires only to specify the geometric boundary conditions without regarding its effects on interior elements. Since the boundary conditions do not enter into the individual finite element equations, the field variable models need not be changed, when the boundary conditions change. Finite element method considers the multidimensional continuity of body. Hence it does not require separate interpolation process to extend the approximate solution to every point with in the continuum. It does not also require the trial solutions that must all apply to the entire multidimensional continuum.

2.4 DEMERITS OF FINITE ELEMENT METHOD

The solution obtained from FEM can be realistic if and only if the material properties are known precisely.

The major drawback of FEM is sensitivity of the solution on the geometry of the element such as type, size, number, shape and orientation of elements used.

The computer programs of FEM require relatively a large computer memory and time.

FEM Programs yield a large amount of data as results. It is very difficult to separate out the require results from the pile of numbers.

III. ANSYS - INTRODUCTION

ANSYS is a commercial FEM package having the capabilities ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. It is available in modules. Each module is applicable to specific problem. For example, Ansys/Civil is applicable to Civil structural analysis. Similarly Ansys/Flotran is CFD software applicable to Fluid Flow. The advantage of Ansys compared to other competitive software's is , its availability as bundled software of pre, post and a Processor. Typical Ansys program includes 3 stages.

- Pre-Processing
- Solution
- Post-Processing

3.1 MODELING

This is the important step of creating the physical object in the system. They are two types of modeling in Ansys.
Direct Modeling & Solid Modeling

1. DIRECT MODELING: In this approach the physical structure is represented by nodes and elements directly. The problem is solved once after the boundary conditions are applied. This approach is simple and straight forward. Takes very little time computation. But this can be applied only for simple problems. When problem becomes complex, this method becomes tedious to apply.

2. SOLID MODELING: Models are directly created either using Ansys Preprocessor or imported from popular CAD soft wares like Mechanical Desktop, Pro/E, CATIA, SOLID WORKS etc. Once the structural model is created, by using mesh tool, the model can be meshed and problem can be solved by applying the boundary conditions. In Ansys Solid modeling is carried out using two methods. They are

1. Bottom Up Approach: To create model, Entities are required. Keypoints, Lines, Areas, Volumes are the entities in Ansys. If model is constructed through Keypoints to Lines , From Lines to Areas, and From Areas to Volumes the approach of modeling is called Bottom Up Approach. This approach is useful when models are complex.

2. Top Down Approach: A 3D Model can be created directly using the Volumes. Once Volumes are created, all the entities below the volumes (areas, lines, keypoints) are automatically created. This approach is easy but can be applied to simple problems.

3.2 TYPES OF BUCKLING ANALYSIS:

Ansys supports two types buckling analysis

- Non-Linear Buckling
- Eigen Value Buckling analysis

3.2.1 .Non-Linear Buckling

Nonlinear buckling analysis is usually the more accurate approach and is recommended for design or evaluation of actual structures. This technique employs a nonlinear static analysis with gradually increasing loads to seek the load level at which the structure becomes unstable. Using the nonlinear technique, model can include features such as initial imperfections, plastic behavior, gaps, and large-deflection response. In addition, using deflection-controlled loading, we can even track the post-buckled performance of the structure (which can be useful in cases where the structure buckles into a stable configuration, such as "snap-through" buckling of a shallow dome).

3.2.2. Eigen value buckling

Eigen value buckling analysis predicts the theoretical buckling strength (the bifurcation point) of an ideal linear elastic structure. This method corresponds to the textbook approach to elastic buckling analysis: for instance, an eigen value buckling analysis of a column will match the classical Euler solution. However, imperfections and nonlinearities prevent most real-world structures from achieving their theoretical elastic buckling strength. Thus, eigenvalue buckling analysis often yields un conservative results, and should generally not be used in actual day-to-day engineering analyses.

Procedure for Eigenvalue Buckling Analysis

Eigenvalue buckling analysis generally yields un conservative results, and should usually not be used for design of actual structures. The procedure of eigenvalue buckling analysis is as follows.

- Building the model.
- Obtaining the static solution.
- Obtaining the eigenvalue buckling solution.
- Expanding the solution.
- Reviewing the results.

1. Building the model

The Model is built is either through Bottom up Approach or Top down Approach and should be meshed with appropriate elements. Proper material and geometric properties (Real properties) should be supplied. Finally Boundary conditions should be supplied.

2. Obtaining the Solution

The following should be followed to obtain proper solution Prestress effects [PSTRES] must be activated. Eigenvalue buckling analysis requires the stress stiffness matrix to be calculated.

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. Therefore, if a unit load is specified, the load factors represent the buckling loads. All loads are scaled. Eigenvalues represent scaling factors for all loads. If certain loads are constant (for example, self-weight gravity loads) while other loads are variable (for example, externally applied loads), we need to ensure that the stress stiffness matrix from the constant loads is not factored by the eigenvalue solution. One strategy that we can use to achieve this end is to iterate on the eigen solution, adjusting the variable loads until the eigenvalue becomes 1.0 (or nearly 1.0, within some convergence tolerance). Design optimization could be useful in driving this iterative procedure to a final answer.

3. Obtaining the Eigen value buckling Solution:

After executing the program for static solution, again solution should be changed to Eigen Buckling and extraction technique should be specified.

4. Expanding the Solution:

Solution should be expanded to obtain the critical buckling loads.

5. Reviewing the results:

Results can be reviewed through /post1. Through result summary critical buckling in the form of natural frequencies can be viewed. By using the read set option for different critical loads, deflection and stress patterns can be viewed.

IV. PROBLEM DEFINTIION AND FINITE

4.1 ELEMENT MODEL DEVELOPEMENT

Definition:

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. The analysis will be carried out using both implicit and explicit techniques.

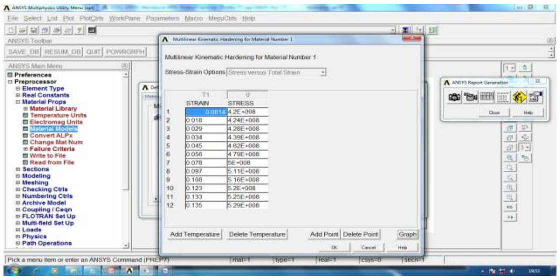
- Analysis of elastic buckling nature of the cylindrical and elliptical vessels under compression loads
- Effect of thickness on stress generation in the structure
- Effect of thickness on buckling strength
- Effect of diameter on buckling strength and stress generation.
- Effect of non linearity in the buckling strength of structures.

Problem dimensions (initial)

- Outer diameter D=400mm
- Initial thickness t=16mm
- Height of the cylinder=1200mm
- Initial minor diameter of ellipse=200mm

Material Specification:

- Material : Structural Steel St42
- Young's Modulus E=200Gpa
- Poison's ratio ν=0.3
- Density ρ=7800kg/m3.



Multilinear Kinematic Hardening for Material Number 1

Multilinear Kinematic Hardening for Material Number 1

Stress-Strain Options Stress versus Total Strain

	T1	0
	STRAIN	STRESS
1	0.0014	4.2E+008
2	0.018	4.24E+008
3	0.029	4.28E+008
4	0.034	4.39E+008
5	0.045	4.62E+008
6	0.056	4.79E+008
7	0.078	5E+008
8	0.097	5.11E+008
9	0.108	5.16E+008
10	0.123	5.2E+008
11	0.133	5.25E+008
12	0.135	5.29E+008

Pick a menu item or enter an ANSYS Command (PREP7) mat=1 type=1 real=1 csys=0 secn=1

Figure.2 Nonlinear Properties of the Material

The Figure.2 shows applied nonlinear material properties of the problem. Most of the analysis is generally done in elastic range. But due to the advances in technology, load carrying capacity of the structures can be studied in the plastic range also. Also it helps in finding factor of safety in the structures. These nonlinear properties are obtained by experimental testing and recording.

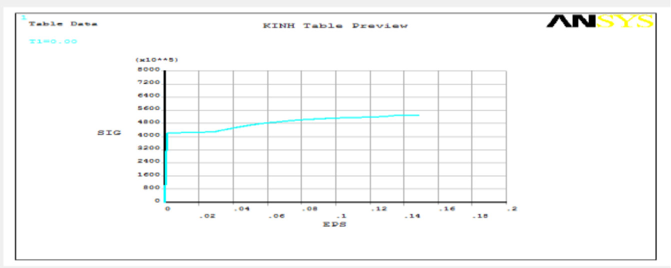


Figure.3 Stress – Strain Graph For The Material

The Figure.3 shows material graph for the problem. Initially the graph is straight and later sloping down indicating yielding of the structure. Generally above the yield by plastic modulus can be calculated using slope values or stress strain data can be supplied directly to find the nonlinear behavior of the structures.

Boundary conditions:

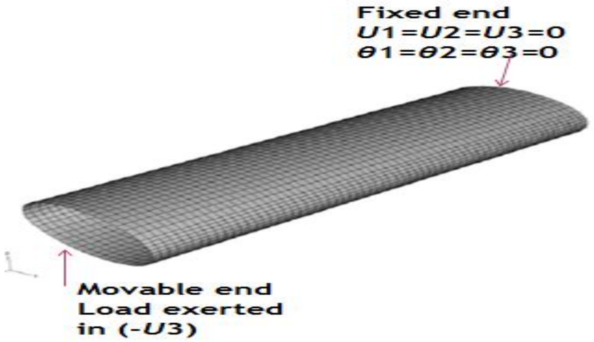


Figure.4 Boundary Conditions

For each of the two ends, two different types of boundary conditions(Figure.4) were used.

At the fixed end, displacement degrees of freedom in 1, 2, 3 directions (U1, U2, U3) as well as rotational degrees of freedom in 1, 2, 3 directions (θ1, θ2, θ3) were restrained to be zero. At the movable end, load was exerted with an even stress distribution in the longitudinal direction U3.

Mesh Size:

- No. of element s: 12000
- No. of nodes: 12120
- Size of element : 3.5m

Methodology:

- Analysis is carried out for EHS(Elliptical Hallow Sections) for minor diameter equal to half the major diameter and cylindrical geometries(CHS) in both elastic and nonlinear range to find the buckling strength.
- Initial thickness=16mm
- Iterations are carried out to find the buckling stress variation for different thickness to compare with theoretical values.

Element Type:

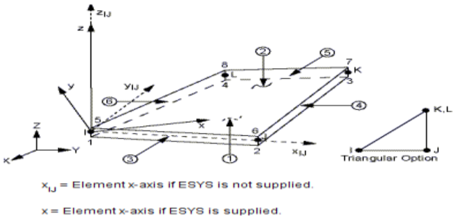


Figure.5 Shell63

SHELL63 (Figure.5) has both bending and membrane capabilities. Both in-plane and normal loads are permitted. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. Stress stiffening and large deflection capabilities are included. A consistent tangent stiffness matrix option is available for use in large deflection (finite rotation) analyses.

4.2 ASSUMPTIONS

- The Member is initially perfectly straight and is axially loaded.
- The material is elasto-plastic with strain hardening materials.
- The self weight of the structure is not considered
- The member will fail by buckling alone.
- Imperfections of members are not considered.

V. RESULTS & DISCUSSION

Initially the linear elastic buckling is carried out for the structures and later nonlinear buckling analysis is carried out using elastic buckling loads to find actual buckling capacity of the structures. Comparative study is carried out between buckling, strength , buckling stress of cylindrical and elliptical members for buckling strength and stress generation. The formulae's are based on elastic material assumption. But this assumption is not true in the nonlinear conditions. So for nonlinear conditions, analysis has to be carried out after getting initial estimation of loads from linear elastic analysis. This linear elastic analysis helps in initial estimates of load carrying capacity and generally to make a comparative study of the material strength. In the beginning analysis is carried out for elliptical members under elastic material behavior. The formulae's for theoretical checking and analysis results are as follows.

Theoretical formulae for Elastic buckling stress for Cylindrical hollow members:

$$\sigma = \frac{2Et}{D\sqrt{3(1-U^2)}}$$

Where

Where E= Young's modulus
t=Thickness of vessel
D= Diameter of the cylindrical member

For Elliptical members buckling stress formulae is

$$\sigma = \frac{Et}{\frac{A^2}{B}\sqrt{3(1-\upsilon^2)}}$$

Where A= Major radius
B=Minor radius
ν=Poison's ratio

Results for Elastic buckling Stress estimates(EHS):
Initially analysis has been carried out for buckling process and results are represented as follows.

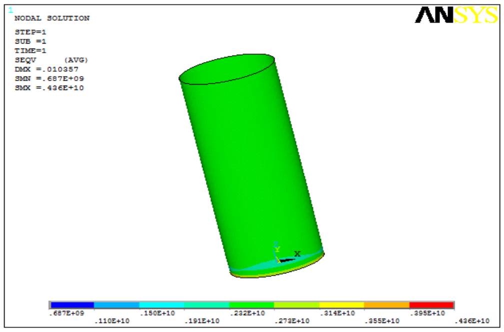


Figure.6: Buckling Stress For Elastic Buckling (Results For T=16mm D1/D2=2)

The Figure.6 shows a buckling stress of around 4360MPa due to elastic buckling. Since material property is assumed as linear, the stress levels are crossing the yield limits. But if material is considered as nonlinear, then the correct stresses can be estimated. This is done in the next stage of analysis. Maximum stresses are observed at the base of the structure.

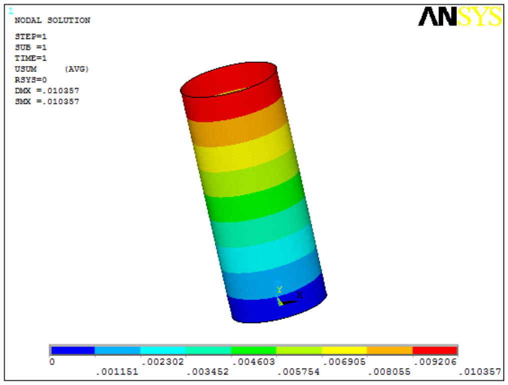


Figure.7 Deformation Plot For Elastic Buckling (Results For T=16mm D1/D2=2)

The Figures.7 shows maximum deformation of 0.010367mm for the given buckling load. Maximum deformation can be observed at the free end and minimum displacement can be observed at the constraint end. The status bar indicates varying displacements across the problem.

Comparative results for EHS
Table.1 Shows The Comparative Results Of Buckling Stresses And Buckling Loads Ehs Members

Thickness (in mm)	Buckling Stress(Theoretical) (in Gpa)	Buckling stress (FEM) (in GPa)	Percentage error (in %)	Buckling Load(FEM) (in N)
16	4.68	4.36	6.82	0.40200E+08
13.3	3.94	4.16	5.45	0.32558E+08
11.42	3.46	3.66	5.78	0.25042E+08
10	3.02	3.16	4.63	0.19233E+08
8.88	2.69	2.78	3.34	0.15239E+08
8	2.2	2.28	3.63	0.12446E+08

Table.1 Comparative Results of Buckling Stresses And Buckling Loads Ehs Members

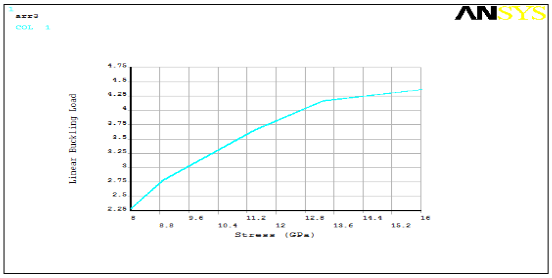


Figure.8 Stress Vs Thickness

The graph (Figure.9) represents stress and thickness relation during buckling process. For smaller thickness stress raise is high, but later the slope is reducing with increase in thickness. So the relation is not linear with buckling estimations.

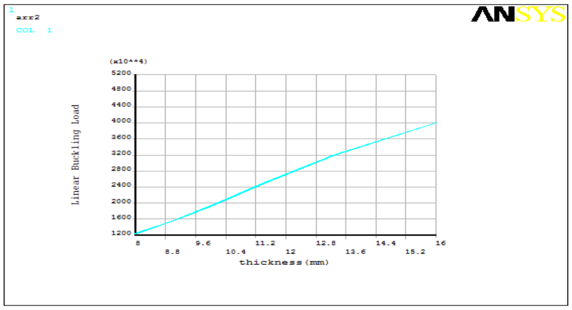


Figure.9 Load Vs Thickness

The graph (Figure.9.) represents Buckling Load and thickness relation during buckling process. Here almost linear relation can be observed for load and thickness due to elastic material assumption. From the graph it can be estimated for possible buckling load for higher thickness.

With nonlinear properties (EHS):

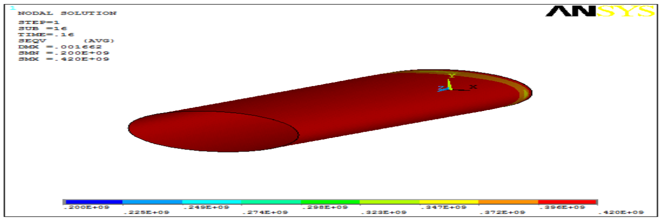


Figure.10 Buckling Stress For Nonlinear Buckling (Results For T=16mm D1/D2=2)

The Figure.10 shows a buckling stress of around 420MPa due to nonlinear buckling. The stresses are practical compared to elastic buckling where stresses becomes infinite and nonreliable. But for comparative studies, the analysis will be done using elastic conditions. Generally Young's modulus is defined in the elastic region and later either plastic modulus or stress strain data will be given after yield point. This stress strain data is obtained through experimental testing for the material.

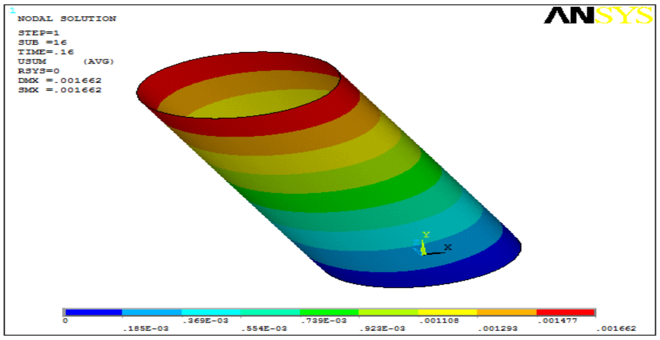


Figure.11 Deformation Plot For Nonlinear Buckling (Results for T=16mm D1/D2=2)

The Figure.11 shows maximum deformation of 0.001662mm for the given buckling load. Maximum deformation can be observed at the free end and minimum displacement can be observed at the constraint end. The status bar indicates varying displacements across the problem.

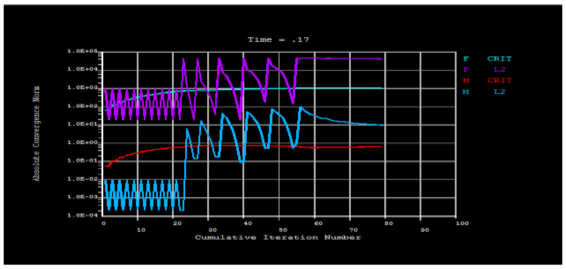
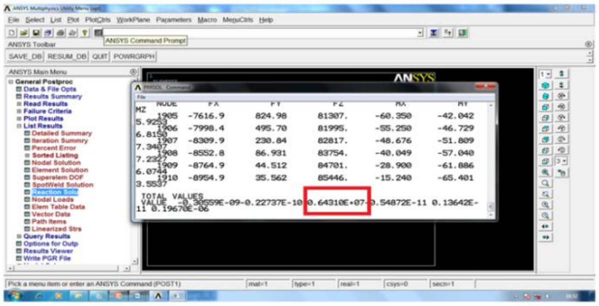


Figure.12 Nonlinear Execution

Whenever material is given nonlinear properties, convergence graphs can be observed. Mainly nonlinear problems defends on indirect or iterative solvers. This iterative solvers splits the problem into number of stages and in each stage they try to get convergence between applied load and reaction loads. These techniques consumes maximum resources of the computer and requires almost 100times more then the elastic buckling problems. Generally Ansys checks with load and reactions along with mathematical convergence for matrix using either L2norm or absolute norm. In the problem both force and moment convergence is considered.

The Figure.12 shows deformed and un-deformed shape of the problem. Maximum deformation can be observed at the loading region. White colour in the plot represents original configuration and blue region indicates deformed region.



```
    NODE       FX          FY          FZ          MX          MY
MZ
    1905   -7616.9     824.98      81307.      -60.350     -42.042
5.9253
    1906   -7998.4     495.70      81995.      -55.250     -46.729
6.8150
    1907   -8309.9     230.84      82817.      -48.676     -51.809
7.3407
    1908   -8552.8     86.931      83754.      -40.049     -57.040
7.2327
    1909   -8764.9     44.512      84701.      -28.900     -61.886
6.0744
    1910   -8954.9     35.562      85446.      -15.240     -65.401
3.5537

 TOTAL VALUES
 VALUE   -0.30559E-09-0.22737E-10 0.64310E+07-0.54872E-11 0.13642E-
11 0.19670E-06
```

Figure13 Reaction Load Estimation For Buckling

The Figure.13 shows reaction loads estimated from nonlinear buckling. Maximum of 6431000N can be observed for the buckling of the system. This value is taken from final reaction value after problem is converged for buckling solution. This nonlinear buckling load is much smaller then the calculated elastic buckling load. So nonlinear analysis gives much better results compared to the elastic buckling analysis.

Table.2 Comparative Table For Linear And Nonlinear Buckling Values

Thickness (mm)	Nonlinear Buckling load (N) - B	Linear Buckling load (N) - A	Ratio(B/A)
16	0.64317E+07	0.40200E+08	.159
13.33	0.52091E+07	0.32558E+08	.159
11.42	0.45074 E+07	0.25042E+08	.179
10	0.40385 E+07	0.19233E+08	.208
8.88	0.35044 E+07	0.15239E+08	.229
8	0.32355 E+07	0.12446E+08	0.255

The Table: 2 shows comparative buckling loads for both elastic and nonlinear problems. The results almost nonlinear loads are almost 15% of elastic loads. Also this proportion is reducing with reduction in thickness. So a nonlinear study helps in finding the actual loading capacity of the EHS columns.

VI. Analysis Results For Cylindrical Members(Chs):

Analysis has been carried out for buckling process and results are represented as follows.

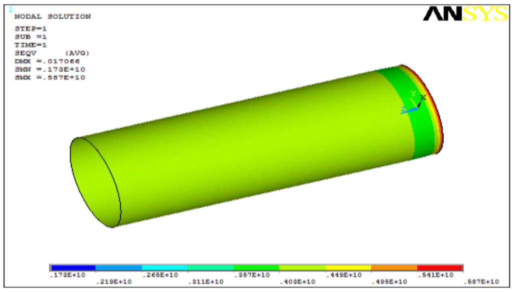


Figure.14 Buckling Stress For Elastic Buckling (Results For T=16mm D1/D2=1)

The Figure:14 shows a buckling stress of around 5870MPa due to elastic buckling. Since material property is assumed as linear, the stress levels are crossing the yield limits. But if material is considered as nonlinear, then the correct stresses can be estimated. Further analysis will be carried out in elastic range. Maximum stresses are observed at the base of the structure.

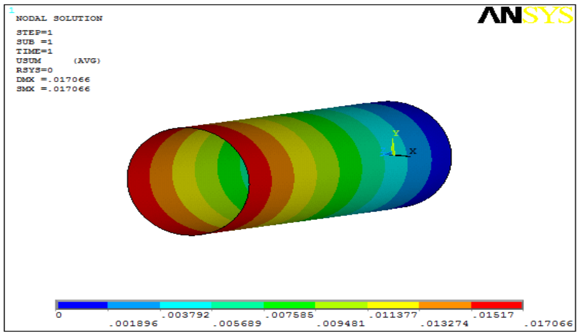


Figure.15 Deformation Plot For Elastic Buckling (Results For T=16mm D1/D2=1)

The Figure.15 shows maximum deformation of 0.017mm for the given buckling load. Maximum deformation can be observed at the free end and minimum displacement can be observed at the constraint end. This value of deformation is higher then the elliptical structure.

Comparative results for CHS

Table.3 shows the comparative results of Buckling stresses and Buckling loads of CHS members

Thickness(mm)	Buckling Stress	Buckling Load
16	5860	0.65899E8
13.3	4880	0.46043E8
11.42	4170	0.34229E8
10	3670	0.26616E8
8	2980	0.17476E8

Table: 3 Comparative Results Of Buckling Stresses And Buckling Loads Of Chs Members

With nonlinear properties (CHS):

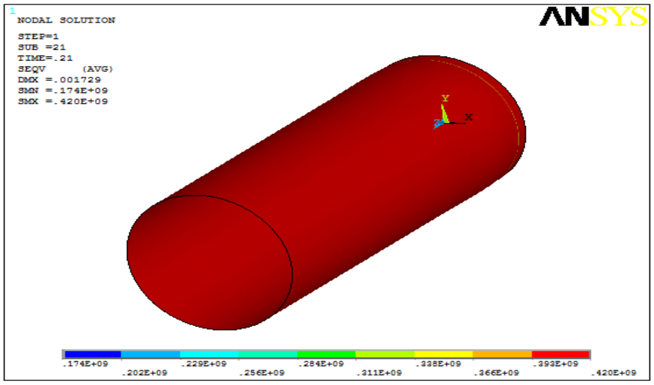


Figure.16 Buckling Stress for Nonlinear Buckling (Results for T=16mm D1/D2=1)

The Figure.16 shows a buckling stress of around 420MPa due to nonlinear buckling. The stresses are practical compared to elastic buckling where stresses become infinite and nonreliable. This stress is similar for the elliptical sections as the stress levels reaching to the yield point of the problem.

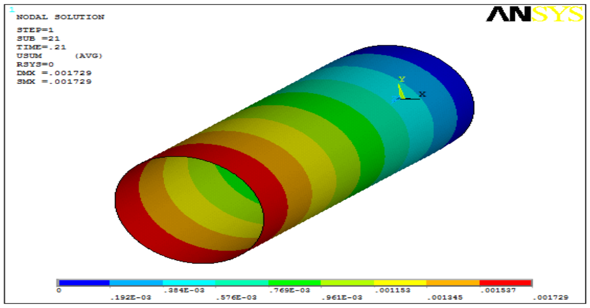


Figure.17; Deformation Plot for Nonlinear Buckling (Results for T=16mm D1/D2=1)

The Figure.17 shows maximum deformation of 0.001728mm for the given buckling load. Maximum deformation can be observed at the free end and minimum displacement can be observed at the constraint end. The status bar indicates varying displacements across the problem.

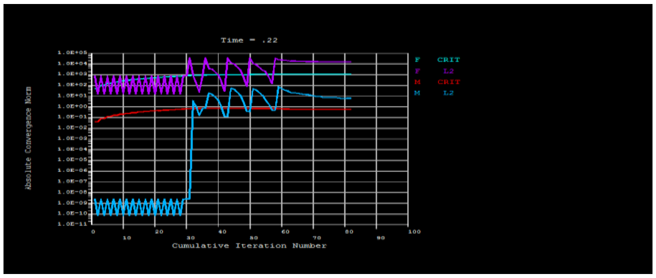
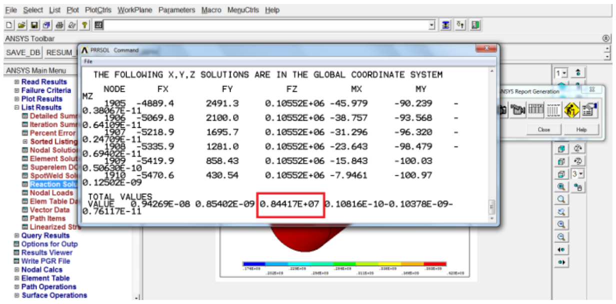


Figure.18: Nonlinear Execution

Convergence process can be observed in the graph. Sudden change of convergence indicates ill conditioned matrix and execution will stop indicating failure of the analysis for the given load or unstable configuration of the problem.(Figure.18)



```
 THE FOLLOWING X,Y,Z SOLUTIONS ARE IN THE GLOBAL COORDINATE SYSTEM
   NODE      FX          FY          FZ          MX          MY
MZ
   1905   -4889.4      2491.3     0.10552E+06 -45.979      -90.239    -
0.38067E-11
   1906   -5069.8      2100.0     0.10552E+06 -38.757      -93.568    -
0.64109E-11
   1907   -5218.9      1695.7     0.10552E+06 -31.296      -96.320    -
0.24709E-11
   1908   -5335.9      1281.0     0.10552E+06 -23.643      -98.479    -
0.69402E-11
   1909   -5419.9      858.43     0.10552E+06 -15.843      -100.03
0.50630E-10
   1910   -5470.6      430.54     0.10552E+06 -7.9461      -100.97
0.12502E-09

 TOTAL VALUES
 VALUE    0.94269E-08 0.85402E-09 0.84417E+07 0.10816E-10-0.10378E-09-
0.76117E-11
```

Figure.19 Reaction Load Estimation For Buckling

The Figure.19 shows reaction loads estimated from nonlinear buckling. Maximum of 8441600N can be observed for the buckling of the system. This value is taken from final reaction value after problem is converged for buckling solution. This nonlinear buckling load is much smaller then the calculated elastic buckling load.

Thickness (mm)	Nonlinear Buckling load (N) - B	Linear Buckling load (N) - A	Ratio(B/A)
16	0. 8445E+07	0.65899E+08	.128
13.33	0.6836E+07	0.46043E+08	.148
11.42	0.6009 E+07	0.34229E+08	.176
10	0.51923E+07	0.26616E+08	.195
8	0.4106 E+07	0.17476E+08	0.234

Table.4 Comparative Table For Linear And Nonlinear Buckling Values

The Table.4 shows comparative buckling loads for both elastic and nonlinear problems. The results almost nonlinear loads are almost 15% of elastic loads. Also this proportion is reducing with reduction in thickness. So a nonlinear study helps in finding the actual loading capacity of the EHS columns.

VII. Linear Buckling Analysis With Hinged Boundary Conditions

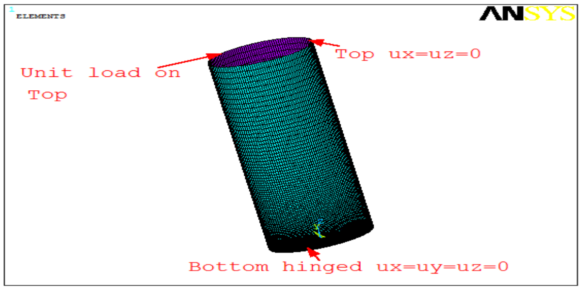


Figure.20 Boundary Conditions For Hinged Problem

The Figure.20 shows hinged boundary conditions for the shell problem. All bottom nodes are hinged and the top nodes are constrained in all translational directions except the unit loading direction. A total of 12000 elements with 12120 nodes are used for representation of the problem.

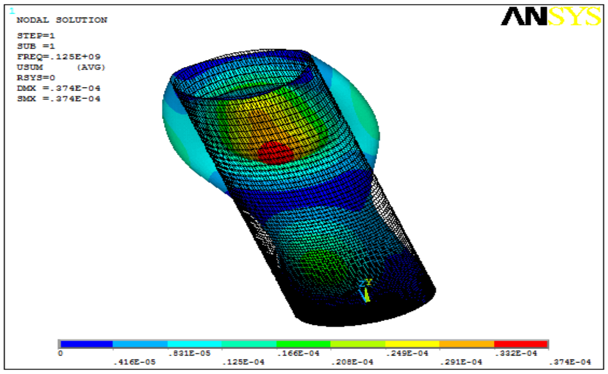


Figure.21 Buckled Mode For 1 St Buckling Load

The Figure.21 shows buckled mode shape of the problem. The buckling load value is around 0.125E9 N. the analysis is carried out for elastic assumption.

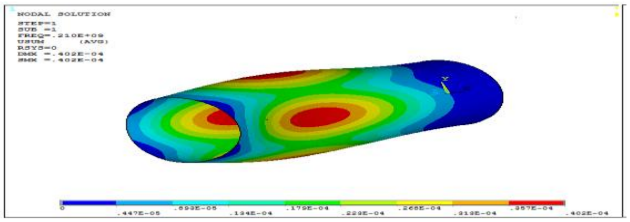


Figure.22 Mode Shape or Buckled Shape for Cylindrical Member

The Figure.22 shows buckled mode shape of the cylindrical shell problem. The buckling load value is around 0.210E9 N. the analysis is carried out for elastic assumption and buckling load is almost double to the elliptical shell.

VIII. ANSYS -LS-DYNA SIMULATION:

The ansys model is solved using Ansys-LS-Dyna solver for the buckling load for 16mm thickness. The system shows almost stress generation equal to 420Mpa which is yielding point of the structure. So Even LS-Dyna solver also shows (Figure.23) yielding of the structure for given nonlinear buckling load.

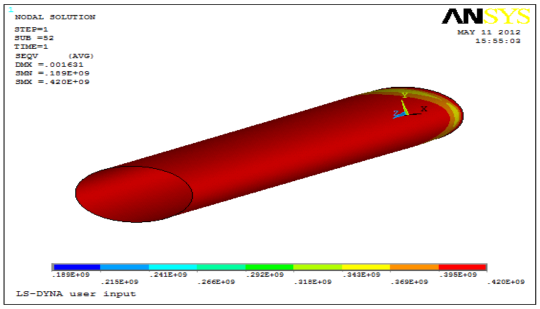


Figure.23 Von-Mises Stress Through LS-DYNA Solver

Maximum stress of 420Mpa can be observed with the nonlinear properties. For the given load for the stress strain data, stress increase in minimum but strain increase will exist in the problem. So it is better to apply strain based theories in the nonlinear buckling regions. No explicit algorithms are directly available to check the cylindrical structure buckling. But can be checked for given loads. Generally if the stress goes beyond allowable or yield stresses, then it can be taken as failure.

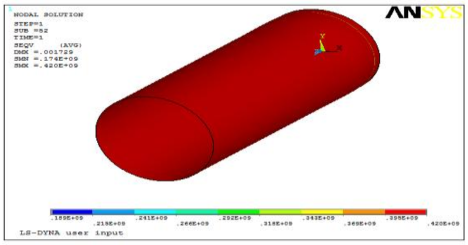


Figure.24 Stress Analysis through Ansys-Ls Dyna Solver

Using Ansys/Ls-Dyna solver the problem is executed. This is almost like Ansys pictures only as the complete post processing (Figure.24) is done through ansys. Only solver interface will change. Only advantage of Dyna solver is no convergence problems faced with Ansys iterative sovles. The problem is done in time history processing for time equal to 1sec which creates equivalent static load for buckling by providing 'G' or gravitational loads.

IX. DISCUSSION

Buckling is main problem with many reactor vessels, pipe lines, storage vessels etc. This buckling may be due to compressive loads created due to self weight or outer members or internal pressure or may imperfections in the system. But structural failure creates huge loss to the inventory of company and also life of the system. So proper check need to be done for buckling of the components. In the present work, cylindrical and elliptical geometries are analyzed for buckling loads. Initially the geometry is built using Ansys preprocessor and later meshed with 4 nodded shell with elastic and inelastic properties. Initially the analysis is done in the linear elastic domain. The results shows improperly higher stresses and buckling loads. This is due to application of linearity in the problem. Later the same members are analyzed actual material properties in the working conditions. The properties represents reality results in the practical conditions. But these may the system to consume more resources like memory and solution time. The loads obtained is almost less then the 15% of elastic buckling loads. The stresses are limited to the yield stress in the structure. The results shows linearity of buckling load for variation in thickness. But stresses are not following linear path. Similarly for nonlinear buckling analysis, along with thickness buckling is increasing but the stresses are almost limiting the yield stresses. The nonlinear analysis is carried out using Ansys-nonlinear-inelastic with multi-linear properties. Hinged boundary condition provides higher buckling load compared to the one end fixed and other end hinged conditions. The problem is also done explicit solvers using time domain. Again the stresses are limiting to yield point indicating the structure has reach to its critical value. The results also shows closeness of linear and nonlinear buckling loads at lesser thickness and large different for higher thickness members.

X. CONCLUSIONS & FURTHER SCOPE

A Finite element analysis is carried out to find buckling strength of cylindrical and elliptical structures. Cylindrical and elliptical members are mainly used in storage and other industrial applications. One estimate is that almost 30% industrial products are coming under these category. The results summary is as follows.

- Initially both cylindrical and elliptical geometries are built
- The geometry is meshed with small elements towards the constraint.
- Buckling analysis is carried out in both linear and nonlinear domain
- The stresses are very high in the linear region and for nonlinear analysis the stresses are with in the practical working conditions.
- Nonlinear solution is a iterative solution and failure loads can be predicted by running the problem with more steps.
- A nonlinear material graph is considered for the nonlinear analysis.
- Shell63 element in the linear range and shell43 element is considered for plastic conditions. Shell63 is a linear element and does not support nonlinearity in the problem where as shell43 is a nonlinear element supports plasticity in the problem. But shell43 takes more time for execution.
- Initially for EHS, both theoretical and analysis values are compared to check Finite element solution with theoretical calculations. Also graphical plots are represented to find effect of thickness on stress and buckling strength estimates.
- The results shows reduction of nonlinear buckling loads compared to the elastic buckling loads. Almost nonlinear loads are only 15% of elastic loads. So nonlinear analysis is important to find actual buckling loads for the structure. Also buckling analysis is considered for hinged conditions. The results shows both elliptical and cylindrical members higher strength compared to the one end fixed and other free.

REFERENCES

[1]. Banks, W. M., and Rhodes, J., (1983) The Instability of Composite Sections, 2nd Int. Conf. On Composite Structures, Paisley, UK, Elsevier, 442-452.

[2]. Barbero, E. J., and Raftoyiannis, I. (1990) Buckling Analysis of Pultruded Composite Columns, Impact and Buckling of Structures, Ed. D. Hui, ASME AD-Vol. 20, AMD-Vol. 114, 47-52.

[3]. Barbero, E. J. and J. Tomblin, (1994) A Phenomenological Design Equation for FRP Columns with Interaction Between Local and Global Buckling, Thin-Walled Structures, 18, 117-131.

[4]. Bank, L. C., and Yin, J., (1996) Buckling of Orthotropic Plates with Free and Rotationally Restrained Unloaded Edges, Thin-Walled Structures, 24 83-96.

[5]. Barbero, E. J. , L. A. Godoy, and I. Raftoyiannis, (1996) Finite elements for three-mode interaction in buckling analysis, I. J. Num. Methods Engineering, 39, 469-488.

[6]. Barbero, E. J., (1998a) Prediction of Compression Strength of Unidirectional Polymer Matrix Composites, J. Composite Materials, 32(5), 483-502.

[7]. Barbero, E. J., (1998b) Introduction to Composite Materials Design, Taylor and Francis, Philadelphia, PA.

[8]. 8 Barbero, E. and J. Trovillion, (1998c) Prediction and Measurement of the Post-Critical Behavior of Fiber-Reinforced Composite Columns, special issue on Civil Infrastructural Applications of Composite Materials, 58(8), 1335-1441.

[9]. 09. Barbero, E. J., Makkapati, S., and Tomblin, J. S. (1999a) Experimental Determination of Compressive Strength of Pultruded Structural Shapes, Composite Science and Technology, 59, 2047-2054.

[10]. 10. Barbero, E. J., Dede, E. and Jones, S. (1999b) Experimental Verification of Buckling-Mode Interaction

in Intermediate-Length Composite Columns, I. J. Solid Structures, to appear.

[11]. 11. Barbero, E. J. and Wen, Ed A., (2000) Compressive Strength of Production Parts without Compression Testing, ASTM STP 1383 Composite Structures: Theory and Practice, ASTM, PA.

[12]. 12 Strength of Materials – S.S. Bhavikatti, Vikas Publishing House Pvt. Ltd., 2000

[13]. Finite Element Procedures – Klaus-Jurgen Bathe, Prentice Hall of India Pvt. Ltd.- Sixth Edition 2002

[14]. 13 Introduction to the Finite Element Method, Desai/Abel – CBS publishers 2002

[15]. 14 Strength of Materials – S. Ramamrutham, R Narayana, Dhanpatrai & Sons - 2002

[16]. 15 Concepts and Application of Finite Element Analysis Rober D. Cook, David S – Joh Wiely & Sons Pte. Ltd. Fourth Edition 2003

[17]. 16 Finite Elements in Engineering – Tirupathi R. Chandrupatla, Ashok D. Belegundu, Prentice- Hall of India Pvt. Ltd, 2003