A

DISSERTATION REPORT

ON

FINITE ELEMENT ANALYSIS OF SHELL STRUCTURES

Submitted in partial fulfillment of the requirements for the award of degree of

MASTER OF TECHNOLOGY

IN

STRUCTURAL ENGINEERING

Submitted by

AJAY KUMAR GAUTAM

(2014PCS5326)

Under the esteemed guidance of

Dr. VINAY AGRAWAL
Assistant Professor

Dr. RAJESH GUPTA
Associate Professor

DEPARTMENT OF CIVIL ENGINEERING

MALAVIYA NATIONAL INSTITUTE OF TECHNOLOGY JAIPUR

JUNE 2016

CERTIFICATE

This is to certify that the seminar work entitled “**FINITE ELEMENT ANALYSIS OF SHELL STRUCTURES**” which is being submitted by **AJAY KUMAR GAUTAM** (2014PCS5326) in partial fulfillment for the award of the degree of Master of Technology in Structural Engineering, MNIT, JAIPUR is a bonfire work done by him under our guidance and supervision.

Dr. VINAY AGRAWAL
Assistant Professor

Dr. RAJESH GUPTA
Associate Professor

Place:

Date:

DECLARATION

I hereby certify that the work which is being presented in the Dissertation report **THE** “**FINITE ELEMENT ANALYSIS OF SHELL STRUCTURES**”, in partial fulfillment of the requirements for the award of the Degree of Master of Technology and submitted in the Department of Civil Engineering of the Malaviya National Institute of Technology Jaipur is an authentic record of my own work carried out during a period from July 2015 to June 2016 under the supervision of my guide **Dr. VINAY AGRAWAL** Assistant Professor and my co-guide **Dr. RAJESH GUPTA,** Associate Professor, Department of Civil Engineering, Malaviya National Institute of Technology Jaipur, India.

AJAY KUMAR GAUTAM

(**2014PCS5326)**

This is to certify that the above statement made by the candidate is correct to the best of my knowledge.

Dr. VINAY AGRAWAL

Assistant Professor

Dr. RAJESH GUPTA

Associate Professor

ACKNOWLEDGEMENT

I express my deep sense of gratitude and great regards to my guide, **Dr. VINAY AGRAWAL** Assistant Professor**,** for his valuable guidance and support. His profound experience and incomparable expertise combined with his kind supportive nature has been a substantial asset for us throughout our learning experience. He spared no efforts to provide us the needed impetus.

I would also like to thank my co-guide **Dr. RAJESH GUPTA,** Associate Professor for his support and suggestions during our project. I feel immensely gratified in presenting the Dissertation report. I thank all our dear friends for their moral support and enthusiasm. I thank our parents for their blessings and prayers.

AJAY KUMAR GAUTAM

(2014PCS5326)

ABSTRACT

An effective finite element scheme should be applicable to both static and buckling of shell behavior and the rate of convergence in either case should be optimal and independent of the shell thickness. Such a finite element scheme is difficult to achieve but it is important that existing procedures be analyzed and measured with due regard to these considerations. In the static analysis structure analysis with static loading with variable end boundary condition but boundary condition did not effect on forces but found effect in deformation shape and bending moments. In static analysis size of meshing found as important phenomena if number of meshing is increased so number of nodes increase also the number of value of nodal solution is increased

The buckling behavior also important phenomena of shell structure Buckling of cylindrical shells subject to axial compression is addressed for shells having foamed metal cores. Optimal face sheet thickness, core thickness and core density are obtained which minimize the weight of a geometrically perfect shell with a specified load carrying capacity. Constraints imposed by wrinkling and yielding of the face sheets and yielding of the core are all considered especially in light of the coincidence of elastic buckling and face sheet yielding in the optimally designed perfect shell. In buckling analysis edge beam found more carrying capacity as compare to short edge. It is found that analysis in program is more time taken solution but more accurate results. The buckling analysis give buckling load with different condition and also give acceptance behavior of engineering structure.

Table of Contents

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1

INTRODUCTION

A Thin shell is defined as a shell with a thickness which is small compared to its other dimension and in which deformations are not large compared to thickness Known as shell. The basic difference between a shell structure and plate structure is that in the unstressed state the shell structure has curvature as opposed to the plate structure which is flat Membrane action in a shell is primarily caused by in-plane forces (plane stress), though there may be secondary forces resulting from flexural deformations. The word shell is an old word and is commonly used covering hard covering of a horizon or flat in other word shell is used to covering hard egg, Crustacea, tortoises, etc. According to dictionary shell is obtain from scalus its word meaning fish scale now there is a clear difference between the tough but flexible scaly covering of a fish and the tough but rigid shell of, say, a turtle. In this article we will discuss about manually made shell structures as used in various branches of engineering. The shell structure has many interesting features of use of shell in engineering. But it has most importance in structural engineering. The theory of structures tends to deal with a class idealized mathematical model. Despoil of many of the features make them recognizable as useful substance in engineering. Thus a beam is generally idealized as a line establish with certain mechanical properties, irrespective of whether it is a large bridge, an aircraft wing, or a flat spring inside a machine. In a similar way, the theory of shell structures deals, for example, with the "cylindrical shell" as an idealized existenc it is a cylindrical surface establish with certain mechanical properties. The treatment is the same whether the actual structure under study is a gas-transmission pipeline, a grain storage silo, or a steam boiler.

1.1 Types of Shell:

The following Popular type of shell structures are

1.1.1. Concrete shell:-

The concrete shell is also called thin shell concrete structure it have not interior column. Thin concrete shells which start appear in the 1920. It constructed from thin steel reinforcement concrete and in many cases lack any rib or additional reinforcing structure relying wholly on the shell structure itself. Shell may cast on site and pre-cast move assemble. The strongest form of shell structure is monoform shell which is cast as a single form he most common monolithic form is the dome, but ellipseshape and cylindrical shapes are also use in similar construction work.

Fig: 1.1 Concrete Shell

1.1.2. Grid Shell: -

In the grid shell structure strength is obtain from double curvature in similar way of fabric structure. The grid structure may obtain from any material but generally in this construction use wood, or steel. Large span grid shells are generally constructed by initially putting out the main frame member flat in Regular Square or rectangular framework.

Fig: 1.2 Grid Shell

1.1.3. Membrane structure:-

The membrane structure is a tensioned structure or tension structure. It works together with the constructed element such as column, cable and other construction member etc. It finds a form Membrane can also use nonstructural cladding recently used in Beijing.

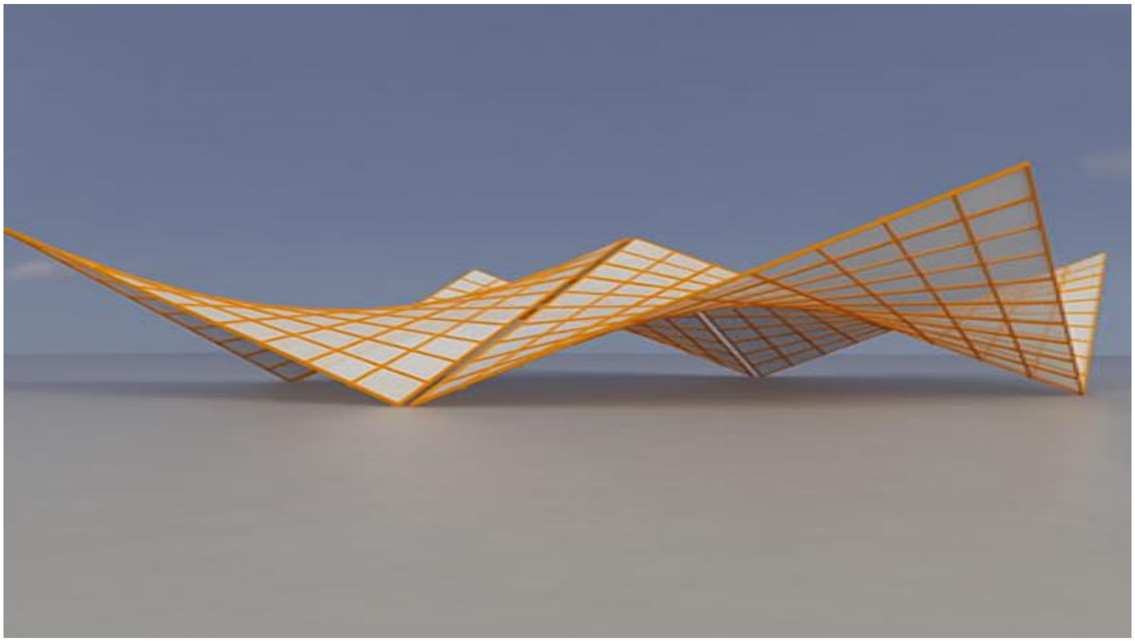


Fig: 1.3 Membrane structure

CHAPTER 2

Literature Review

2.1 Membrane Behavior of Reinforced concrete shell structure and Limits on Reinforcement

In the year of 1988 Kye J.Hanand S.T Mau in their study examined of Reinforced concrete shell elements under membrane stress these are three types of failure under mode:

1. Yielding of reinforcement in both the direction
2. Yielding of reinforcement in one direction and yielding of concrete
3. Yielding of concrete

The element behavior is ductile in case of the first two mode and brittle in the last mode. The classification of an element depends on quantity of reinforcement, the ratio of applied principal stresses and their orientation with respect to the reinforcement direction. The three failure mode are separated by boundary condition. These boundary condition are represented by 2-D in the terms of reinforcement ratio C_x, $C_{y.}$ The boundary between the first two modes called inner boundary and also defines the limit of reinforcement beyond which strength formula of reinforcement yielding is not valid. The other boundary between the last two mode defines the limit beyond which element behaves in a brittle manner. Euler's investigation (1744) of the elastic stability of slender column paved the way for the development of "Classical theory" of elastic buckling of structures in which the governing equation of the problem are linearized and the resulting Eigen value problem yield the buckling load as Eigen vector. That kind of theory had wide application and it was applied to the buckling cylindrical shell between 1911 and 1934 by many independent works including Lorenz, South well, Timoshenko and Gare (1965) and Brush and Almorth (1975). Classical theory predicts that cylindrical shell buckle under uniform axial compression stress $\sigma_{cr}= \frac{E}{\sqrt{3(1-v^2)}}(\frac{t}{r})$

Where E is young modulus of elasticity, t thickness, radius, v poison's Ratio,

The free vibration characteristics of the joined spherical–cylindrical shell with various boundary conditions are investigated. The boundary conditions considered here in are free simply supported free and clamped free for the joined cylindrical, spherical shell structures. The Flugge shell theory and Rayleigh's energy method are applied in order to analyze the free vibration characteristics of the joined shell structure and individual shell components. In the modal test, test module is used to determine mode shapes and obtain natural frequencies of the shell cylendrical structure.

2.2 Buckling behavior Load analysis of reinforced concrete shell structure

Load analysis of shell structures are usually initialize with a linear buckling analysis. The results are buckling mode and load factors. Load factors are evaluated for a higher limit of the ultimate load. Buckling mode show that how the structure will buckle. As it is well known the structures are more sensitive against irregular in the shape of the lower buckling modes, they also give an idea of a conservative irregularity and can be applied to the ideal model as geometric simplification which is a simple function in the ansys

2.3 Interaction of Bending and Stretching Effects

The mechanical properties of a shell element describe its resistance to deformation in terms of separable stretching and bending effects. Loads applied to the shell are carried in general through a combination of bending and stretching actions, which generally vary from point to point. One of the major difficulties in the theory of shells is to find a relatively simple way of describing the interaction between the two effects. This aspect of the theory has been troublesome from the beginning. Rayleigh[1] argued that the deformation of a thin hemispherical bowl would be primarily in extensional, and accordingly he developed a special method of analysis which took into account only the bending energy of the shell.

On the other hand Love[2] argued that for thin shells stretching was the dominant effect. At that time Love had not grasped the strong contrast between open and closed shells. The controversy was resolved by Lamb[3] and Basset[4], who solve Love's general equations for a cylindrical shell and demonstrated the possibility of a narrow boundary layer in which there was a rapid transition between bending and stretching effects. The width of the layer was determined by the interaction between those effects.

2.4 Catastrophic Failures

The property of closed shell structures being rigid and strong is of great practical values. But it should not be in ignorance of a well-known design principle: efficient structures may fail catastrophically. Here the term "efficient" describes the consequences of using the closed shell principle. By designing a shell

Structure is a covered box rather than an open one we may be able to use thinner sheet material and hence produce a more efficient design. On the other hand, thin shells under compressive membrane forces are prone to buckling of a particularly unstable kind. The rapid change in geometry after buckling and consequent decrease of load capacity leads to catastrophic collapse. This is illustrated by the well-known experience of "crumpling" of thin wall cylinders like soda cans, under axial compression.

2.5 Optimization of shell structure

Expected form shells optimized for stiffness under sustain loading and membrane structures act in a pure membrane state of stresses, either because bending is minimized or not minimized even present by definition. Physical experiments as soap films and hanging out models have been used since centuries to generate optimal shapes of membranes in a tension and shells in compression. In this article present a numerical methodology to simulate the physical experiments as well as how they can be merged among each other and with the most general technology of structural optimization. Light weight structure as shell membrane are defined as the optimistic use of material to carry external load or pre stress. Material is used optimally within a structural member if member is subjected to membrane force rather than bending. The structural response may be based on linear elastic, eigenvalue, and geometrically nonlinear analyses. In particular, the imperfection sensitivity with respect to buckling is discussed. A few selected examples demonstrate the versatility of optimization schemes in shell design, among these are the tuning of a bell and the form finding of a classical reinforced concrete dome shell.

2.6 Buckling behavior of Shell structure

Buckling is a very important phenomena of civil engineering. Buckling is give the bending effect of the structure. Buckling is time step process. Bucking give the limitation of applied load it is give that after a critical condition the standing structure may fail without warning it known as buckling of structure. It is refer in shell analysis so it is known as buckling of shell structure. Buckling is depends on the end condition of shell tendency of applied load behavior of applied load and nature of load. The end condition of compression member and shell structures are depend on requirement of object. In compression member the buckling load is calculate by Euler's method similarly in shell structure for calculate of critical load in different methods. But appropriate method calculate critical load if finite element method. Finite element is a modern technique for evaluate the buckling load analysis of shell structure. It can apply for any shape of shell structure i.e cylindrical, elliptical, parabolic, circular shell. These above shell structures can used with under water construction and open space construction. For the buckling analysis the most appropriate is for axial loading. Buckling introduce a secondary effect of moments. The failure give meaning failure of structure with less than ultimate compressive stress at that point of material withstanding. Buckling is a sudden drawdown condition of the structure. It can evaluate with mathematical model and finite element model. Buckling also depends on length, radius of gyration, moment of inertia etc.

In the dynamic buckling analysis, structure behavior is depends on frequency of vibration, motion of wave, time step, direction of wave motion. For the long span structure this phenomena also included. Dynamic analysis is also depends on zone of structure. Basically country divided into different zones. In Ansys program for dynamic analysis also give option of harmonic and transit analysis of structure so this phenomena also important for civil engineering analysis.

CHAPTER 3

ANSYS SOFTWARE

3.1 Introduction:

Ansys is a finite element program it is used to analysis a complex finite element problem. This is a marketing and a commercial program. It have ability of solve linear, nonlinear, elastic, isotropic, non-isotropic problems. Ansys program used in structural engineering to analysis structural design problems, and for fluid mechanics problem used flatiron. In Ansys program also used in harmonic form transit, buckling substructure form. These form are structural analysis form. Similarly in fluid mechanics analysis it have another options and for thermal analysis it have different options.

For civil engineering analysis perform in static, harmonic, transit, buckling, and in mechanical form fluid, thermal etc. Ansys program is based on finite element technique it is analysis a element strip so it is form structure in meshing form.

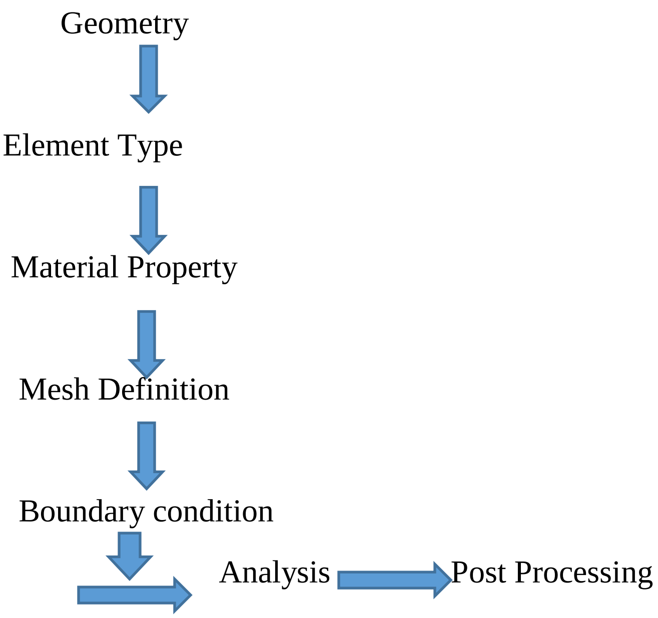
- Pre Processing
- Solution
- Post Processing

3.2 Methodology of Ansys

The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. Finite element is reliable method of analysis of shell structure solution frequency is high and accurate. It is time require methodology in Ansys program solution is obtain in nodal form for take more accuracy analyze structure node to node so this method highly efficient. The next few sections of this chapter cover general steps that are common to most analyses. In analysis problem program follow simple three step and obtain appropriate solution of structure.

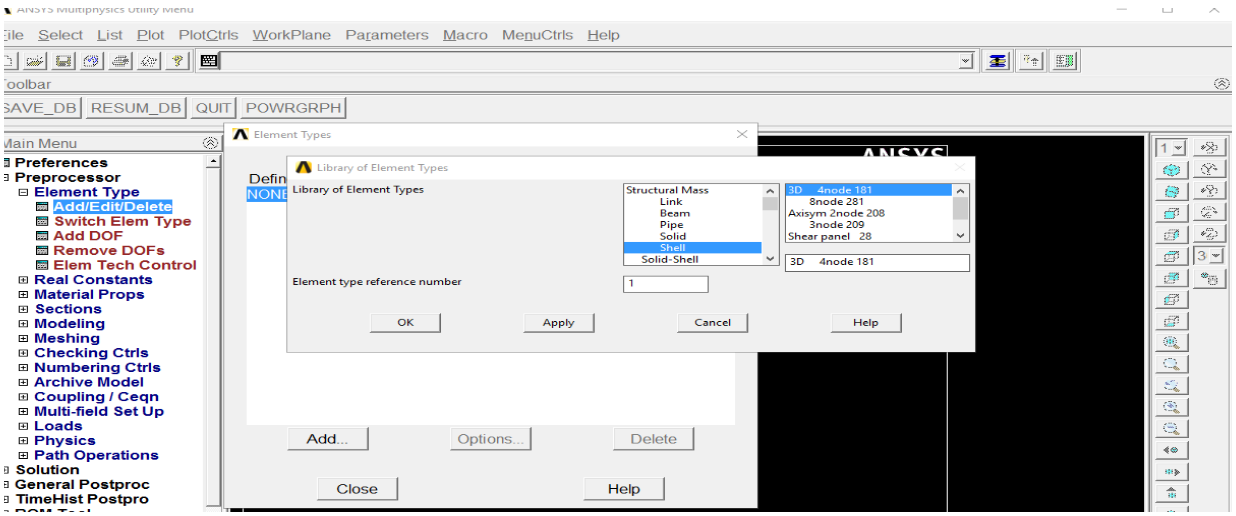
3.2.1 Modeling with Ansys

The following steps adopt in Ansys Modeling



3.2.1.1 Geometry& Element Type:

The Geometry of the structural element select from table. In the table of element type. There are various types present



Library of Element Types: Structural Shell, 3D 4node 181

Element type reference number: 1

Fig: 3.1 Selection of Element in Ansys

According to suitable use elements are select i.e. Link, Beam, Pipe, Solid, Shell, couple field, Thermal etc. Let take a Shell structure. In Ansys Different types of shell structure exists 4node181, 8node281, Axisym3node208, 3node209, shear panel and others. As suitable selection consider.

3.2.1.2 Material Property:

After the element selection material property will assign.

- Linear and nonlinear property
- Isotropic property, orthotropic, & anisotropic property
- Constant heat temperature and dependents of temperature

The material property is depends on physical as well as chemical composition of material. Mostly in Ansys Analysis software material property depends on poison's ratio and modulus of elasticity. I.e. Modulus of Elasticity of Steel (E_s) =$2x10^5$ & Poisson's Ratio (PRXY) =0.28, similarly modulus of elasticity of Aluminum (E_A) = 69 Gpa & Poisson's Rati (PRXY) = 0.69. These value insert in provide a particular space in software. For a composite material the value of poison's ratio and modulus of elasticity provides.

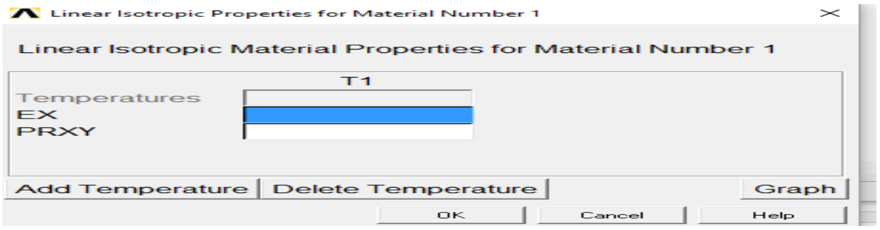


Fig: 3.2 Material of Element in Ansys

3.2.1.3 Selection of section

When define element property next step is create analytical model. The analytical model is generate as per requirement of object. It may be symmetric model may be unsymmetrical model. In Ansys program is generate in form of node and member. There are two methods to genrate the finite element model.

Menu → Preprocessor → Material property → material model

(i) Solid modeling.

(ii) Direct Generation

3.2.1.3.1 Solid Modeling:

In Solid modeling describe the geometry shape of model then instruct the Ansys program to automatically mesh. Geometry with element, node and key point. We can also control size manually and shapes of element that program creates.

3.2.1.3.2 Direct Generation:

In direct generation we defined the coordinate and location of each node and the connectivity of each element. Node or key point define as per actual or design dimension of structure. Nodes and Key points are two different things but they can exists on each other. First Node/ can define then key point vice versa also exists.

(i) Key point: Key points are merely geometric constructions that we create while creating a geometric model of the given problem.

(ii) Nodes: Nodes are obtained after the complete the meshing operation. These are the points where desired values are obtained. Also nodes are the points where the loads are applied.

3.2.1.4 Boundary Condition:

Support condition means end condition of a structure. Support condition is use to bound the solid region and it is use to restrain structure against rigid body motion. Support is done by joining structure to earth ground through foundation. The resulting boundary conditions are called motion restrain. It may two dimensional or three dimensional.

Two types of boundary condition.

I. Geometric or essential boundary condition which are imposed on primary variable like displacement

II. Natural boundary condition which imposed on primary variable like force.

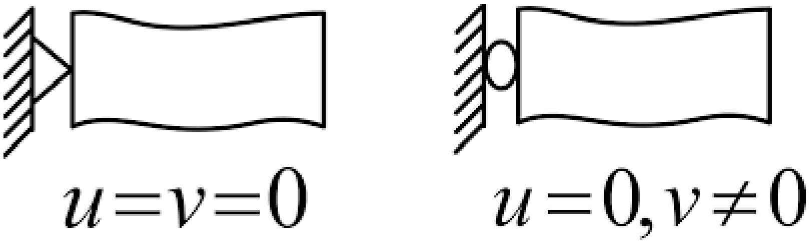


Fig: 3.3 Geometric Boundary condition

The Natural boundary condition is imposed at during the evaluation of the element of structure. While essential boundary conditions are imposed after the assembly of the element.

3.2.1.5 Load/Force

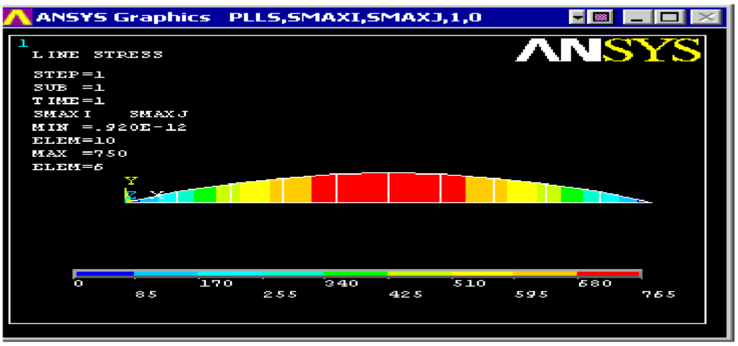
Loads are define by symbol F. the load form is given by F=kd form where F is action force on structure and k is stiffness coefficient and d is displacement form. Applied force may be concentrated form, uniformly distributed form, pressure form, it is depends on analytical condition of structure.

i. Concentrated Load: It is defined with command F and it has three option
    i. The Node number where the load is applied
    ii. The Load Type
    iii. Value of load

$\mathbf{F_X}$, $\mathbf{F_Y}$, $\mathbf{F_Z}$, are forces in x,y,z direction and $\mathbf{M_X}$, $\mathbf{M_Y}$, $\mathbf{M_Z}$ are acting moments in reactively directions.

ii. Commands:
iii. F,3,F_X,2000 Force x direction at node 3
    F, 5, F_Y,-10e3 Force -y direction at node 5
    F, 2, M_X, 5e3Moment about x axis at node 2
    F, 1, M_Z, 48e3 Moment about z axis at node 1

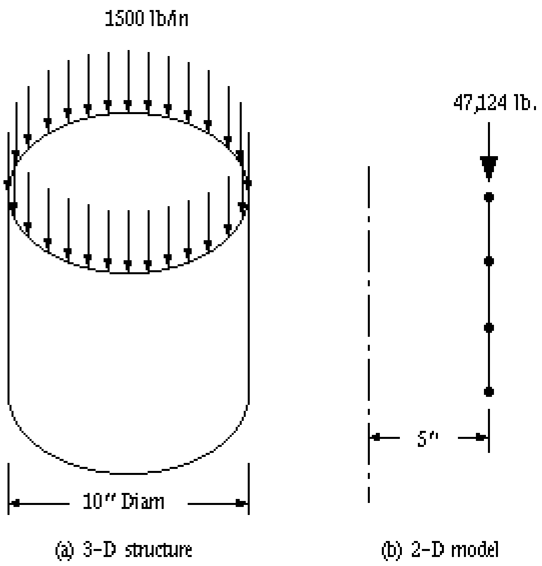
iv. The load distribution command have following options
    i. The element number where the load is applied on section.
    ii. Pressure value that applies uniformly throughout on section
    iii. PRES, for pressure on examine section



```
1
LINE STRESS
STEP=1
SUB =1
TIME=1
SMAXI     SMAXJ
MIN =.920E-12
ELEM=10
MAX =750
ELEM=6
```

Fig: 3.4 Loading Figure

v. General distributed Load: The load distribution load is found in several ways
    i. The node number where the load is applied
    ii. PRES for pressure or CONVS for thermal convection
    iii. The pressure value for PRES & For CONV



(a) 3-D structure (b) 2-D model

Fig: 3.5 Load on Structure

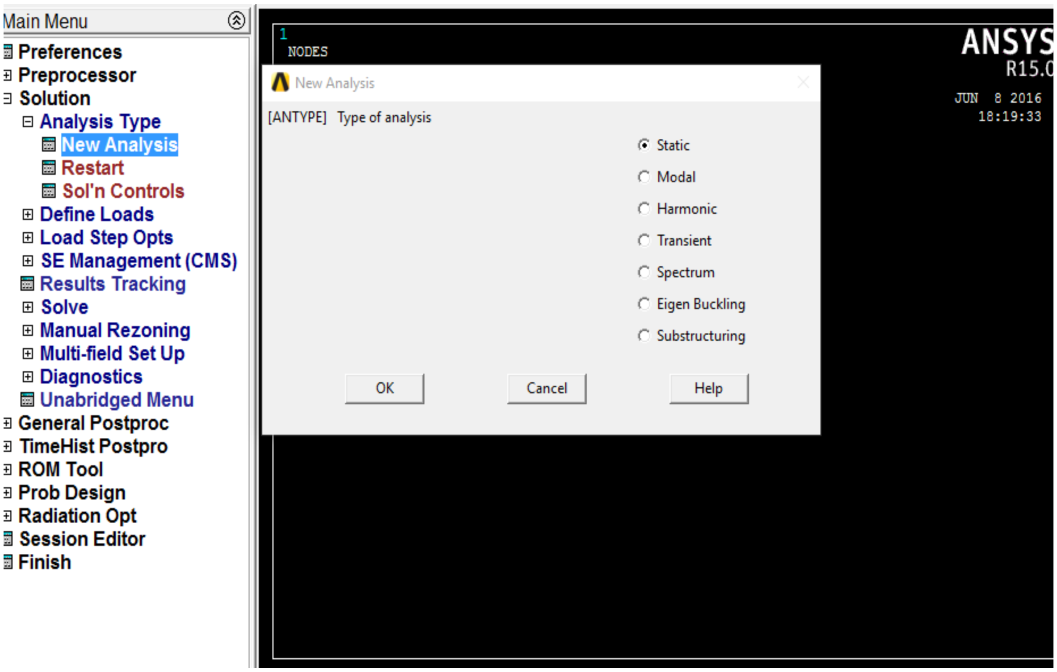
3.2.1.6 Analysis or Solution:

The Analysis type to be used is based on the loading condition and the response wished to calculate. I.e. If Natural frequency and mode shape are to be calculate then a model analysis ought to be chosen. The Ansys Software offered different following types of Analysis

i. Static Analysis
ii. Transit
iii. Harmonic modal Analysis
iv. Spectrum
v. Buckling Analysis
vi. Sub structure Analysis

The above analysis is valid for structural mode and these above not valid for thermal mode. In static analysis give behavior of nodal solution, in transit analysis give displacement mode similarly in harmonic analysis give time step solution and buckling load analysis give the buckling/ critical form of the structure. These analysis has important analysis property for structural design.

Buckling analysis is not only important for shell structure but also it is important for compression member i.e. column. It is give critical load condition of structure.



[ANTYPE] Type of analysis

● Static
○ Modal
○ Harmonic
○ Transient
○ Spectrum
○ Eigen Buckling
○ Substructuring

Fig: 3.6 Analysis type

3.2.2 Post Processing:

The General post processing mode is a review technique in program. The general post processing is give idea of behavior of cylindrical shell structure. In this mode first obtain result of applied load of given figure result is in the form of nodal solution and member load also in Ansys program. General post processing mode give figure of time step solution it give idealize behaviors of shell structure. In analysis process can check a particular section in mode of buckling, harmonic, static, in structural analysis. In the post processing solution is obtain in graphical form also. The post processing mode give deformation shape deformation shape give idea of structure give towards figure of instability at critical condition. In this mode obtain result of maximum and minimum value of nodal solution and maximum value of stress in respective nodes

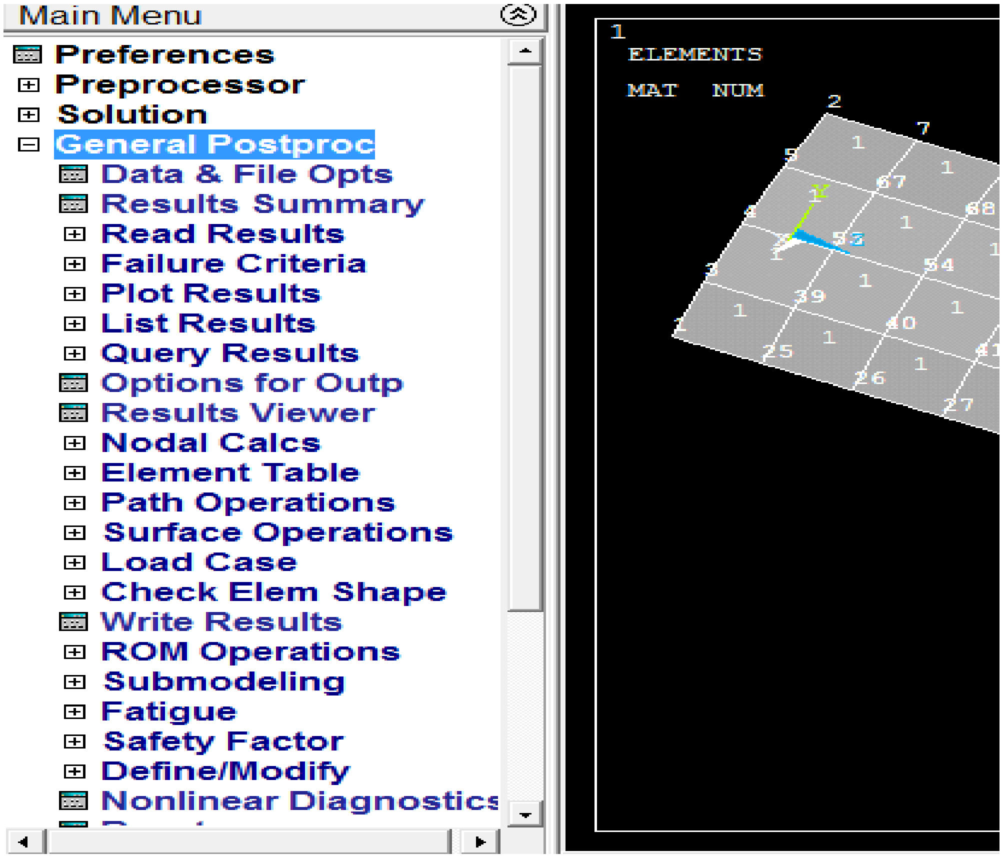


Fig: 3.7 General post processing

CHAPTER 4

FINITE ELEMENT ANALYSIS OF SHELL STRUCTURES

4.1 Introduction

“The finite element Analysis of shell structures” Give meaning study of a structure about its behavior. Finite element is a computerized method for determine how a structure react to real existing world forces. Also effect of vibration, heat, flow fluid and different others physical effects. Finite element shows whether a product will break, wear out or work the way it is designed. Finite element is subdivision of a large problem into smaller & simpler parts called finite element method.

4.2 Finite Element Analysis of Shell

General discussion of shell structure types, classification shape of shell in pervious chapter. In this chapter will disuses how shell structure behaves with static loading and also predicting the buckling behavior of shell structure. For Analysis of Shell structure first Design a model of shell structure then analyze as Static analysis after Buckling analysis of shell structure then check compression with different size, different boundary condition, and different loading check behavior of shell element.

4.2.1 Modeling of Shell Element.

For Design of shell structure first design its modeling. For the modeling require some numerical .Data for design and Analysis.
Size of curve Shell element

Table 4.1 Modeling Size

S.No	Component	Size (m)
1	X-Component	6.0 m
2	Y-Component	0.60m
3	Z-Component	24.0m
4	Radis (R)	6.0m
5	Angle	60˚ (Degree)

The process of deign of model have been discussed in pervious chapter.

4.2.1.1 The process of modeling first define the structural discipline in preferences

4.2.1.2 Preprocessor

4.2.1.2.1 Element Types:

I. For particular such problem shell element type is Type1 Shell 181. Shell 181 is 3 node and 3D Element.

II. In the next step some technical terms element stiffness is (K_1) Bending and Membrane

III. Storage of layer data(K_8) is bottom 1^{st} top last

IV. User thickness option (K_9) is No U Thick routine

These above terms are user define terms

4.3.1.2.2 Real Constant:

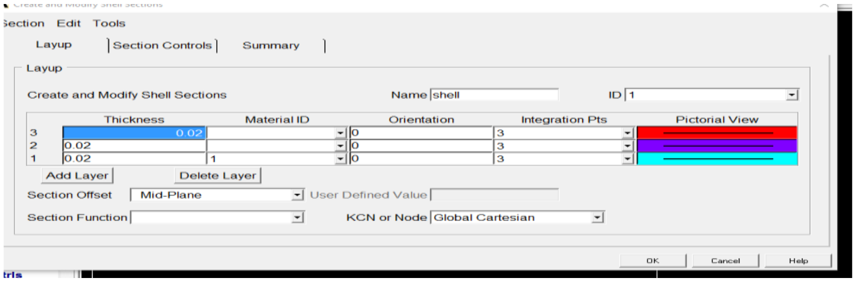
For this pre define element no real constant define because for Shell 181 element real constant predefine in this Ansys software.

4.3.1.2.3 Material Property

The material property is user define i.e. Liner, Nonlinear, density, thermal expansion, damping, friction, etc. For Shell 181 element. For this particular problem we selected Linear and isotropic for Shell 181 and this analytical problem for this model we used the value of elasticity and poison's Ratio are (E_1=25000,) N/mm^2 and poison's ratio is (PRXY) 0.28

4.3.1.2.4 Sections

In the section classification select shell from different element and give thickness of layer of shell model. In this software section we can distribute the model in different model elastic property and poison's ration can also allocate to different layer. Thickness of shell element can also vary as per requirement. And orientation (0˚, 30˚, and 45˚ 90˚) can user defines.



Layup

Create and Modify Shell Sections

Name shell

ID 1

	Thickness	Material ID	Orientation	Integration Pts	Pictorial View
3	0.02		0	3	🟥
2	0.02		0	3	🟪
1	0.02	1	0	3	🟦

Section Offset Mid-Plane

User Defined Value

Section Function

KCN or Node Global Cartesian

Fig: 4.1 Section and thickness selection

4.3.1.2.5 Modeling:

In process of modeling there various process of create model i.e. Area method, volume method, node, Line, key point. Shell181is create by volume method. The above value inserts in a specified space and created model.

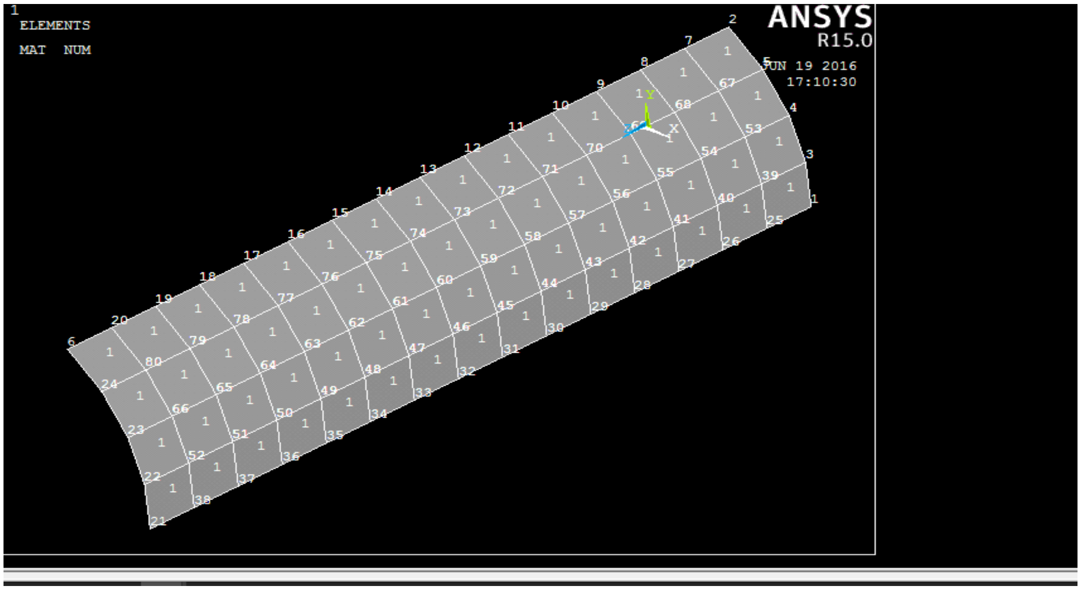


Fig: 4.2. Shell181 Model

4.3.1.2.6 Analysis:

For the Analysis of shell structure apply a pressure of intensity -1000N/m^2 on area of shell element and support condition is fixed the following results obtained.

The analysis is depends on size of meshing and number of meshing. In the program the shape of meshing is Tet, Hexagonal. It is found that if node is not defined and we used arbitrary meshing in program the nodes automatic defined. If the number of meshing is increased so number of nodes increased automatic, and efficiency of result also increased.

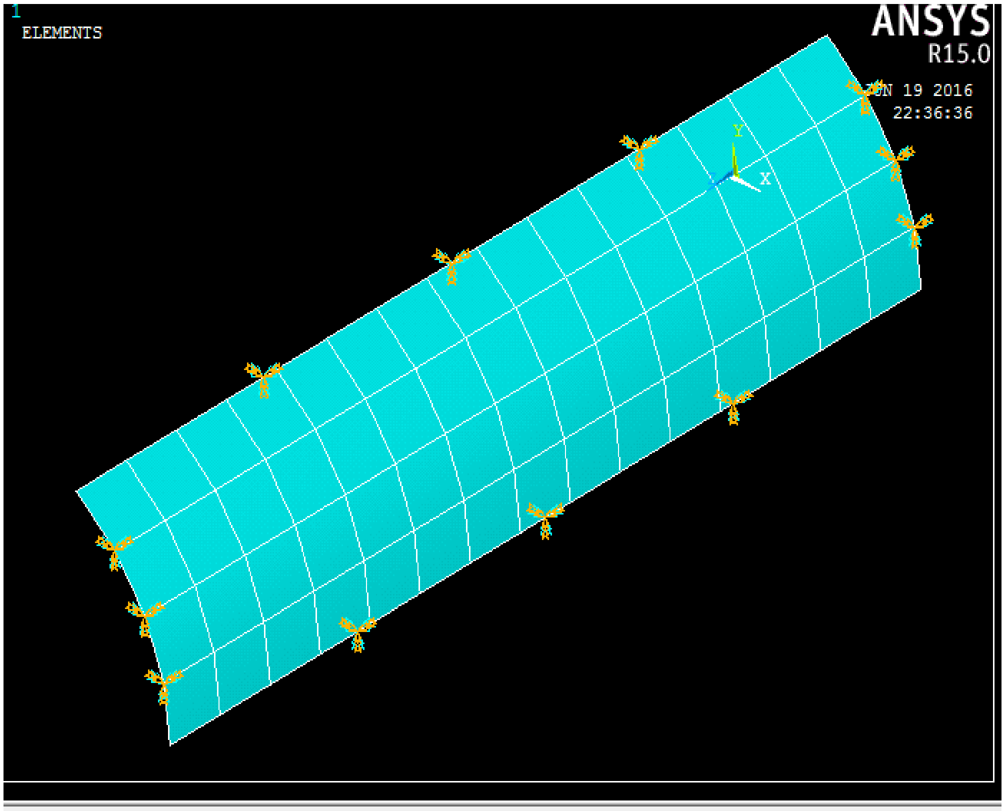


Fig: 4.3 Analytical form of shell

4.3.1.3 Post Processing:

In the preprocessing mode can analyze shell structure. After Analysis format comes in post processing mode. In The post processing mode program show different nodal solution i.e. Nodal stresses, Strain, forces, displacement, variations in two different quantities. In this Analysis structure we have define 80 nodes so every node show force in X, Y, Z Direction and moment in X,Y,Z direction these forces and moments are shown in following.

Case-1 First Apply a pressure of -1000N/m^2 on area of shell structure & Boundary condition is fixed on other end. (All DOF is zero)

The following X,Y,Z loads are in The global coordinate system

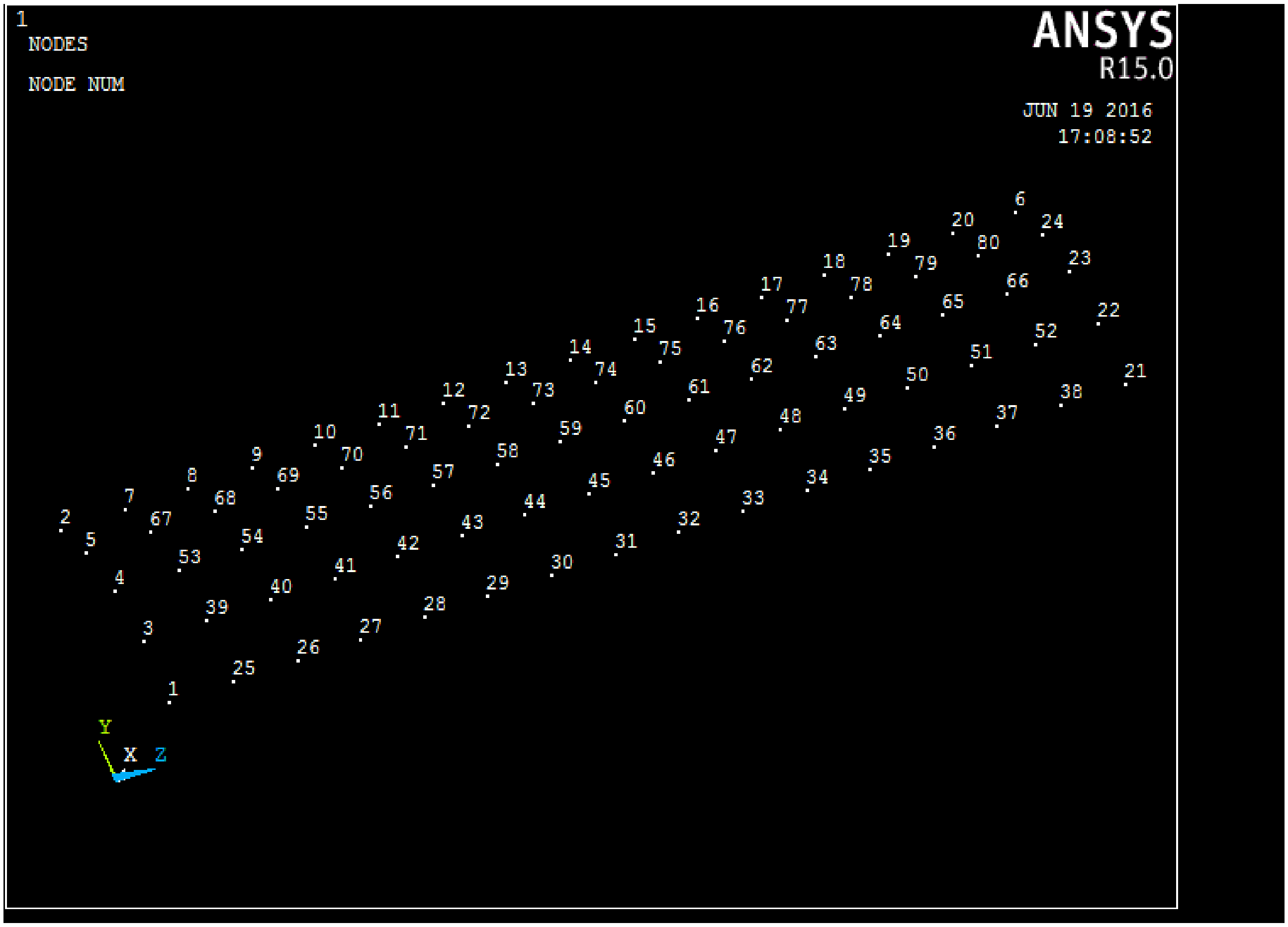


Fig: 4.4 Shell structure in Node form

```
NODE     FX          FY          FZ          MX          MY          MZ
 1   -243.8      -3264.      -1837.       1.290      -10.36       30.94
 2   -2949.       1421.      -1837.       8.325      -6.296      -30.94
 3   -159.8      -2037.      -1485.       39.11      -102.0       4.570
 4   -797.7      -460.6      -562.5       84.39      -146.2     -0.1200E-08
 5   -1844.       880.3      -1485.       68.80      -84.88      -4.570
 6   -2949.       1421.       1837.      -8.325       6.296      -30.94
 7   -5989.       2875.      -1240.      -3.081       5.182      -107.4
 8   -8151.       4638.      -427.0       2.309      -2.754      -18.72
 9   -8487.       4944.       119.4      -1.115       1.311      -10.63
 10  -8208.       4677.      -18.10      0.5796     -0.6279      -32.10
 11  -8306.       4776.      -19.46     -0.4561      0.4651      -20.01
```

```
  NODE     FX          FY           FZ           MX           MY           MZ

  12   -8288.       4758.        6.461       0.3894      -0.3605       -24.36
  13   -8288.       4757.       -7.422      -0.3731       0.3412       -23.31
  14   -8288.       4757.        7.422       0.3731      -0.3412       -23.31
  15   -8288.       4758.       -6.461      -0.3894       0.3605       -24.36
  16   -8306.       4776.        19.46       0.4561      -0.4651       -20.01
  17   -8208.       4677.        18.10      -0.5796       0.6279       -32.10
  18   -8487.       4944.       -119.4        1.115       -1.311       -10.63
  19   -8151.       4638.        427.0       -2.309        2.754       -18.72
  20   -5989.       2875.        1240.        3.081       -5.182       -107.4
  21   -243.8      -3264.        1837.       -1.290        10.36        30.94
  22   -159.8      -2037.        1485.       -39.11        102.0        4.570
  23   -797.7      -460.6        562.5       -84.39        146.2      -0.1202E-08
  24   -1844.       880.3        1485.       -68.80        84.88       -4.570
  25   -504.5      -6624.       -1240.       -2.948        5.259        107.4
  26   -58.96      -9378.       -427.0        1.230       -3.377        18.72
  27    38.48      -9822.        119.4      -0.5780        1.621        10.63
  28   -53.10      -9447.       -18.10       0.2540      -0.8159        32.10
  29   -16.51      -9581.       -19.46      -0.1747       0.6275        20.01
  30   -23.52      -9557.        6.461       0.1176      -0.5175        24.36
  31   -24.45      -9556.       -7.422      -0.1090       0.4937        23.31
  32   -24.45      -9556.        7.422       0.1090      -0.4937        23.31
  33   -23.52      -9557.       -6.461      -0.1176       0.5175        24.36
  34   -16.51      -9581.        19.46       0.1747      -0.6275        20.01
  35   -53.10      -9447.        18.10      -0.2540       0.8159        32.10
  36    38.48      -9822.       -119.4       0.5780       -1.621        10.63
  37   -58.96      -9378.        427.0       -1.230        3.377        18.72
  38   -504.5      -6624.        1240.        2.948       -5.259        107.4
  39    0.2892E-09  0.9459E-10 -0.1883E-09 -0.4160E-11  0.8902E-10  0.5035E-10
  40   -0.2228E-09 -0.1910E-10 -0.1762E-09 -0.2526E-10  0.2101E-09  0.2487E-12
  41    0.1692E-09  0.2547E-10 -0.1560E-09 -0.2218E-10  0.1723E-09  0.4452E-11
  42   -0.8527E-10 -0.7276E-11 -0.2112E-09 -0.2847E-10  0.2263E-09  0.1676E-11
  43    0.2296E-10  0.4547E-11 -0.1301E-09 -0.1878E-10  0.1482E-09 -0.2659E-11
  44    0.6139E-11  0.9095E-12 -0.9231E-10 -0.1281E-10  0.1056E-09 -0.2318E-12
  45   -0.1637E-10  0.1364E-10  0.8004E-10 -0.1216E-11 -0.7601E-12 -0.4918E-12
  46    0.7049E-11  0.1091E-10 -0.6298E-10 -0.2501E-11  0.3152E-11 -0.1381E-11
  47   -0.7049E-11  0.6366E-11  0.1080E-09  0.1404E-10 -0.1002E-09  0.5822E-12
  48    0.2865E-10  0.7276E-11  0.1453E-09  0.2160E-10 -0.1513E-09  0.1751E-11
  49   -0.8049E-10  0.5457E-11  0.2183E-09  0.3067E-10 -0.2214E-09 -0.7695E-11
  50    0.1937E-09  0.2819E-10  0.1648E-09  0.2417E-10 -0.1755E-09 -0.1011E-10
  51   -0.2376E-09 -0.3365E-10  0.2001E-09  0.2539E-10 -0.2069E-09  0.1001E-10
  52    0.2709E-09  0.9004E-10  0.2067E-09  0.3773E-11 -0.7452E-10  0.5618E-10

        ***** POST1 NODAL TOTAL    FORCE SUMMATION *****

LOAD STEP=     1  SUBSTEP=     1

 THE FOLLOWING X,Y,Z FORCES ARE IN THE GLOBAL COORDINATE SYSTEM


NODE       FX          FY          FZ          MX          MY          MZ
 53   0.6048E-10  0.3456E-10 -0.1669E-09 -0.7962E-10  0.1348E-09 -0.3272E-11
 54  -0.2274E-11  0.4093E-11 -0.1842E-10 -0.6999E-12 -0.6246E-11 -0.5045E-12
 55   0.1091E-10 -0.2046E-10 -0.7958E-11  0.3428E-11 -0.2697E-11  0.6837E-11
 56  -0.2274E-11  0.6366E-11  0.2274E-12 -0.3460E-11  0.6035E-11  0.5373E-12
 57  -0.7276E-11 -0.3183E-11  0.7958E-11 -0.5711E-12  0.1081E-11 -0.5613E-12
 58  -0.3183E-11 -0.1728E-10  0.9322E-11 -0.4032E-11  0.3546E-11  0.3553E-12
 59   0.1637E-10 -0.3183E-11  0.6821E-12  0.5763E-11 -0.7982E-11  0.7177E-11
 60   0.4547E-12  0.2728E-10 -0.7276E-11 -0.1546E-11  0.1080E-10 -0.4867E-11
 61   0.1364E-11  0.7276E-11 -0.2956E-11 -0.3612E-11  0.1216E-10  0.1440E-11
 62  -0.3183E-11  0.9095E-11 -0.1251E-10  0.4994E-11 -0.1328E-10 -0.1666E-11
 63   0.9095E-11  0.9095E-11 -0.1728E-10 -0.5986E-11  0.6402E-11 -0.6735E-11
```

```
 NODE      FX          FY          FZ          MX            MY            MZ
 64  -0.1000E-10 -0.8185E-11 -0.8185E-11  0.1268E-11 -0.2466E-11 -0.4572E-11
 65   0.5912E-10  0.2547E-10 -0.1523E-10 -0.1830E-10  0.3448E-10  0.1540E-10
 66   0.5321E-10  0.4138E-10  0.1535E-09  0.8204E-10 -0.1375E-09 -0.4839E-11
 67   0.2028E-09  0.2005E-09 -0.4900E-10 -0.6404E-10  0.4362E-10 -0.5878E-10
 68  -0.1382E-09 -0.1723E-09 -0.5230E-11 -0.1741E-09  0.1331E-09  0.5564E-11
 69   0.1010E-09  0.1446E-09 -0.6594E-11 -0.1386E-09  0.1083E-09 -0.9097E-12
 70  -0.5321E-10 -0.7503E-10  0.1819E-11 -0.1806E-09  0.1352E-09 -0.7418E-11
 71   0.8185E-11  0.2410E-10  0.2501E-11 -0.1129E-09  0.8469E-10 -0.3003E-11
 72  -0.7276E-11 -0.1182E-10  0.2956E-11 -0.9353E-10  0.7455E-10  0.2188E-11
 73   0.4093E-11  0.1364E-11  0.7844E-10  0.2252E-11 -0.1247E-11  0.8977E-11
 74  -0.5457E-11 -0.4093E-11 -0.7344E-10 -0.2369E-11  0.4338E-11  0.1070E-10
 75   0.4547E-12 -0.7276E-11  0.2728E-11  0.8546E-10 -0.6493E-10  0.5035E-11
 76   0.3047E-10  0.2137E-10 -0.4320E-11  0.1154E-09 -0.9006E-10 -0.6095E-11
 77  -0.5093E-10 -0.7549E-10 -0.6139E-11  0.1792E-09 -0.1388E-09 -0.6804E-11
 78   0.1205E-09  0.1410E-09 -0.4320E-11  0.1421E-09 -0.1102E-09  0.4251E-11
 79  -0.1091E-09 -0.1578E-09 -0.4320E-11  0.1487E-09 -0.1067E-09  0.8585E-11
 80   0.2228E-09  0.2215E-09  0.4025E-10  0.8148E-10 -0.6081E-10 -0.6825E-10

          ***** POST1 NODAL TOTAL    FORCE SUMMATION *****

 LOAD STEP=     1  SUBSTEP=     1

  THE FOLLOWING X,Y,Z FORCES ARE IN THE GLOBAL COORDINATE SYSTEM

 ***** SUMMATION OF TOTAL    FORCES AND MOMENTS IN THE GLOBAL COORDINATE SYSTEM
*****
  FX  =  -124707.7     FY  =  -72000.00     FZ  = 0.5254464E-1 MX  = 864000.0
  MY  =  -1496492.     MZ  =  0.6912160E-10
```

Forces and moments in the global coordinate system

```
NODE       FX              FY                FZ             MX             MY         MZ

 TOTAL VALUES
 VALUE    -124707.7 -72000.00       0.0000   864000.    0-1496492.      0.0000
```

Minimum and Maximum Value of Stress.

```
MINIMUM VALUES
NODE          8          26           54          1           1           2
 VALUE     -85152.    -0.13298E+06  -43179.      6343.9     -27814.      -
22257.
MAXIMUM VALUES
NODE          1           2           4          8          21           6
 VALUE      -835.19     -18167.     -2667.2      65339.      27814.      22257.
```

Minimum and Maximum Value of Strain.

```
MINIMUM VALUES
NODE           8          26           4           1           1           2
 VALUE -0.24778E-05 -0.49268E-05 -0.90216E-06 0.64961E-06 -0.28481E -05-
0.22791E-05

 MAXIMUM VALUES
NODE         27            2           4           8          21           6
 VALUE   0.18293E-05-0.15807E-06 0.69634E-06 0.66907E-05 0.28481E-05 0.22791E-
05
```

Table: 4.2 Nodal Forces, moment Strain, Stress

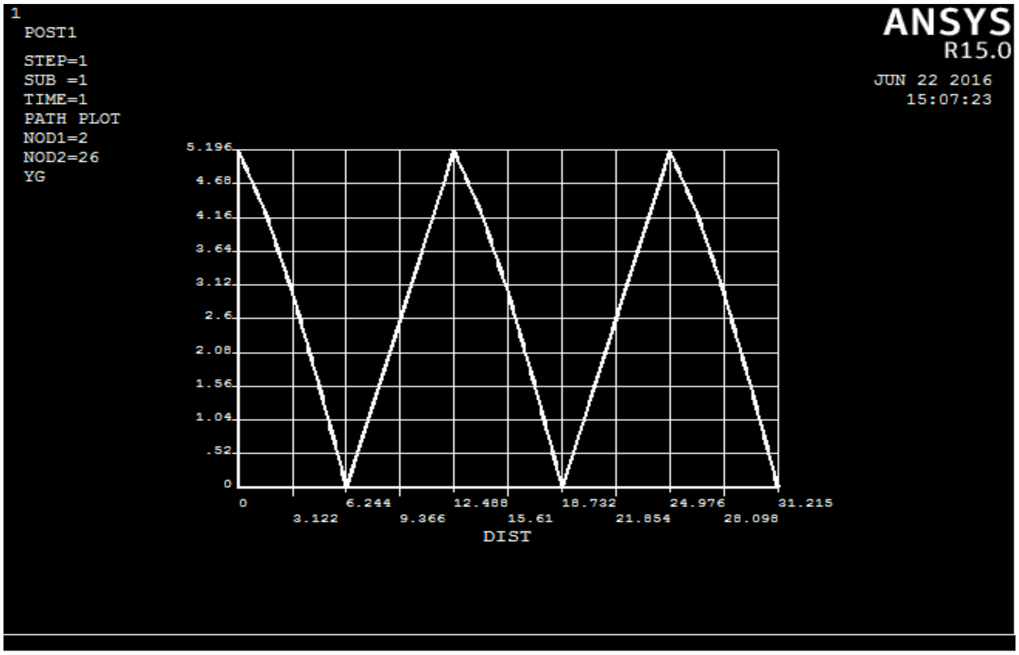


Fig: 4.5 Stress-Deformation Curve

4.3.1.4 Deform Shape

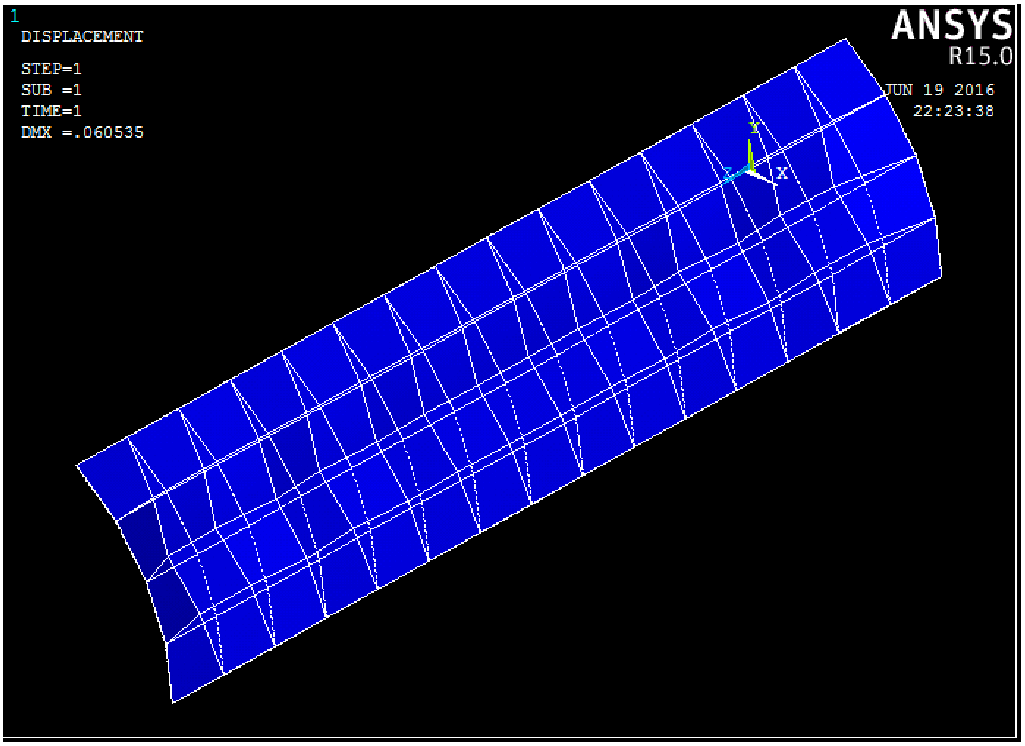


Fig: 4.6 Deform Shape

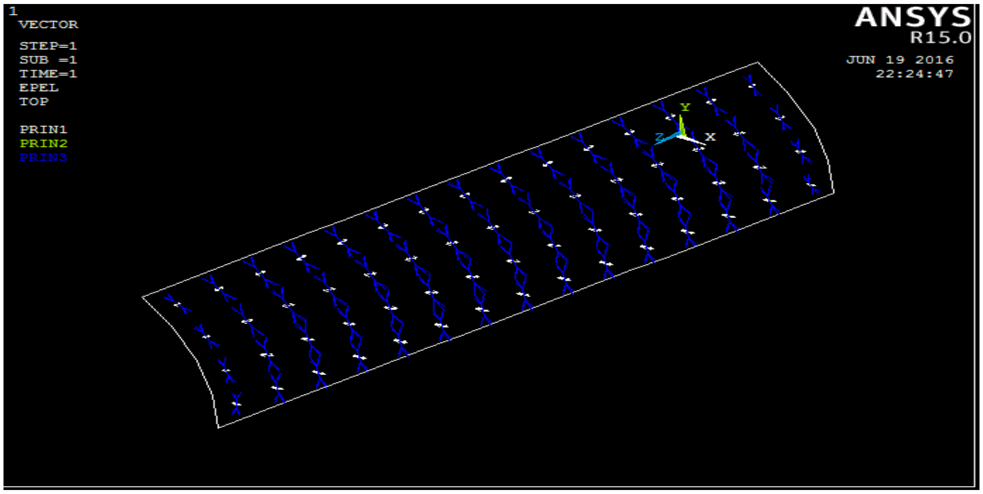


Fig: 4.7 Vector strain Figure

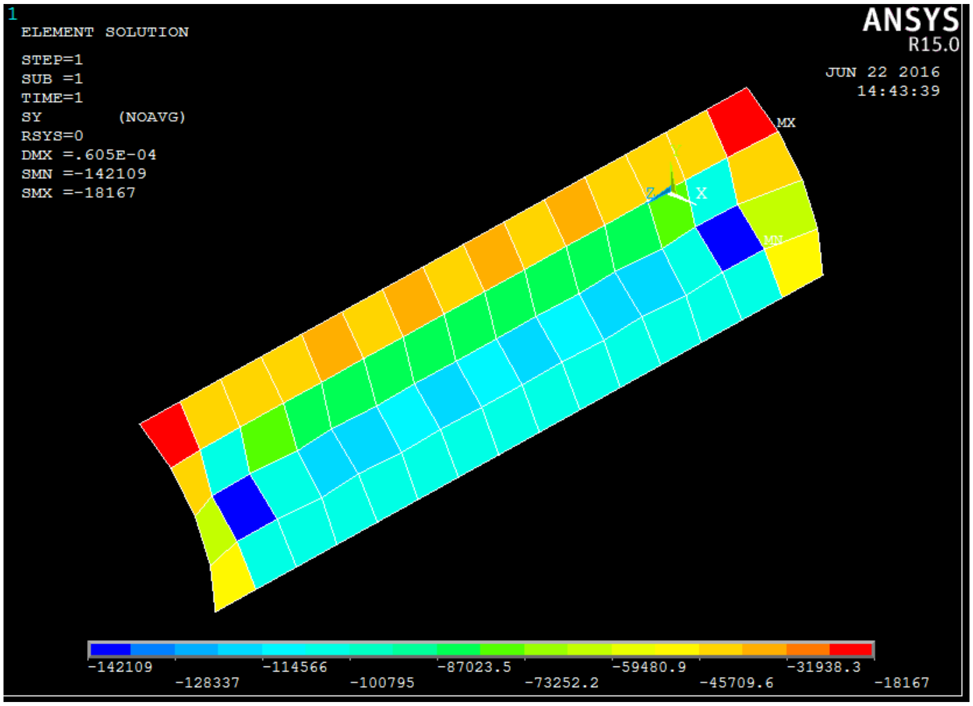


Fig: 4.8 Element Solution

The above analysis performed when end condition of column is fixed (DOF=0)

Case-II When All end condition of shell structure is not fixed.) (Translation is not allow and rotation is allow) at the same loading (-1000N/m^2) vertical pressure acting on the area at vertical direction the following results obtain.

F_X = -124707.7 N F_Y = -72000.00 N F_Z = 0
M_X = 864000.0 N-m M_Y = -1496492.N-m M_Z = 0

Minimum and Maximum Stresses:

```
MINIMUM VALUES
NODE         19          26            4           1           1           2
 VALUE   -82123.    -0.12825E+06 -59392.      7131.2     -34445.     -27563.

 MAXIMUM VALUES
NODE          1           2            4           8          21           6
 VALUE   -938.84     -20422.      5392.4      63015.      34445.      27563.
```

Minimum and maximum strain

```
MINIMUM VALUES
NODE          8          26            4           1           1           2
 VALUE  -0.23786E-05-0.47405E-05-0.12424E-05 0.73024E-06-0.35271E-05-0.28224E-
05

 MAXIMUM VALUES
NODE         37           2            4           8          21           6
 VALUE   0.17123E-05-0.19671E-06 0.11295E-05 0.64527E-05 0.35271E-05 0.28224E-
05
```

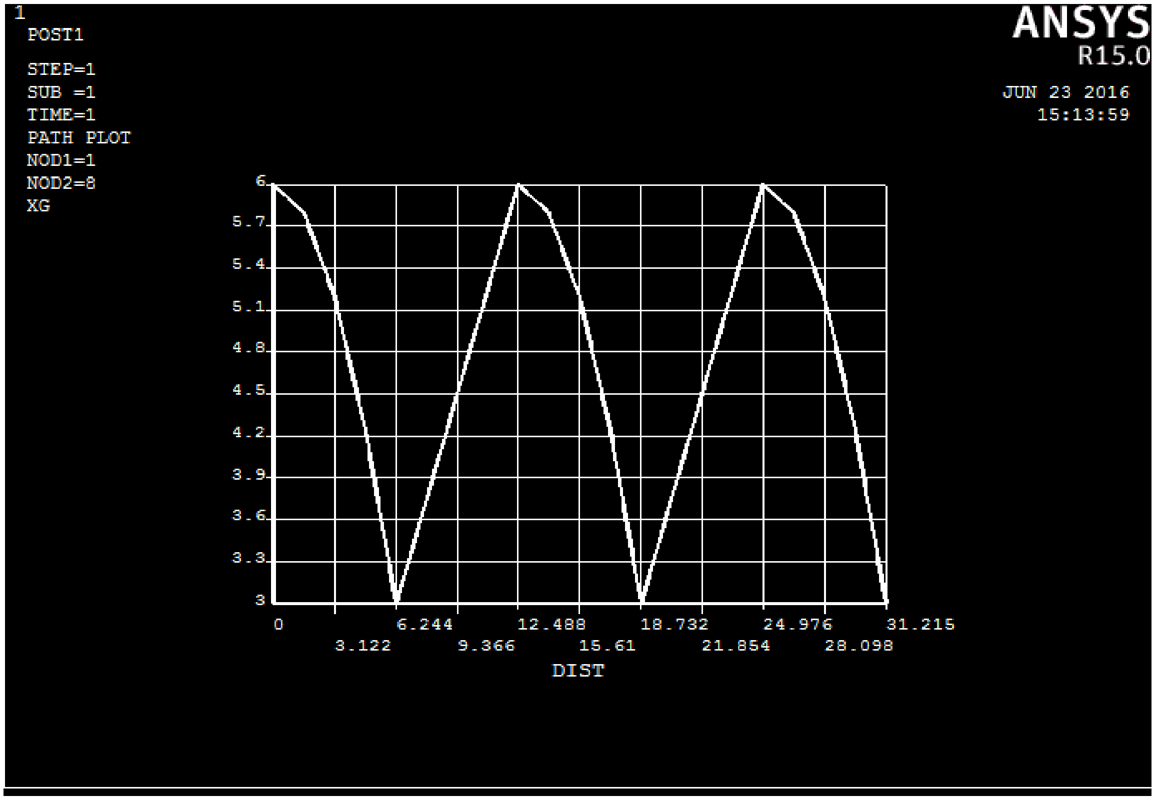


Fig: 4.9 Stress in Global X-direction

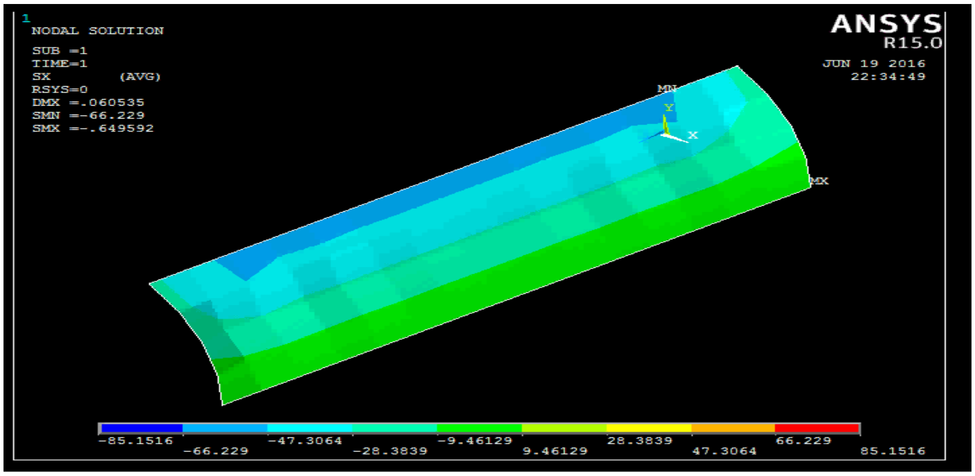


Fig: 4.10 Deform Shape of Shell

4.4 Mathematical Model of Concrete Structure by Membrane Method:

The membrane theory was igiven by designer with the assumptions that the shell is regarded as a perfectly flexible member of infinite element, carrying without indirect force in its plane only. Over a limited zone at sufficient distance away from the boundaries. The stresses in the small slab approach a distribution which is statically determinate and may be found by the membrane theory. This procedure is applicable to shell whose span to radius ratio is less than 0.5.

The equilibrium of the shell element as shown in fig 4.11 is examined under the given set of direction force with following notation

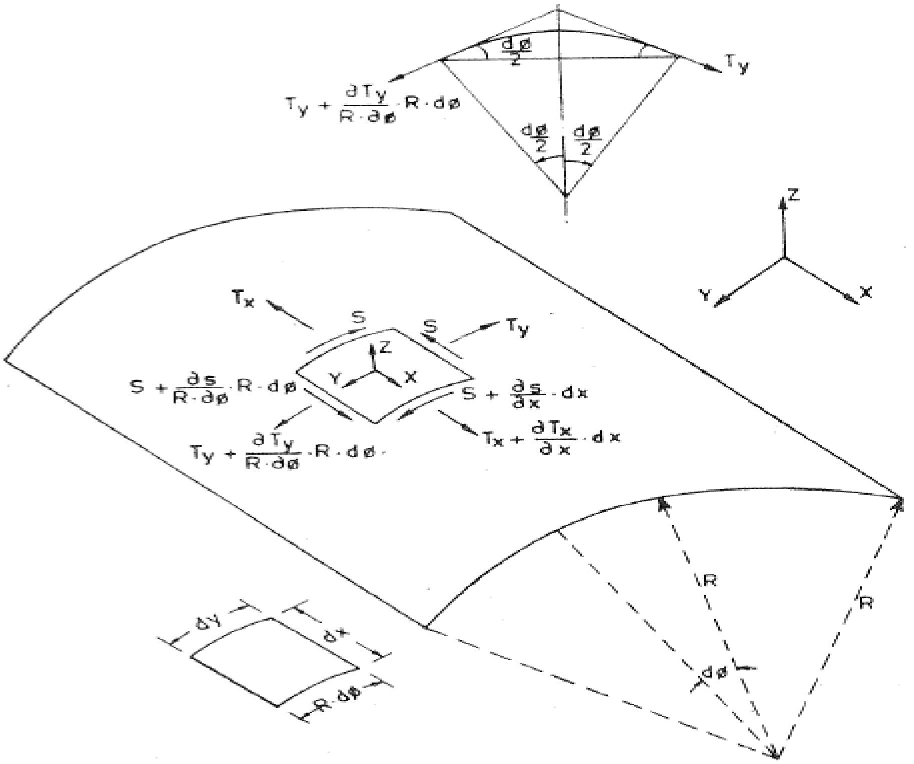


Fig: 4.11 Shell force component **(sorce R.c structure Krishna Raju)**

X-Direction of generatrix

Y-Direction of tangent to directrix at A

Z-Direction of the outward normal at A

Now, T_x, T_y, & S are the forces per unit length and R is Radius of the curve shell in X, Y, Z direction force component of external loads per unit length of element.

$$dy = R.d\phi$$

From the equation of equilibrium.

ƐX=0, ƐY=0, ƐZ=0…………………….. (i)

(a) Force in the direction of X

$$\frac{\partial Tx}{\partial x}\ .\ R.d\phi.dx+\frac{\partial s}{R.\partial\phi}\ .\ R.d\phi.dx+ xdx.R.d\phi. =0$$

$$\frac{\partial Tx}{\partial x}+\frac{\partial s}{R.\partial\phi}+x=0\ldots\ldots\ldots\ldots..\ (ii)$$

(b) Force in the y -direction

$\frac{\partial Ty}{R.d\phi}+\frac{\partial s}{\partial x}+y=0$…………….(iii)

(c) Force in z -direction

$2T_y.dxSin\frac{d\phi}{2}+\frac{\partial Ty}{R.d\phi}$ R.dϕdx $Sin\frac{d\phi}{2}$ –Z .dx R.dϕ=0…….(iv)

Since $Sin\frac{d\phi}{2}$ is very small so $Sin\frac{d\phi}{2} = \frac{d\phi}{2}$

So, T_y-R.z=0…………. (v)

Now the above model can analyze mathematical

Given Radius of Shell =6.0m, P= -1KN/m^2

Angle ϕ=60˚

Span 2L =24m, Thickness (t) 60mm

Force in X-direction (T_x) at x=0, ϕ=0

= [3P (L^2-x^2) ($\cos^2$ ϕ-Sin^2 ϕ)]/ 2R

T_x=36KN/m,Similarly T_y = 6KN/m, S, =0

At ϕ=60˚ the value of T_y =1.5KN/m, and maximum B.M =108KN-m maximum Stress=0.72N/mm^2

Now from Ansys

At x=0, ϕ=0˚

T_x =24.80KN, T_y = 3.855KN,S=0

At ϕ= 60˚ T_y 1.8 KN, B.M is 149.64KN-m and maximum stress is 0.8351N/mm^2

4.5 Buckling Analysis of Shell Structure

Above analyze a concrete shell structure with both degree of freedom. Now this concrete structure analyze for buckling. The buckling is a most important behavior for shell structure. The Eigen value buckling analysis give idea of the theoretical buckling and strength of an ideal elastic structure. However, in actual life, structural imperfections and nonlinearities prevent most actual world structures from approaching their Eigen value predicted buckling strength which means that this method is detriment the expected buckling loads. Ansys has the ability to formulate solutions for individual elements before putting them together to represent the complete problem. The solution of the problem basically depends on the element size. Ability to formulate solutions for individual elements before putting them together to represent the entire problem. The solution of the problem mainly depends on the element size.

Many others theoretically studies have been done in cylindrical structures by considering the cylinder as thin, and the variation of stress in the direction of thickness as negligible. This research project intends to combine the classic shell theories with the belonging numerical approach and this thesis primarily focused into, large thickness variation by considering cylinder as thin, slightly thick as well as thick by varying, span to depth and radius to thickness ratio. This thesis gives the critical buckling load of cylindrical shells subjected to external pressure. The Donell's relation for use basic stability analysis. The finite element modeling of the structure can perform in Ansys15.0 version The load carrying capacity of cylindrical structures can be increase by using stiffeners, the analysis in variation of stiffener geometry and orientation can be done by considering the cylinder in the form of, internally stiffened, externally stiffened and spiral stiffened.

4.5.1 DONELL'S FORM OF THE LINEAR EQUILIBRIUM EQUATION:

According to Donell’s principle of cylindrical shell which is supported on the two ends and applies a uniform pressure on the area of shell surface under the loading the buckling deformation of the shell is axisymmetric in the favor of uniformly axisymmetric bending is assumed. The buckling form can be defined as the lowest pressure at which the body decrees its stability and tends to buckle.

$$\frac{P_e a}{Eh} = \frac{\left[(\pi a / L)^2 + n^2\right]^2}{n^2} \frac{(h/a)^2}{12\left(1-\nu^2\right)} + \frac{(\pi a / L)^4}{n^2\left[(\pi a / L)^2 + n^2\right]^2}$$

Where

P_e= Critical buckling load in N/m^2

E =Young modulus of elasticity of cylindrical material

R=a= Radius of cylindrical shell in m

L= Longitudinal length of cylinder in m

H=t= Thickness of cylinder

n=wavelength parameter it is adopt trial and error method.

4.6 Buckling Load from Ansys:

Buckling is most important phenomena of cylindrical shell element. Now for the problem of shell structure The Radius of shell is 6.0m, Thickness is 60mm and angle is 60˚&pressure -1KN/m^2

Case-I when long edge of shell is fixed

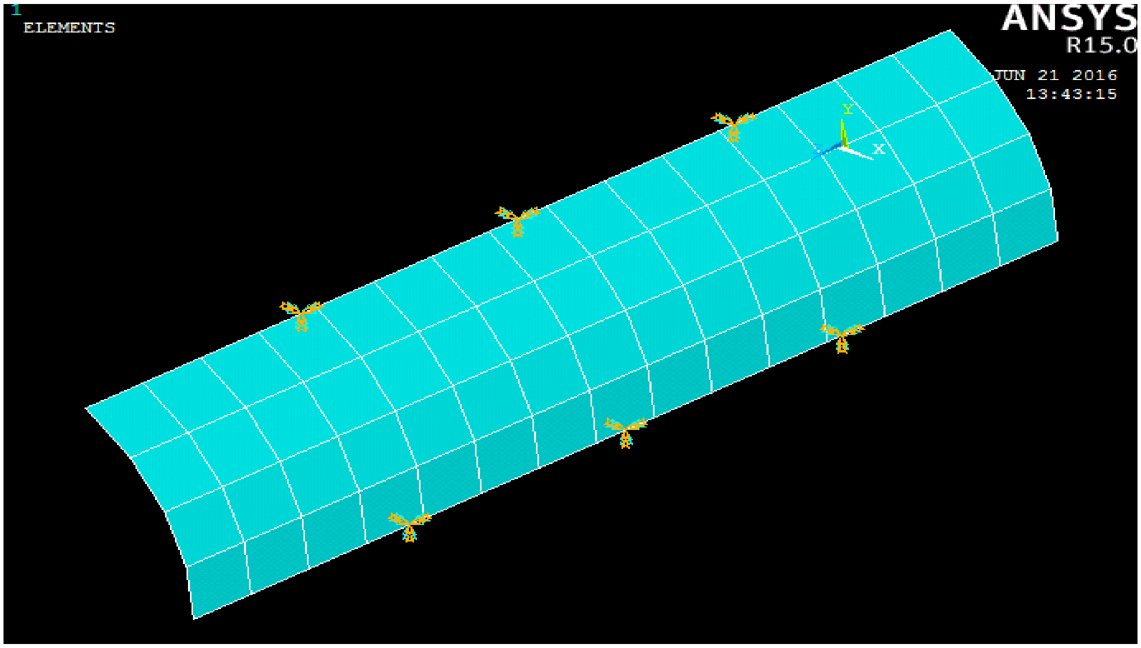


Fig: 4.12 Model of shell for buckling

Maximum buckling load 0.39254 KN and mathematically it is found 0.32552KN

Error 6.70%

4.6.1 Justification of Error

In the general shell theory, it was considered that flexural stress and membrane stress plays the same order of importance. If one of the above, flexural & membrane stresses are negligible in comparison with the other one, then it is possible to introduce appreciable simplifications in the general shell theory. If the flexural stress are negligible compared with the membrane stress, then these type of state of stress is called a membrane state of stress. The governing equation of the membrane theory can be determined directly from the general shell theory by neglecting the effect of bending; transverse and twisting shear effects and these types of stress state exist in thin shells. If to the contrary, the flexural stress is much high as compared to membrane stress, then such a type of stress is termed as pure flexural or moment state of stress.

4.6.2 Deformation Shape

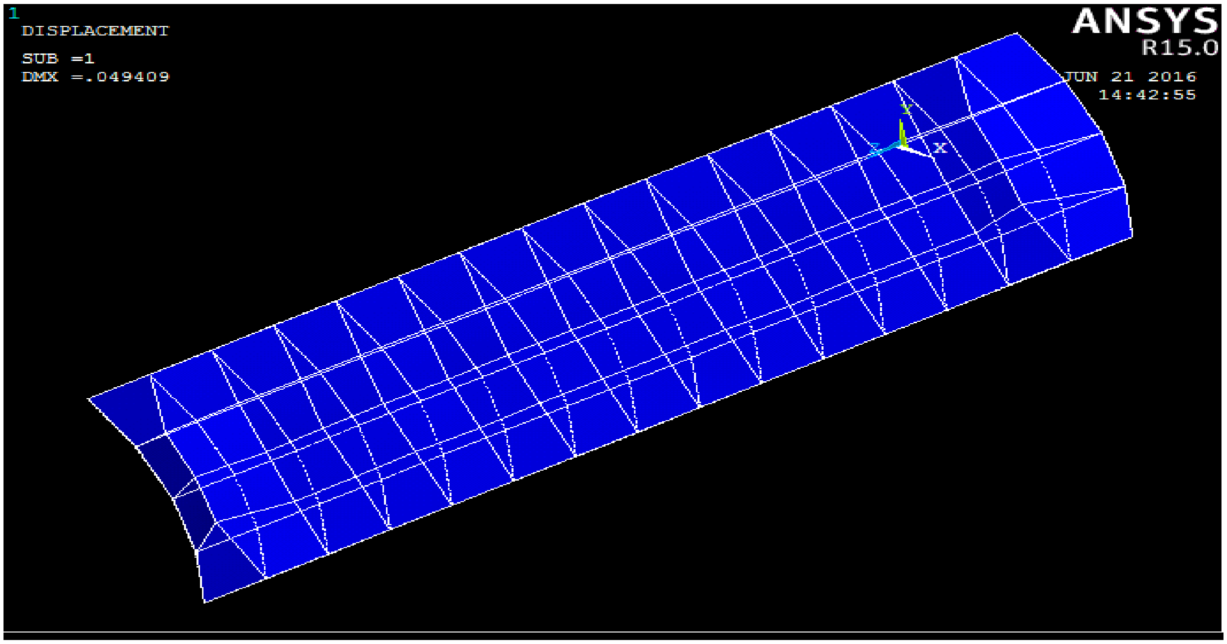


Fig: 4.13 Deformation Shape

The shape of deformation is shown above. In this deformation figure show that the boundary condition of long edge is fixed so, the deformation is occurs in short end of cylindrical shell structure

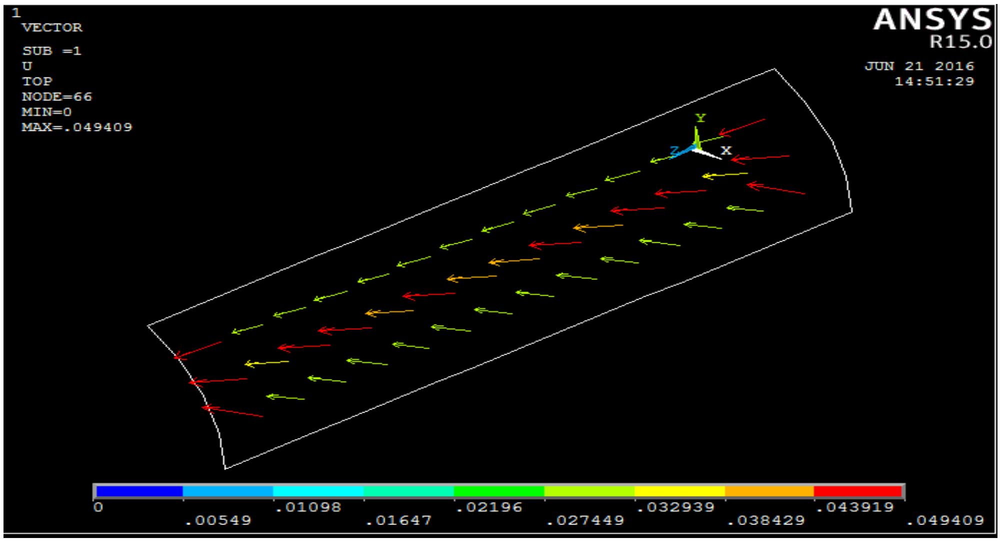


Fig: 4.14 Displacement form of cylindrical shell

In the figure show the maximum displacement is .049409m in the free direction of shell structure

Case-II When short edge of shell is fixed

For this case the buckling load is found 0.69326E-02 KN. The short edge is less effective as compared to long edge fixed. So short edge fixed is early fail as compared to long edge fixed shell

4.6.3 Deformation Shape:

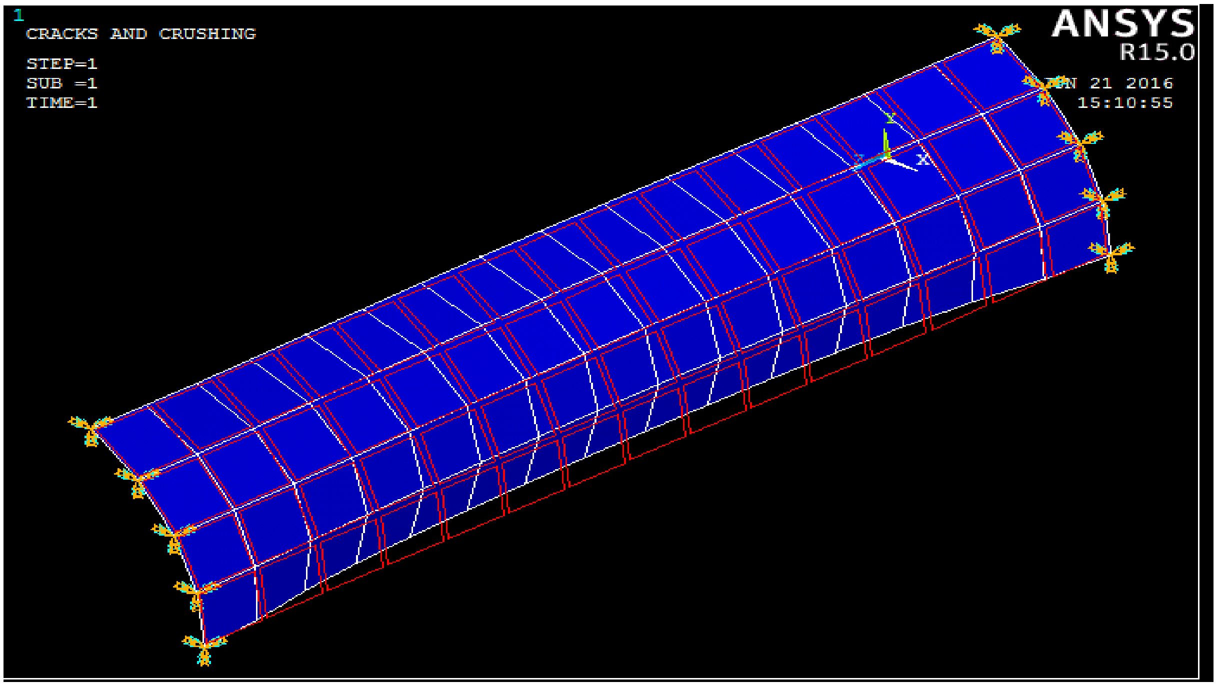


Fig: 4.15 Deformation Shape of shell

CHAPTER 5

RESULT

Analysis of cylindrical shell structure gives the various results of analysis but we require some important results. Experimentally on software the various nodal and member force found. The maximum value of nodal force -8487.1N in x-direction at node no.9. The value of force in x-direction is increase at edge beam. Similarly in Y-direction the value of nodal force is -9822.3N at node no. 27. This is also present near at edge beam.

The stress variation also shown in figure. In this graph stress v/s Node distance shows as symmetrical variation it is clear that at the edge beam the value of stress increase then decrees. This figure regularly going in symmetrical view.

- **Stress in Global x-Direction:**

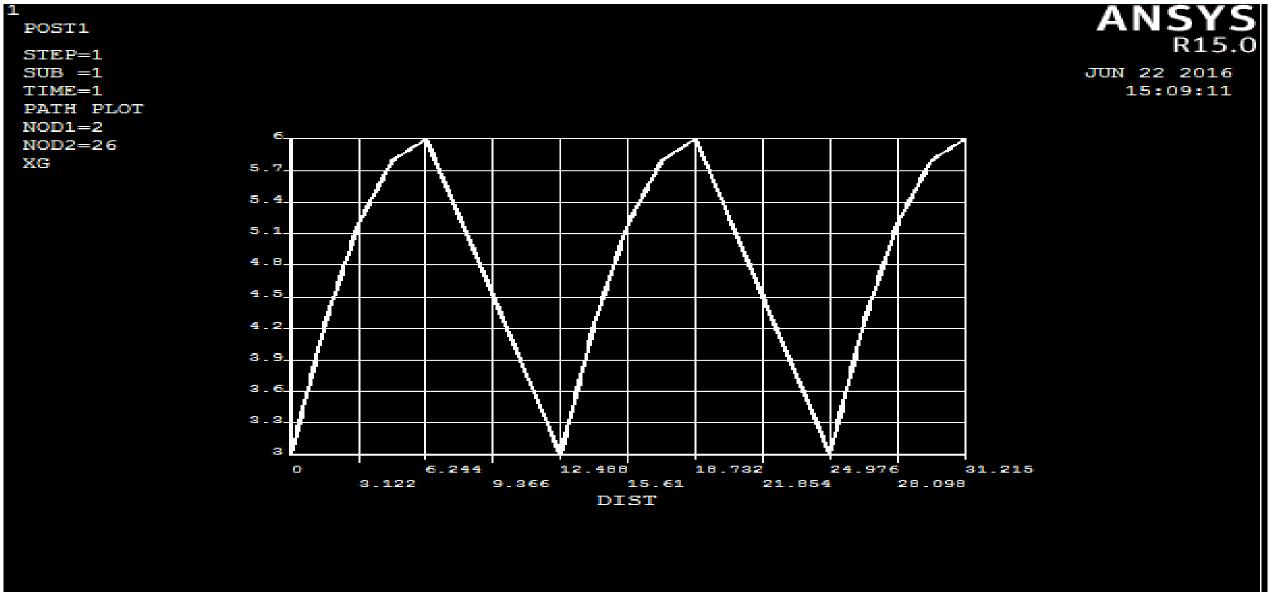


Fig: 5.1 Stress variation in X-Direction

- **Stress in Global Y-Direction:**

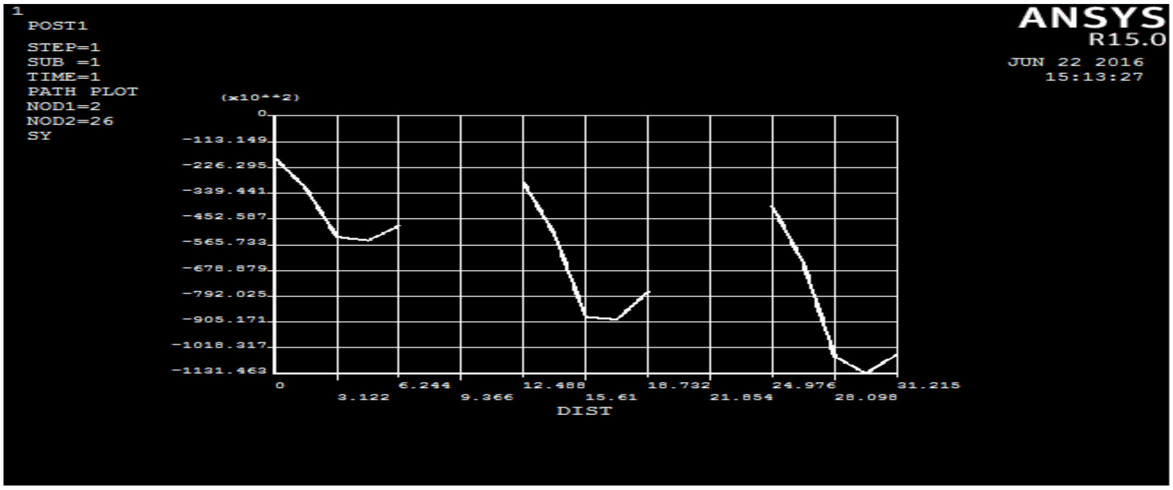


Fig: 5.2Axial stress (SY)

- **Strain in Global X-Direction:**

In the axial stress the variation is shown as discontinuous after a distance again take its pervious shape.

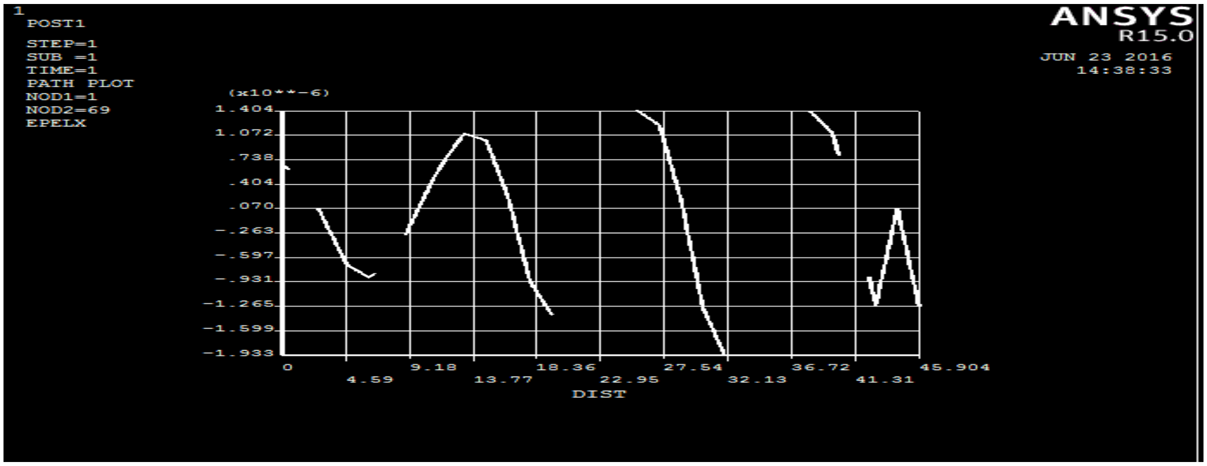


Fig: 5.3 Strain in X-direction

The variation in the strain is found in both x & y direction is unsymmetrical. It is clear the variation in strain with respect to node distance is increase at near the edge beam.

- **Strain in Global Y-Direction:**

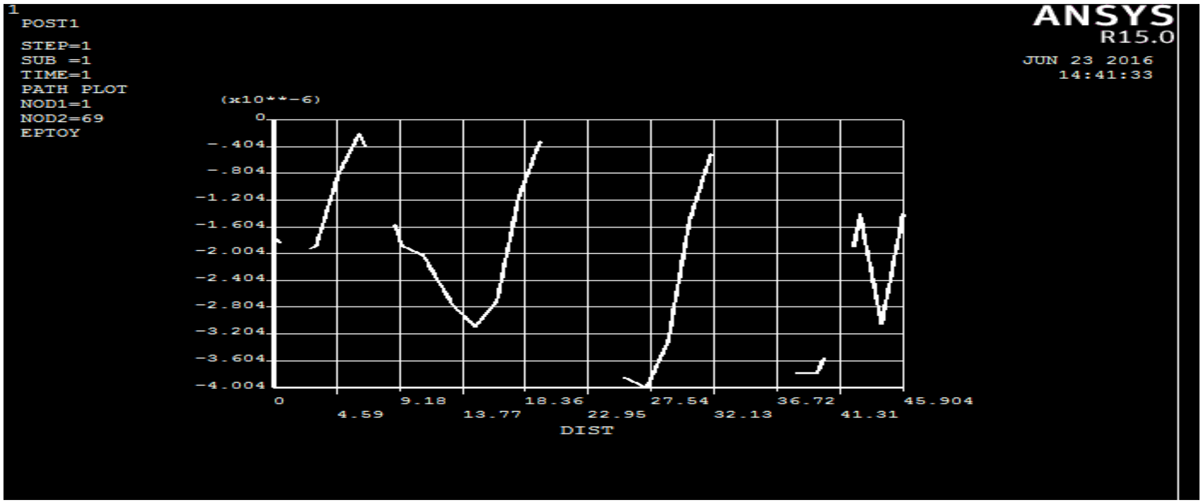


Fig: 5.4 Strain Y-direction

These above variation show when end is fixed.

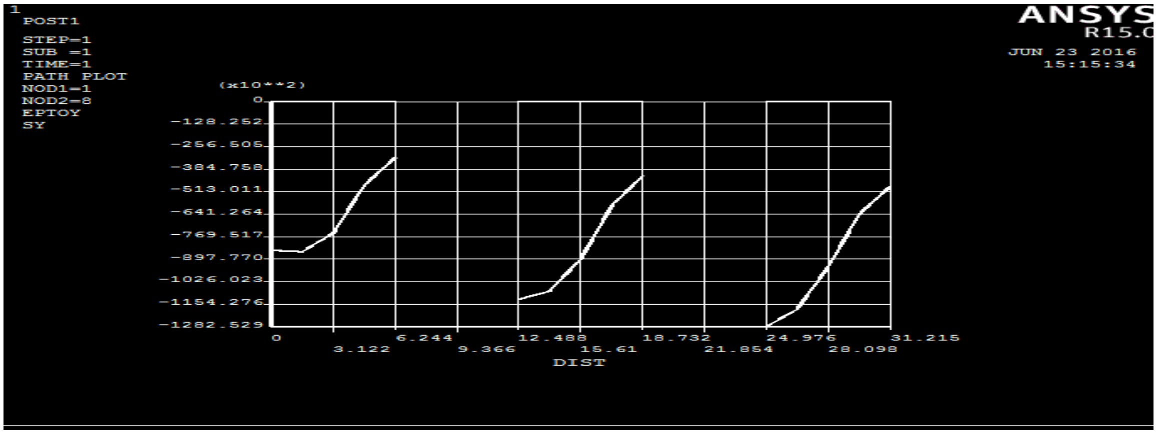


Fig: 5.5 Stress-Strain Y-direction

In the stress strain deformation is curvilinear at fixed interval it is shows that at a particular distance this follows path of strain–stress curve.

- **Buckling of cylindrical shell.**

We have discussed that buckling is an important phenomenon of shell structure analysis. In the buckling analysis we found that when beam edge is fixed the buckling load is increased and when short edge is fixed the buckling load is decreased. Approximate 2.56 times less access the value of buckling load. For the design purpose also buckling also considers. It is found that at the short at short edge the shell is not fissile to work.

CHAPTER 6

CONCLUSION

The primary goal of that our project to study the performance of cylindrical structures subjected to external pressure. As the external loads are carried by shell surface and bending responses, the variation in boundary condition can affect the load carrying capacity of engineering structure. This thesis primarily focused into, considerable thickness. Considering cylinder as support end change and boundary condition changed it observed that it is found from the analysis shows that, the Donnell's relation can be applied to thin shells more accurately and analysis by Ansys program is found correct Due to the increased bending or flexural effect in thick shells, the analysis shows some error value while comparing Donnell's relation with numerical solution.

From the static analysis by Ansys program is found point to point nodal solution and member solution of different forces and bending monuments this value also helpful for analysis the structure. The problem is also done exist solvers using time domain. Again the stresses are limiting to yield point indicating the structure has reach to its critical value.

From the static analysis explain basic theoretical considerations regarding the analysis of shells, and discuss how to make easy for the development of improved general finite elemental analysis procedures. The analysis of a finite element procedure that is general and optimistic for both static and buckling very difficult and mathematical analyses of available finite element schemes hardly exist. Therefore, it is danger to have presence appropriate quantities test problems, and use these in a judicious manner to examine the capability of a finite element process.

- **Future scope of this work**

  Work on Ansys gives the better idea about analysis of cylindrical shell. This is advance work other than manual work. After the analysis the standard code available and other foreign code available for compare. Load carrying capacity can increase by using increase elastic limit of structure.

- **Benefits of Ansys program**

The Ansys program is based on finite element method.

1. It is give more accurate value of different forces and moments.
2. This program also give more precision value
3. Different manual method give the total member force but Ansys give point to point solution
4. It is easy to handle.
5. Creative, believable and high quality production and processes
6. A seamless working exchange of data, regardless of location, industry, CAD environment, etc.

REFERENCES

- *B. Basset, On* “the extension and flexure of cylindrical and spherical thin shells”, *Phil. Trans. Royal Soc. London, Series A, 181, 433-480, 1890.*
- *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.*
- *Burcher, R. & Rydill, L. (1994)* “Concepts in submarine design.” *Cambridge ocean technology series. Cambridge University Press.*
- *C. Mandala, H. Qinb,*, B.C. Vemuric:* “A novel FEM-based dynamic framework for subdivision surfaces”
- *D. Roylance*, “Finite Element Analysis,” *tech. rep., Department of Materials Science and Engineering,* Massachusetts Institue of Technology, 2001.
- *“Design Optimization of a Pressure Hull for Dynamic Loading”.* International Journal of Engineering Science and Innovative Technology (IJESIT) Volume 2, Issue 6, November 2013
- *“Finite Element Procedures” –* Klaus-Jurgen Bathe, Prentice Hall of India Pvt. Ltd.- Sixth Edition 2002
- *International journal of engineering science and innovative technology volume2,issue 6,november 2013 ISSN:2319-5967*
- *K.R.SHIVADAS*:- “pre buckling analysis of shells subjected to combined loads”
- *Krishna raju:* “Design of Rc structure”
- *L. F. MORI*, “Deformation and fracture modes of sandwich structures subjected to underwater impulsive loads”
- *Lord Rayleigh,* “on the infinitesimal bending of surfaces of revolution”, *Proc. London Math. Soc., 13, 4–16, 1881. Also The Theory of Sound, Vol. 1, Ch. 10, MacMillan, London, 1884.*
- M. Millais, *Building Structures: From Concepts to Design*. Taylor and Francis, 2005.
- *Rinu Cherian,* “Buckling Analysis of Underwater Cylindrical Shells Subjected to External Pressure”.

- *T. Subramani*, “Finite Element Analysis of Thin Walled-Shell Structures by ANSYS”
- *Zorin D*. “Smoothness of stationary subdivision on irregular meshes, Constructive Approximation”, *submitted for publication, available as Stanford Computer Science Lab Tech. Rep. CSL-TR-98-752, 1998.*