BUCKLING AND POST BUCKLING OF STRUCTURAL COMPONENTS

by

SHRUTI DESHPANDE

Presented to the Faculty of the Graduate School of
The University of Texas at Arlington in Partial Fulfillment
of the Requirements
for the Degree of

MASTER OF SCIENCE IN MECHANICAL ENGINEERING

THE UNIVERSITY OF TEXAS AT ARLINGTON
December 2010

ACKNOWLEDGEMENTS

I am sincerely grateful to my thesis advisor Dr. Kent Lawrence, for his valuable guidance and support throughout this thesis. He has consistently helped me improve and supported me to gain my accomplishments. The requirements of this thesis were achieved in a timely and organized manner under his guidance. Without him the entire work would be a reckless venture.

I would like to thank Dr. B. P. Wang and Dr. Seiichi Nomura for taking the time to serve on my thesis committee.

I am indebted to my family and especially to my husband Chinmay for his continuous support and encouragement in all my endeavors. My Master's degree wouldn't have been possible without him. I also thank all my friends, especially my roommates Athena and Rasika who were a great source of encouragement.

November 23, 2010

ABSTRACT

BUCKLING AND POST BUCKLING OF STRUCTURAL COMPONENTS

Shruti Deshpande, M. S.

The University of Texas at Arlington, 2010

Supervising Professor: Kent Lawrence

Structural members and components, such as shells and trusses of different geometries, form the intricate and deep seated parts in the manufacturing of missiles submarines, rockets, airplanes, automobiles etc and find applications in civil structures such as bridges. These members are comprised of components like conical frusta, cylindrical panels and shallow trusses. In many cases, the sole purpose of such shell structures is to absorb the energy generated due to impact which means, that these structures are subjected to heavy loads and can experience failure due to buckling.

The objective of this work is to study the buckling and post buckling behavior of such members. The study is carried out using finite element analysis. The widely implemented softwares ANSYS APDL and ANSYS Workbench are used to perform the analysis.

The components analyzed consist of shell structures such as conical frusta and cylindrical panels, and other structures like the shallow truss, diagonal truss and the shallow arch. These structures are analyzed for their buckling and post buckling behavior when subject to specific loading conditions and geometric, contact and material non-linearity. The results compare favorably with known solutions.

TABLE OF CONTENTS

LIST OF ILLUSTRATIONS

Figure Page

LIST OF TABLES

Table Page

CHAPTER 1

INTRODUCTION

The current manufacturing industry emphasizes on the use of structural members, which encompass light weight with high strength and are expected to absorb high energy and carry heavy loads. These members are comprised of Shell structures and shallow trusses.

The shell structures have a significant advantage in regards to load carrying capacity. When load is applied to a shell it results into different types of internal forces which can cause bending, twisting, transverse shearing and buckling. Commonly used components are cylindrical panels, cylindrical and spherical shells and conical frusta. In many cases, these shell structures are the deep seated structures in manufacturing submarines, missiles, tanks and their roofs and fluid reservoirs. These components act as energy absorbers in most of these cases.

These energy absorbing devices may be classified on the basis of the following principles,

- extrusion
- friction
- material deformation

Shell structures such as cylindrical tubes and conical frusta are the most frequently used devices that absorb energy by material deformation. Such axis-symmetric shell structures are widely implemented in aircraft and spacecraft structures, nuclear reactors, storage tanks for bulk solids and liquids, pressure vessels, pipelines etc.. Conical shells also find applications in marine engineering. These are mainly implemented as transition elements beween cylinders of different diameters. Shell structures also find versatile applications in the field of civil engineering. Some examples that have thin shells as structural components are silos, roofs, container, tanks, pipes, pressure vessels, submarines and aircraft wings.

Structural Members such as Shallow Truss and arch find many applications in the field of civil engineering. These form an important part of various structures like air plane hangars, bridges, railways, roof structures etc. Arches are used for underground structures such as drains and vaults and are widely used structural members in bridges and buildings too. The arch can carry a much greater load thus making it more significant.

Buckling of these kind of structures is dependent on various number of variables like geometry, material properties and the applied load. The Finite element methods (FEM) technique is extensively used to analyze the buckling and post bucking behavior of such structural components. Several finite element softwares like ADINA, NASTRAN, ANSYS and ABAQUS are implemented for buckling and post bucking analysis.

Several papers that study the theories of the collapse of shell and truss structures have appeared in the literature. The buckling analysis of such structures using finite element methods occurs in the literature.

1.2 Scope of Study

The focus of this thesis is the study of buckling and post buckling of structural components using ANSYS APDL and ANSYS Workbench. The thesis will discuss a number of components and structural members such as The Shallow Arch, Shallow Truss, Diagonal Truss, Cylindrical Panel and Conical Frusta.

The entire work area comprised of a number of chapters. For conducting the analysis an intense literature review on the buckling analysis of thin shells like cylindrical shells, conical shells, cylindrical panels and trusses is conducted, which is briefed in the second chapter. A brief review of the buckling and post buckling concept is presented in the third chapter. The fourth chapter details about the analysis of Shallow truss, Diagonal truss and the shallow arch for buckling and post buckling behavior using a finite element methods software, ANSYS APDL. The fifth chapter details about the buckling and post buckling analysis of the shells, such as the cylindrical panel and the conical frustum using the finite element methods software ANSYS Workbench. A study of Eigen value buckling for all the cases is also included in the fifth chapter. This is followed by conclusion and recommendation in the sixth chapter and then the references used for the entire achievement.

CHAPTER 2

LITERATURE REVIEW

S Aghajari, K Abedi and H Showkati have studied the Buckling and post-buckling behavior of thin walled cylindrical steel shells with varying thickness subjected to uniform external pressure [1].

For the experimental study, four different cylindrical shell specimens with varying thickness were tested to collapse. To study the buckling and post buckling of these structures using ANSYS which is a finite element modeling software, material and geometric non linearities were considered in the analysis performed. Shell elements with large deflection, stress stiffening and non linear capabilities were used to model the cylindrical shells. From obtained results, it was observed that the buckling load obtained using ANSYS was higher than that obtained experimentally, though the percentage difference was less.

Thus, the study shows that the numerical behavior predicted by the non linear finite element collapse analysis is close to the experimental results. Consequently, finite element modeling is found to be reliable enough to be used to perform non linear analysis for the study of buckling and post buckling behavior.

N. K. Gupta, N. Mohamed Sheriff, R. Velmurugan have studied the buckling of thin conical frustum under axial loads [2]. They performed experiments with certain specimens of conical frusta. The non1 linear material used for these specimens was Aluminum. Experiments were performed by subjecting the conical shells to quasi static loading. Axial compression of the shells was carried out by compressing each specimen between two rigid platens. The load deflection curves were obtained from these experiments were compared with the results obtained from numerical analysis.

The compressive failure mode was simulated using ANSYS. Material, geometric and contact non linearity were included. The material non-linearity was included using the actual stress strain curve obtained from the experiments. The results thus obtained were higher but could be compared well with the experimental ones. Using finite elements modeling thus facilitates the analysis of intermediate stages of buckling which reduces the cost and time.

A.Pica and R. D. Wood have studied the post buckling behavior of plates and shells using a mindlin shallow shell formulation[3]. This paper presents a geometrically non-linear analysis of shallow shells using finite element mindlin formulation. It gives results for post buckling behavior of square and circular plates subject to direct in plane loading and square plate subject to in plane shear loading. Analysis of shallow truss and cylindrical and spherical shells are also presented, all exhibiting snap through behavior. For a number of post buckling solutions the 9 node lagrangian element was used which demonstrates the ability to model curved boundaries.

Chawalit Thinvongpituk and Pisit Techarungpaisarn have studied the Buckling of Axially compressed conical shells of linearly variable thickness using structural model[4]. The study was conducted with a series of experiments performed using conical specimens with constant thickness, which were crushed till the buckling load was recorded. For comparison FE model was constructed using ABAQUS to simulate the experiment. The buckling loads obtained from the experiment and the FE model were observed to be in good agreement with each other. The FE model was further used to investigate the cone with non constant thickness.

It was observed that variation of thickness in axial direction results in the reduction of buckling load. The reduction of buckling load, due to the small thickness variation in axial direction is proportional to the thickness variation parameter ε, where ε is the ratio of the difference of minimum and maximum wall thickness to the minimum wall thickness of cone.

Huu Tai Thai and Seung Eock Kim have performed the Inelastic post buckling analysis of space steel trusses using the generalized displacement control method[5]. Space steel trusses used extensively for domes or roofs. In this paper the authors have extended the application of Hill [11] model for inelastic post buckling analysis of space steel trusses. Geometric and material non linearity are considered for the study.

This paper presents an algorithm that can trace the equilibrium paths of the non linear problem with multiple limit points and snap back points. A computer program is developed to predict the inelastic post buckling behavior of space truss structures. The paper includes a number of examples solved to prove the accuracy of the proposed procedure.

A.B. Sabir and A. C. Lock have studied the application of Finite Element Methods to the Large Deflection Geometrically Non Linear behavior of Cylindrical Shells[9]. The paper presents a method that is capable of dealing with the study of post buckling of structures subjected to in plane and lateral loading. It employs linearized incremental method and Newton Raphson method.

The algorithm based on the linearized incremental and Newton Raphson methods, has been used on cylindrical shells subjected to lateral loading and a flat plate subjected to in plane loading. A family of shells of similar dimensions but varying thickness has been examined and load deflection curves exhibiting complicated behavior are obtained.

H Ramsey has studied the plastic buckling of conical shells under axial compression [6]. This research paper presents analysis and experimental results of plastic axisymmetric buckling of steep, truncated conical shells subjected to axial compression. For this, specimens of 6061-T6 aluminum and stainless steel were tested by subjecting to compressive load. In spite of the remarkable difference in the stress-strain curve for these materials, same buckling modes - were observed for the geometrically identical cones. The analysis uses perturbation method, wherein no minimum load is required for initiation of buckling, but, the plastic zone must be sufficiently developed for the formation of preferred buckling mode. The analysis method described does not predict the maximum load, but gives good description of the buckling deformation.

N K. Gupta and H. Abbas have studied the axisymmetric axial crushing of thin frusta[7]. It is normally observed in frusta with semi-epical angles <10 degrees. The axisymmetric crushing of thin walled Aluminum frusta is presented which considers total outside straight folds considering the variation of circumferential strain during the formation of convolution. A mathematical model for the calculation of variation of crushing load is developed. Experiments were performed by subjecting Aluminum frusta to axial compression. During experiments it was observed that the limb of the first fold is straight as compared to other folds thus indicating less absorption of flexural energy in it. The fold size and crushing load are in good agreement with the experimental results.

A Spagnoli, M. K. Chryssanthopoulos have studied the Elastic buckling and post buckling behavior of widely stiffened conical shells under axial compression[8]. In this paper, the linear and non-linear elastic buckling response of the conical panel is studied for a wide range of shell and stiffening parameters by means of an appropriate finite element model. Linear analysis is used for the determination of classical buckling load. The non-linear analysis of imperfect conical shells is used for the study of imperfection sensitivity. The FE package ABAQUS is used for modeling and analysis.

B S Golzan and H. Showkati have studied the buckling of thin walled conical shells under uniform external pressure[10]. This paper presents a detail study of the facility developed for the buckling experiments on conical shells. The experiments were performed using six test specimens which were loaded by applying external pressure, and the respective buckling load and buckling modes were studied. The experimental buckling loads obtained were compared with the ones derived from FEA and Jawad equation.The experimental buckling load is observed to be lower that obtained from FEA and the Jawad equation.

H. A. Mang, G Hofinger and X. Jia have studied the Sensitivity Analysis of the Post Buckling Behavior of Elastic Structures[14]. This paper presents a study that focuses on the sensitivity analysis and the conversion of imperfection sensitive structures to imperfection insensitive structures. Several examples are presented that focus on special cases of buckling.

CHAPTER 3

BUCKLING AND POST BUCKLING

3.1 Buckling

Buckling is that mode of failure when the structure experiences sudden failure when subjected to compressive stress. When a slender structure is loaded in compression, for small loads it deforms with hardly any noticeable change in the geometry and load carrying capacity. At the point of critical load value, the structure suddenly experiences a large deformation and may lose its ability to carry load. This stage is the buckling stage.

The critical buckling load for a pin-pinned column is given by the formula

$$P_{cr} = \frac{\pi^2 EI}{l^2} \qquad \text{Eq. 3.1}$$

The structural instability of buckling can be categorized as

Bifurcation buckling

Limit load buckling

In Bifurcation buckling the deflection when subjected to compressive load, changes from one direction to a different one. The load at which bifurcation occurs is the Critical Buckling Load. The deflection path that occurs prior to the bifurcation is called as the Primary Path and that after bifurcation is called as secondary or post buckling path.

There are two types of buckling failure problems in thin shells: Axis-symmetric problems and non axis-symmetric problems. Axis-symmetric problems refer to problems in thin axis-symmetric shells subjected to axis-symmetric load. For these structures instability may occur in the form of axis-symmetric snap through, or non axis-symmetric bifurcation.

3.2 Post Buckling

Post buckling stage is a continuation of the buckling stage. After the load reaches its critical value the load value may not change or it may start decreasing, while deformation continues to increase. In some cases the structure continues to take more load after certain amount of deformation, to continue increasing deformation which eventually results in a second buckling cycle. Post buckling analysis being non-linear, we obtain far more information than we obtain from linear Eigen-value analysis.

The nonlinear load displacement relationship, which can be a result of the stress strain relationship with a nonlinear function of stress, strain and/or time; the changes in geometry due to large displacements; irreversible structural behavior upon removal of external loads; change in the boundary conditions such as change in the contact area and the influence of loading sequence on the behavior of the structure, requires a nonlinear structural analysis.

The structural nonlinearities can be classified as, a geometric nonlinearity, a material nonlinearity and a contact or a boundary nonlinearity.

3.2.1 Geometric Non linearity

Geometric nonlinearity arises from the presence of large strain, small strain and/or rotations and loss of structural stability. Large strains may occur in rubber structures and metal forming. Slender structures such as bars and thin plates may experience large displacements and rotations with small strains. Pre-stressed structures with small strains and displacements may undergo a loss of stability by buckling.

Geometric non-linearity can be classified in two types,

Large Deflection and rotation: If the structure undergoes large displacements compared to its smallest dimension and rotations to such an extent that its original dimensions and position, as well as the loading direction, change significantly, the large deflection ad rotation analysis comes becomes necessary.

Stress Stiffening: When the stress in one direction affects the stiffness in other direction stress stiffening occurs. A structure, having little or no stiffness in compression, but having considerable stiffness in tension exhibits this behavior.

3.2.2 Material Non linearity

Material nonlinearities arise from the presence of time independent behavior such as plasticity, time dependent behavior such as creep and viscoelastic / viscoplastic behavior where both plasticity and creep effects occur simultaneously. These may result in irreversible structural behavior. Non-linear behavior in ANSYS is characterized as,

Plasticity- permanent time independent deformation.

Creep: Permanent, time dependent deformation.

Non-linear elastic: non linear stress strain curve, structure returns to original state on unloading, no permanent deformation.

Viscoelasticity: Time dependent deformation under constant load. Structure Returns to original state upon unloading.

Hyper elasticity: Rubber - like materials.

3.2.3 Contact Non linearity

Nonlinearity due to contact conditions arises because the prescribed displacements on the boundary depend on the deformation of the structure.

This paper focuses on the study of buckling and post buckling of various structural members using Finite element software ANSYS APDL and ANSYS Workbench 12.1.

3.3 Finite Element method and ANSYS

The finite element method (FEM) is a numerical technique for finding approximate solutions of partial differential equations (PDE) as well as of integral equations. The solution approach is based either on eliminating the differential equation completely (steady state problems), or rendering the PDE into an approximating system of ordinary differential equations, which are then numerically integrated.

Finite Element Analysis (FEA) is a computer simulation technique used in engineering analysis by using the numerical technique of finite element method (FEM). One mechanical engineering software widely used for such analysis is ANSYS.

ANSYS structural mechanics offers wide range of analysis from concept simulation to advanced analysis. With a full complement of linear and nonlinear elements, material laws ranging from metal to rubber and the most comprehensive set of solvers available, ANSYS simulation tools are applied widely by users across industries. Additionally, the adaptive architecture of ANSYS software tools provides one with the flexibility for customization and interoperability with other tools such as third-party software.

CHAPTER 4

TRUSS AND ARCH STRUCTURES

Trusses find wide range of applications in structures such as bridges, roof structures, railways, etc. The arch is significant because when subjected to vertical loads, its two ends develop reactions inwardly in horizontal direction. The arch can carry a much greater load than a flat beam of the same size and material, because downward pressure forces the elements together instead of apart. Thus, these structures are expected to sustain heavy loads.

The Finite element analysis is used for the study of post buckling behavior of these structural components. The method is implemented by the use of softwares like ANSYS APDL and ANSYS Workbench.

4.1 Investigation Process

In order to analyze the structural components using ANSYS the following process is proposed:

The first step is to create a simplified model of the structure to be analyzed;

The second step is to apply material properties and the element type and create a mesh of appropriate density;

The third step is to apply defined boundary conditions and set up the non-linear solution controls and solve the problem to give the post buckling results.

4.2 ANSYS Simulation

This work includes the buckling and post buckling analysis of several structural components. These include Shallow Truss, Shallow Arch, Diagonal Truss, Cylindrical Panel and Conical Frustum.

4.2.1 Shallow Truss

A stable truss design exists when deformations increase as the applied load increases; an unstable design occurs when deformations increase as the load decreases. To understand this type of structural response, the equilibrium paths of key connection points are plotted. The shallow truss geometry prompts snap-through buckling of many members rather than local buckling of a single member.

The Shallow Truss is analyzed here by subjecting it to two different sets of Boundary conditions;

- Clamped
- Hinged

4.2.1.1 ANSYS Simulation

a) Geometry: The Shallow Truss geometry is shown in the following Figure 4.1

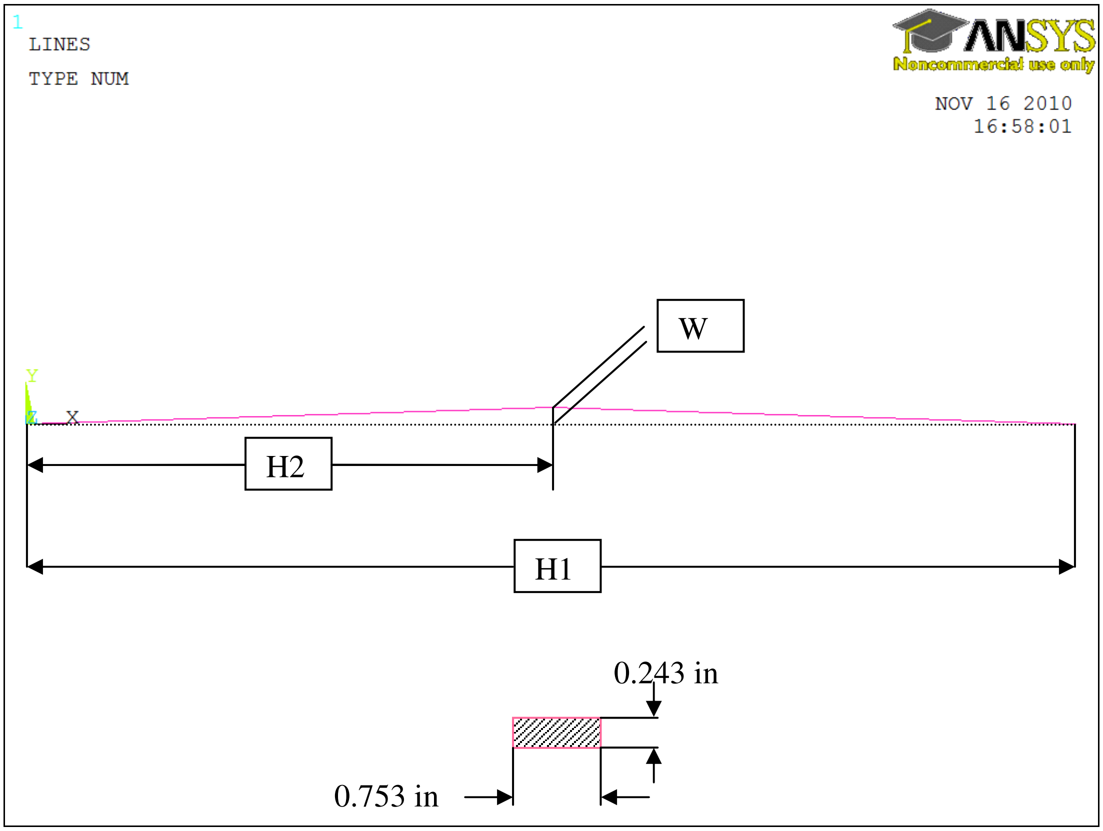


Figure 4.1 Shallow Truss geometry

The dimensions of the Truss are as follows,

Table 4.1 Dimensions of Shallow Truss

H1	25.886 in
H2	12.943 in
W1	0.386 In

b) Material Properties

ANSYS offers a wide range of material properties to be used for a particular analysis. For the truss, the material is assumed to be homogenous linear isotropic, hence the only properties needed are;

Table 4.2 Material Properties for Shallow Truss

Young's Modulus	10.3×10^6 lb/in^2

c) Boundary Conditions

- Clamped

For the clamped truss, the mesh is formed by using BEAM3 elements. Beam3 is a uniaxial element with tension, compression, and bending capabilities [18].

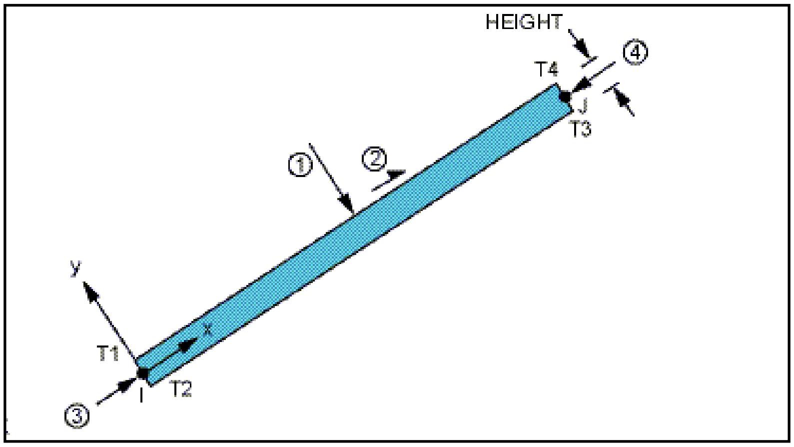


Figure 4.2 BEAM3 element

The element has three degrees of freedom at each node: translations in the element X and Y directions and rotation about the element z-axis. The element is defined by two nodes, the cross-sectional area, the area moment of inertia, the height, and the material properties and has special features like large deflection [13].

The truss is analyzed by discretizing the domain into different number of elements and the mesh that gave a satisfactory solution is implemented. After a number of trials the appropriate number of elements found to give favorable result is 6.

For constraints, the structure is fixed in all degrees of freedom at both ends and a displacement of 0.9 in is applied in the negative Y direction at the center node.

When a displacement is applied at the central node of the truss, the node is pushed in the downward direction creating a reaction force. This reaction force represents the force that will be needed to achieve the particular amount of displacement.

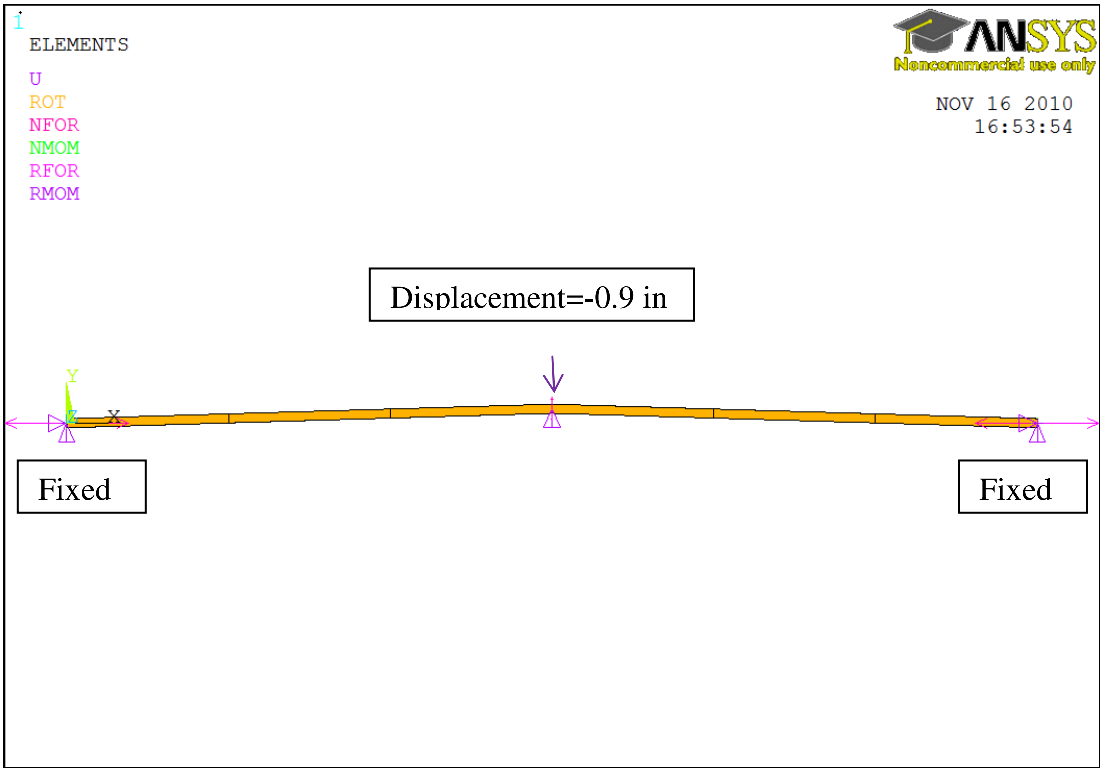


Figure 4.3 Clamped Truss subjected to Boundary conditions

- Hinged

For the truss with hinged boundary conditions, element type used for the mesh is LINK1.

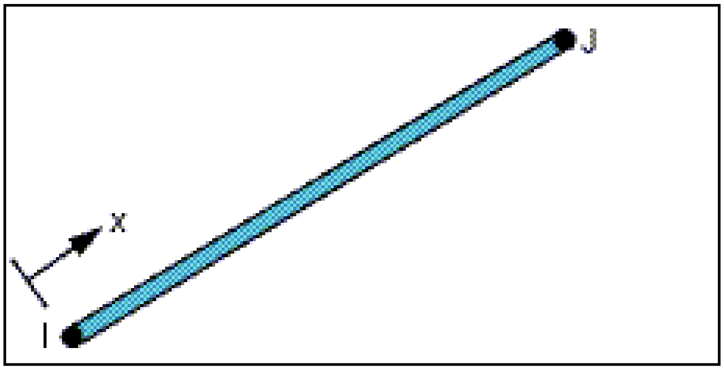


Figure 4.4 LINK1 element

The element is defined by two nodes, the cross-sectional area, an initial strain, and the material properties with features like plasticity, large deflection, birth and death etc. As in a pin-jointed structure, no bending of the element is considered [13].

In this case, the truss is hinged at both ends i.e. displacements in X and Y directions are fixed and a displacement of 0.9 in the negative Y direction is applied at the central node.

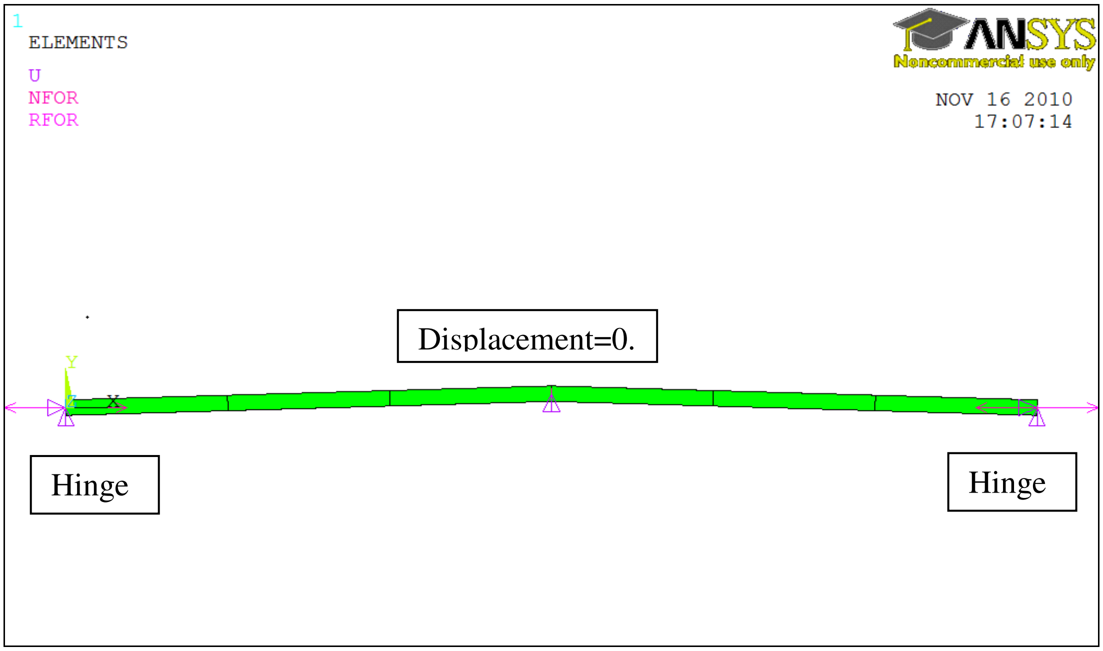


Figure 4.5 Truss subjected to hinged boundary conditions

4.2.1.2 Results

The equilibrium plots of the above analyzed structures are plotted in figure 4.7 and 4.10. The nonlinear instability region is the region in which “snap-through” occurs and in which the equilibrium path goes from one stable point (a) to another new stable point (b). The nonlinear behavior places the critical limit load at point (a) equal to that at point (b), but the load limit corresponds to a new structural shape. In case of bifurcation buckling at bifurcation point the structure immediately becomes unstable and buckles. The member is unable to support any additional load, which is not the case for nonlinear snap-through buckling.

The above cases are analyzed in ANSYS APDL. After subsequent trials a mesh of 6 elements is used.

a) Clamped Shallow Truss

For the clamped shallow truss, the resulting deformation is shown in figure 4.6.

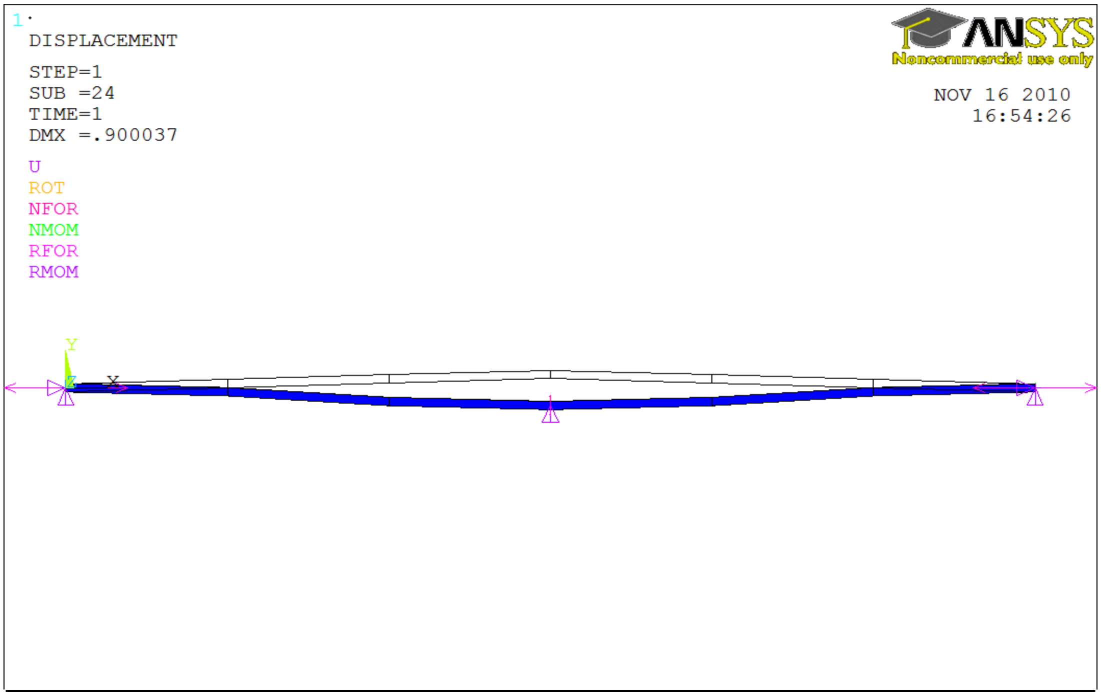


Figure 4.6 Deformed clamped truss

The graph of vertical load reaction at the central node against the displacement is plotted in the figure 4.7. The results match with the results in [3]. Vertical load reaction is the force reaction that is evaluated at the point where displacement is applied. The analytical limit point for the truss is obtained at a load value of 38 lb and deflection of 0.22 in. In reference [3] the limit point occurs at 33.3 lb and deflection of 0.2 in.

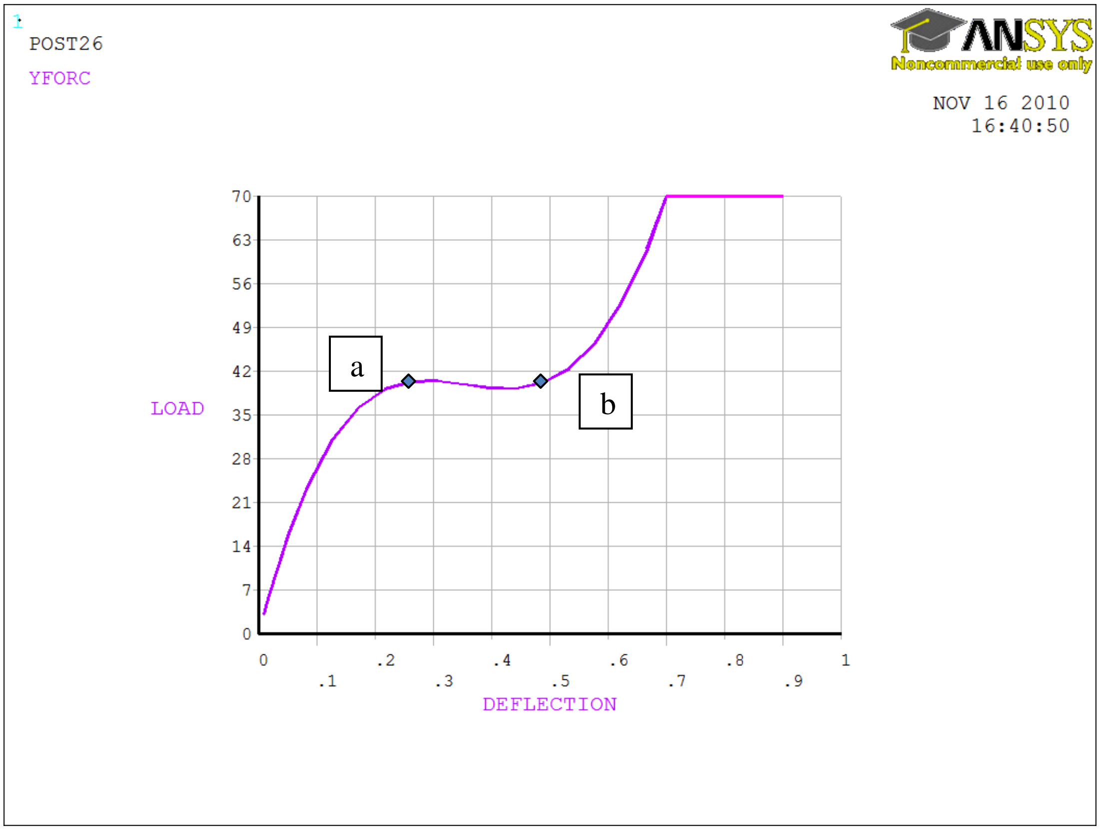


Figure 4.7 Load (lb) Vs displacement (in)

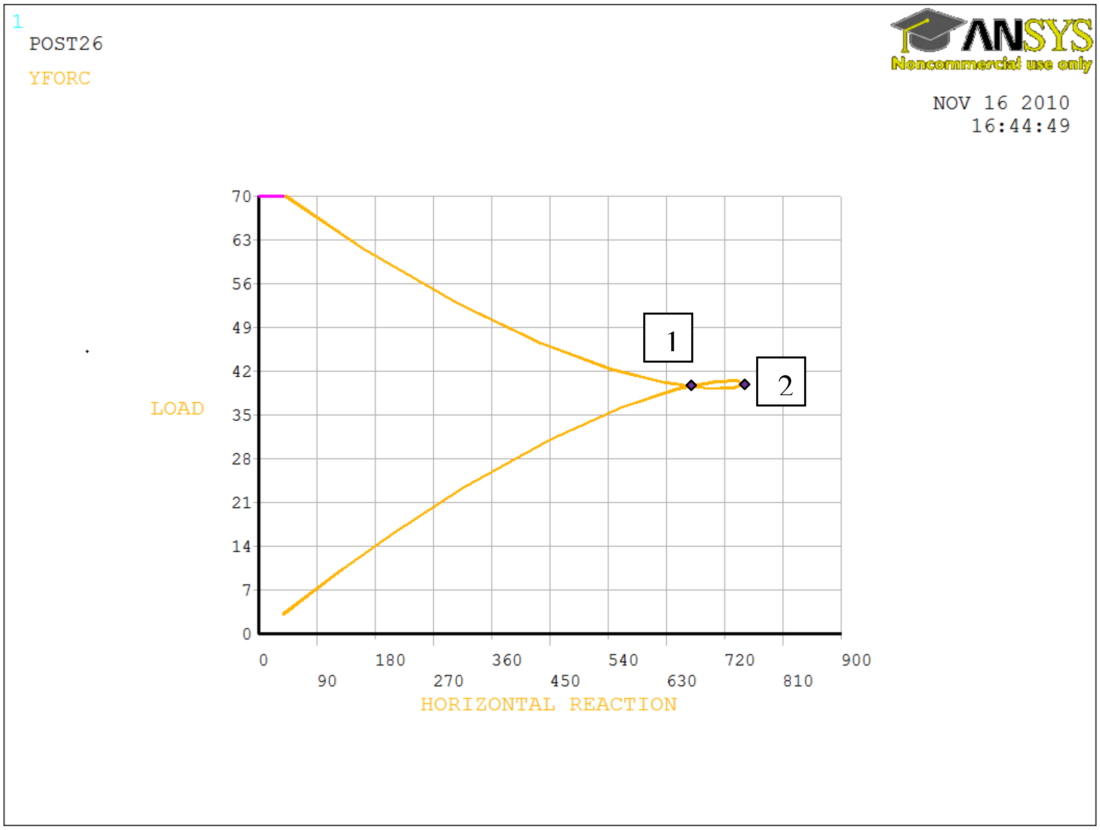


Figure 4.8 Load (lb) Vs Horizontal reaction

The maximum compressive force obtained in the members is 750 lb. In this case, as the central node is gradually pushed downward, the vertical force reaction goes on increasing till it reaches point (2) and so does the horizontal reaction force evaluated at the truss ends. The vertical load values in (1-2-1) region of the graph in Figure (4.8) correspond to those in the post-buckling region in Figure (4.7). The horizontal reaction goes on decreasing after it reaches point (2).

b) Hinged Shallow Truss

The shallow truss with hinged boundary conditions is deformed as depicted in the Figure (4.9)

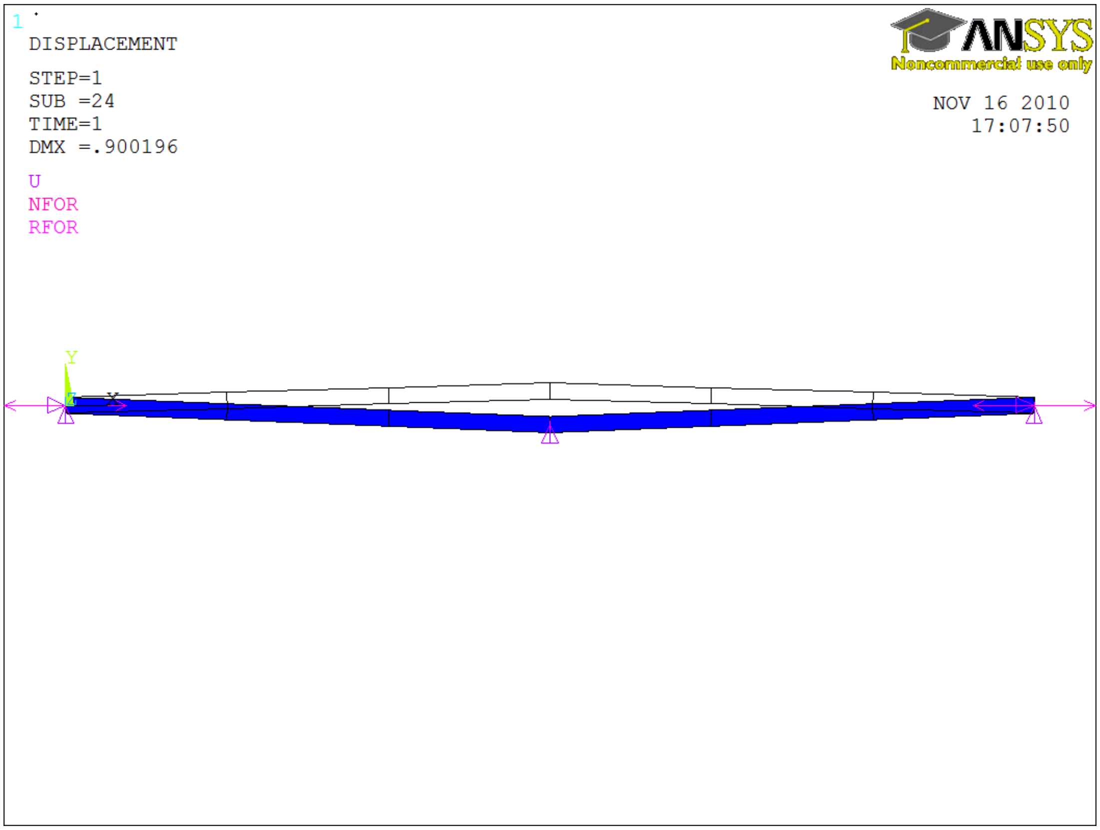


Figure 4.9 Deformed hinged truss

The deformation of the central node is plotted against the force reaction at that node and the observed trend is plotted in the following Figure 4.10. The graph displays the post buckling behavior of the structure. After the first peak of load, the displacement is still increasing but the evaluated force reaction is reducing till the next negative peak is reached and starts increasing again. The result matches with the results in [3].

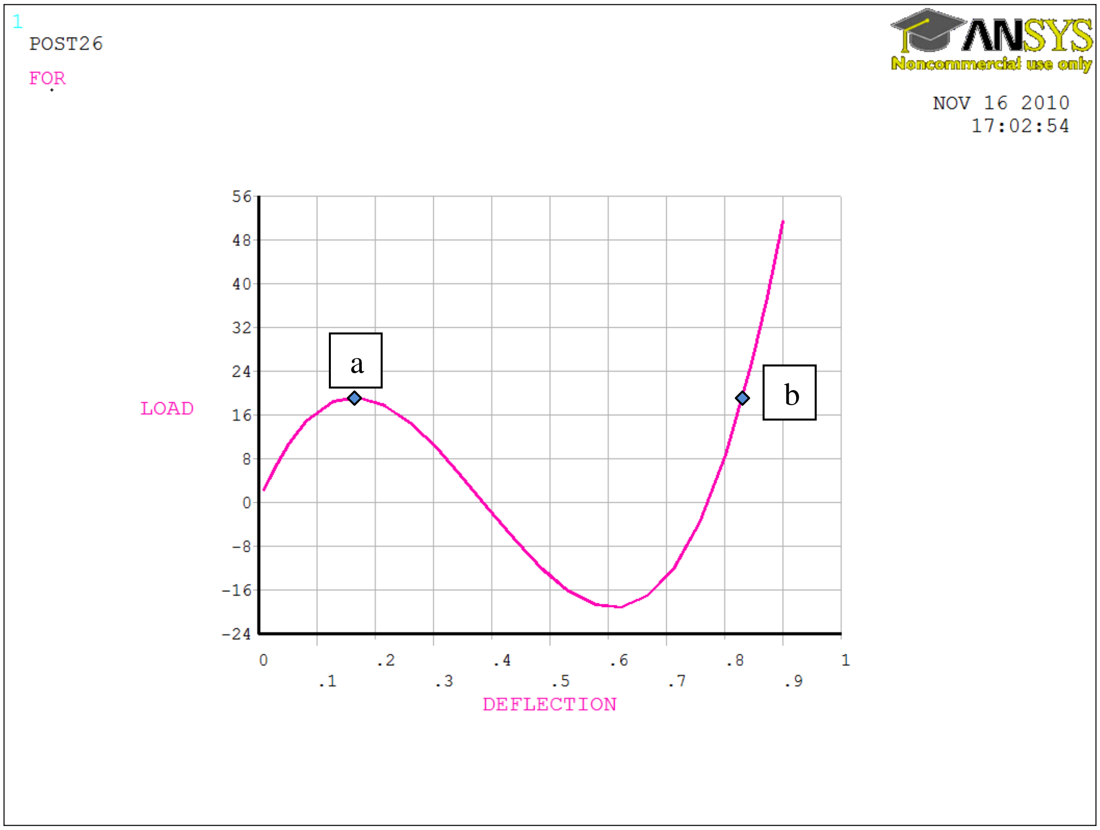


Figure 4.10 Load (lb) Vs deflection (in)

The limit point (a) for the truss occurs at 18 lb and 0.16 in.

The horizontal reaction force evaluated at the ends of the truss is plotted against the vertical load reaction in Figure 4.11. The maximum compressive force obtained is 850 lb. After reaching the maximum value at point (1) horizontal force decreases gradually. At point (1) the vertical force becomes zero and then continues to be in the negative range till it again attains the value of zero at point (2). The results match with the results in [3]. Reference [3] utilizes a 5 element mesh for both cases to analyze the post buckling behavior of structures.

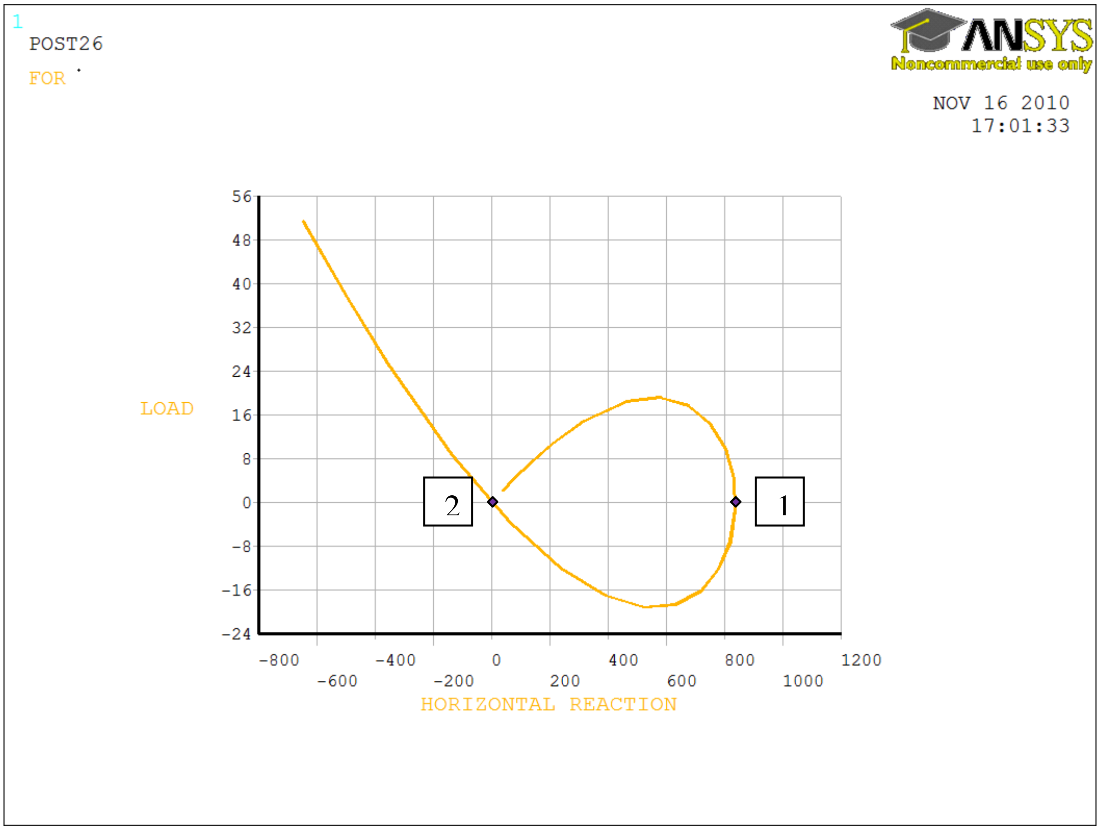


Figure 4.11 Load (lb) Vs Horizontal Reaction (lb)

4.2.2 Diagonal Truss

The diagonal truss is analyzed by assuming geometric and material non linearity.

4.2.2.1 ANSYS Simulation

a) Geometry

The geometry of the Diagonal Truss is as shown below in figure 4.12;

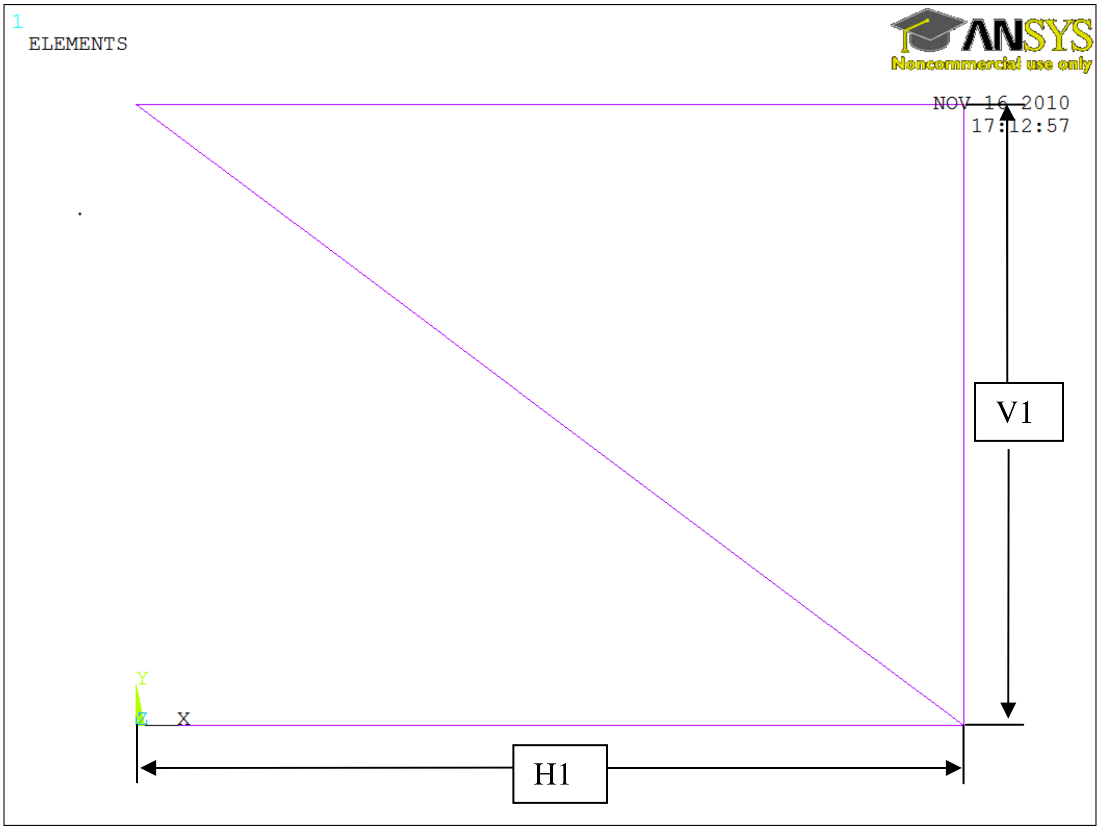


Figure 4.12 Diagonal truss Geometry

The dimensions used to model the truss are given in the following table.

Table 4.3 Diagonal Truss Dimensions

H1	48 in
V1	36 in
Cross sectional Area	1 in^2

b) Material Properties

A bilinear isotropic material is defined for the truss;

Table 4.4 Material Properties for Diagonal Truss

Young's Modulus	3×10^7 psi	2.07×10^{11} Pa
Yield Stress	30,000 psi	2.07×10^8 Pa
Tangent Modulus	3×10^5 psi	2.07×10^9 Pa

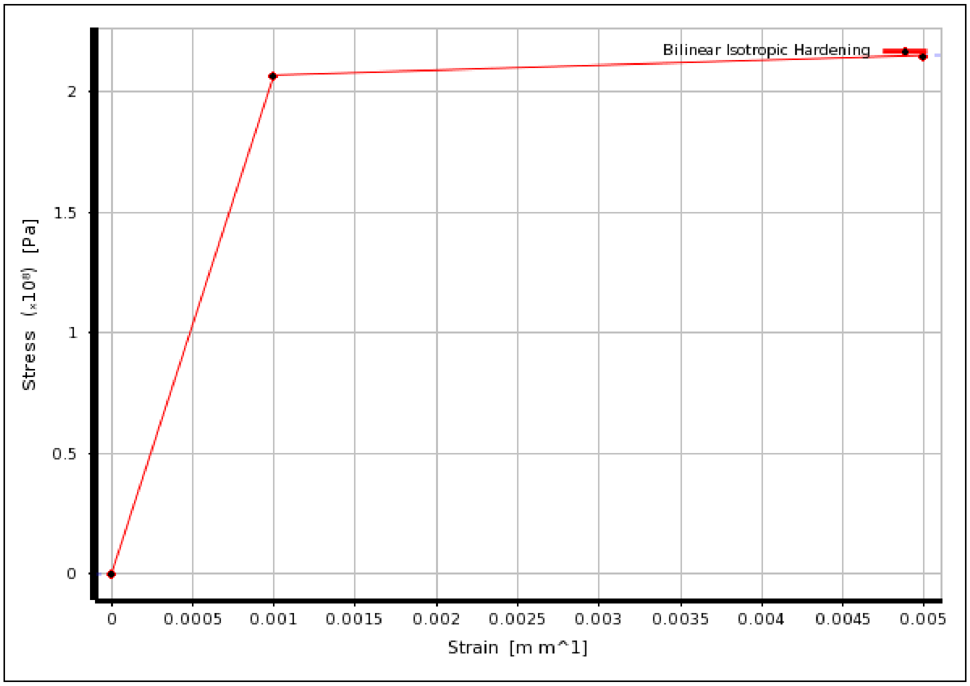


Figure 4.13 Stress Strain Curve

The tangent modulus is useful in describing the behavior of materials that have been stressed beyond the elastic region. When a material is plastically deformed, initially the relationship between stress and strain is linear with a high slope, and later after point (1) is reached, the slope of the line changes to a smaller value. The tangent modulus quantifies the "softening" of material that generally occurs when it begins to yield. Although the material softens, it is still generally able to sustain more load before ultimate failure. Therefore, more weight efficient structure can be designed when plastic behavior is considered, hence in structural analysis the tangent modulus is used to quantify the buckling failure [19].

c) Boundary Conditions

A displacement of 3 in is applied on node 6 in the negative Y direction. Nodes 9 and 1 are fixed as shown in figure 4.13.

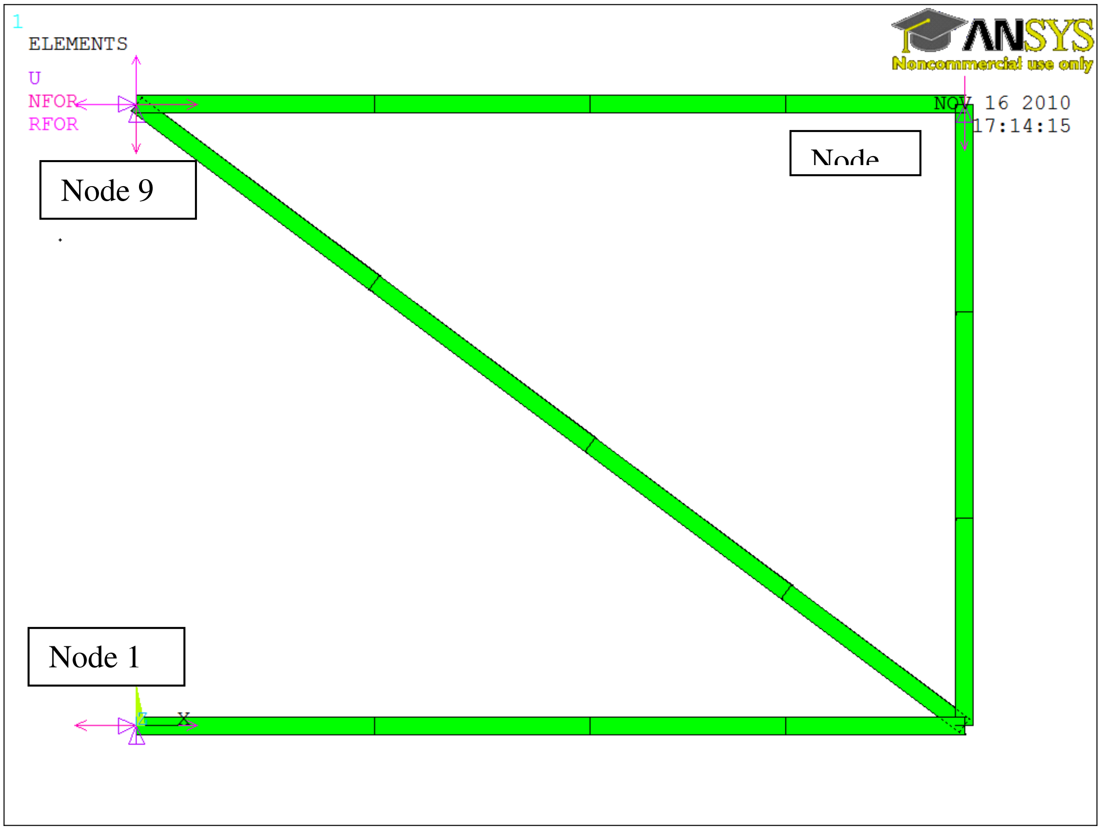


Figure 4.14 Boundary conditions for Diagonal Truss

4.2.2.2 Results

Due to the applied boundary conditions the truss is deformed. The deformed shape of the truss is as shown in the Figure 4.14.

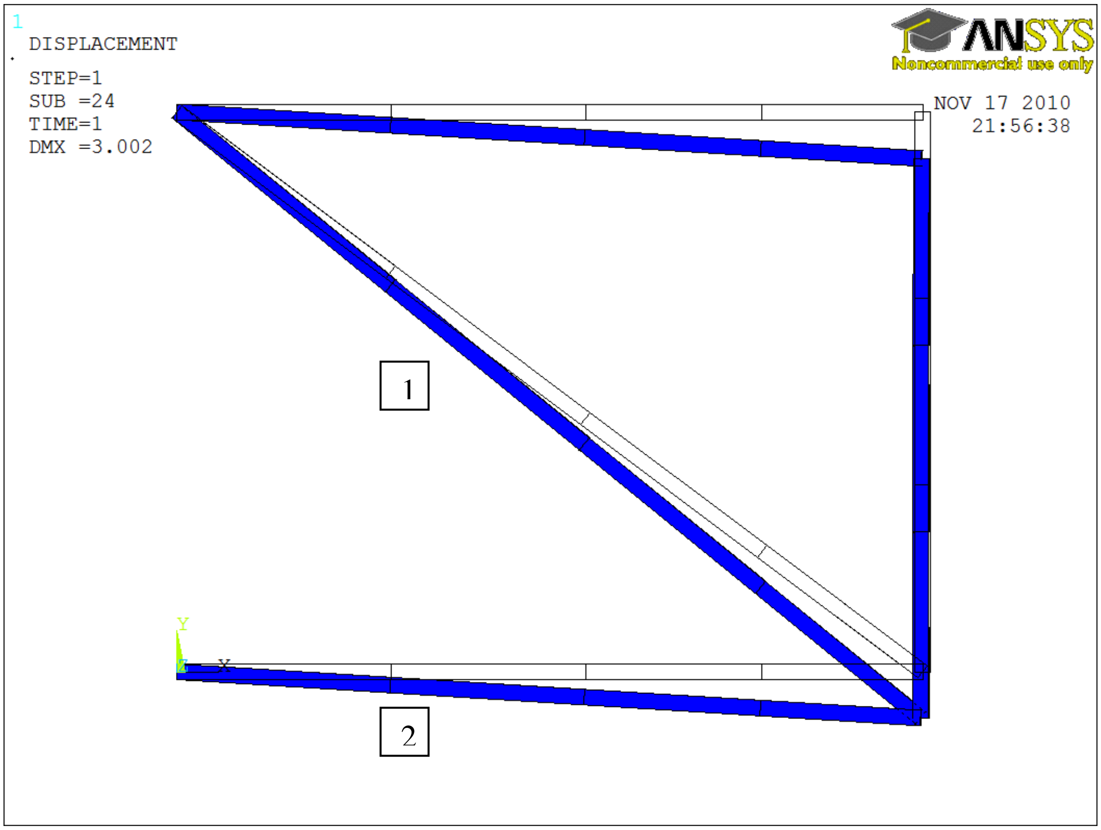


Figure 4.15 Deformed Shape of Diagonal Truss

Following is the plot of load reaction Vs deflection in Figure 4.16. The limit point of the truss occurs at a load of 1800 lb and a deflection value of 0.2 in. When the applied displacement pushes the truss in the negative Y direction, buckling is experienced in members 1 and 2 leading to a plastic deformation of the truss. The results match with the reference [15], where a non linear analysis program ADINA is used.

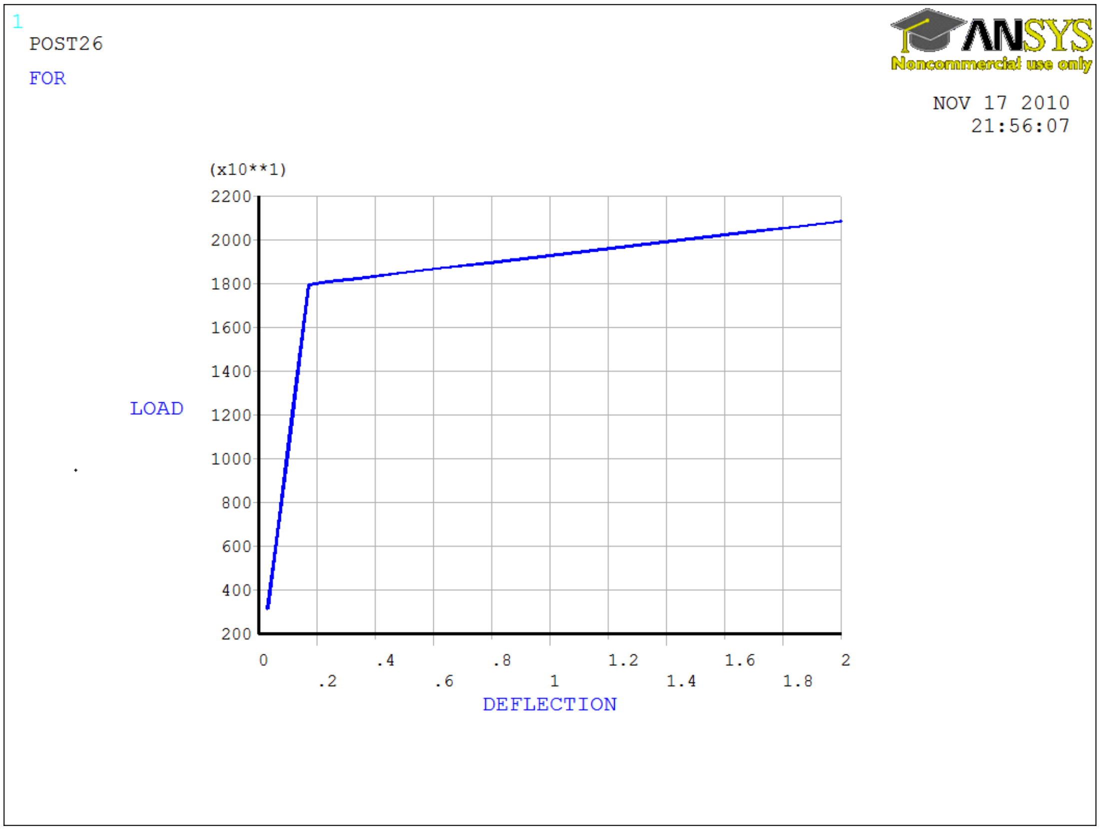


Figure 4.16 Load (lb) Vs Deflection (in)

4.2.3 Shallow Arch

4.2.3.1 ANSYS Simulation

The shallow arch is modeled using BEAM3 elements.

a) Geometry

The shallow arch is modeled using the following dimensions,

Table 4.5 Dimensions of Shallow Arch

Cross Section Area	0.188 in^2
Area Moment of Inertia	0.00055 in^4
H	1.09 in
H	0.1875 in
L	34 in

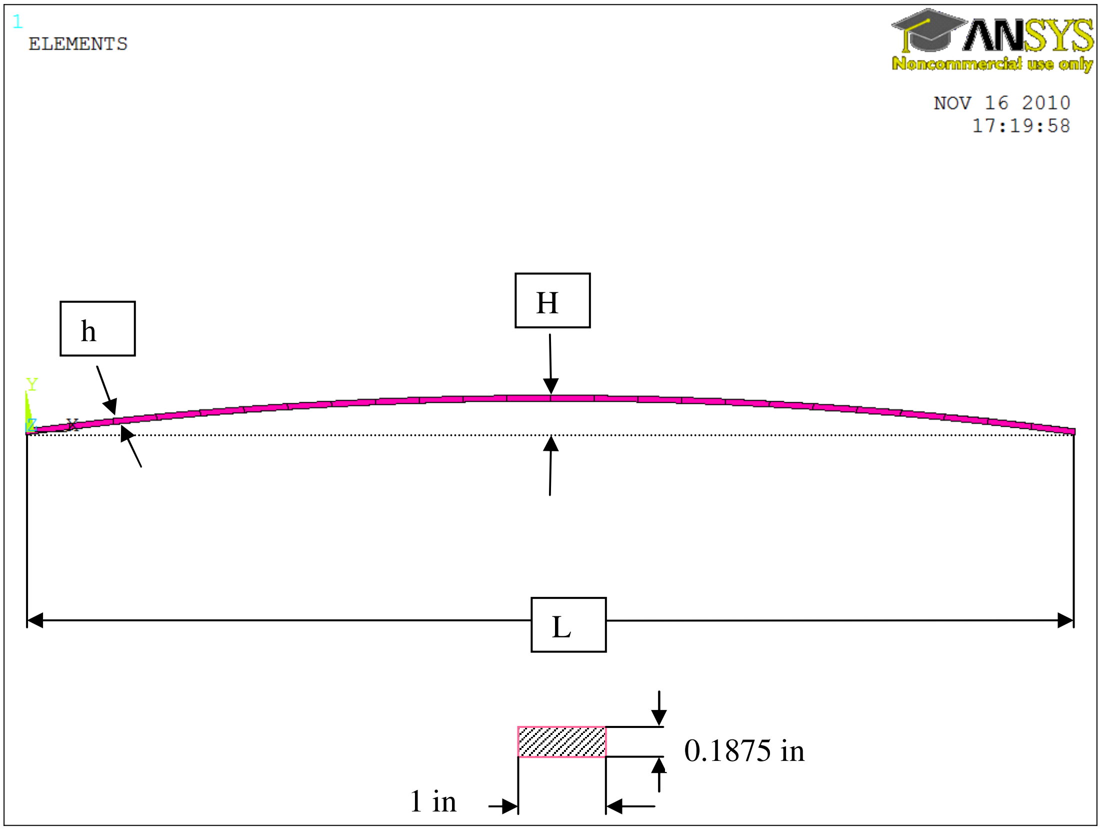


Figure 4.17 Shallow Arch geometry

b) Material Properties

The material is assumed to be linearly elastic with the following properties,

Table 4.6 Material Properties of Shallow Arch

Young's Modulus	10×10^6 lb/in²
Poisson's Ratio	0.2

c) Boundary Conditions

The arch is fixed at both ends for all degrees of freedom and a displacement of 1.8 in, in the negative Y direction is applied at the central node.

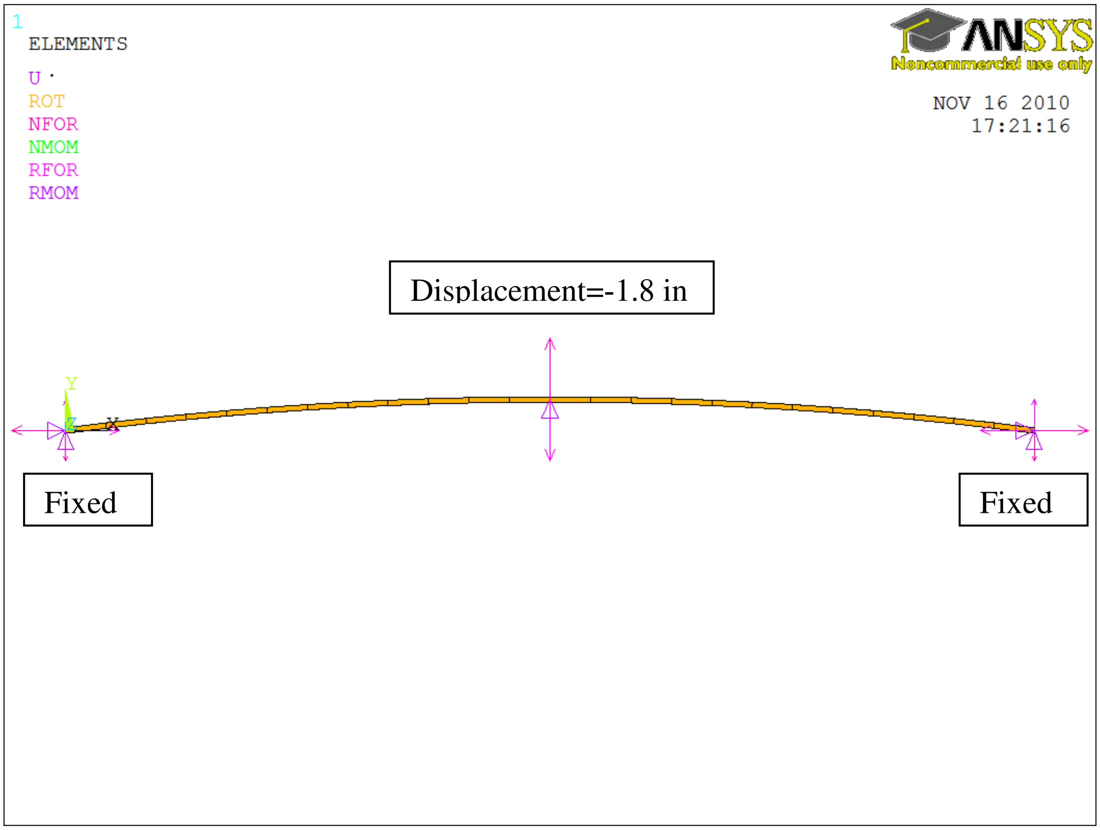


Figure 4.18 Boundary Conditions for Shallow Arch

4.2.3.2 Results

Mesh convergence is obtained on discretizing the domain by employing 24 elements. The applied boundary conditions cause the arch to deform at the center and towards the ends as shown in the figure 4.19 and 4.20 attached below.

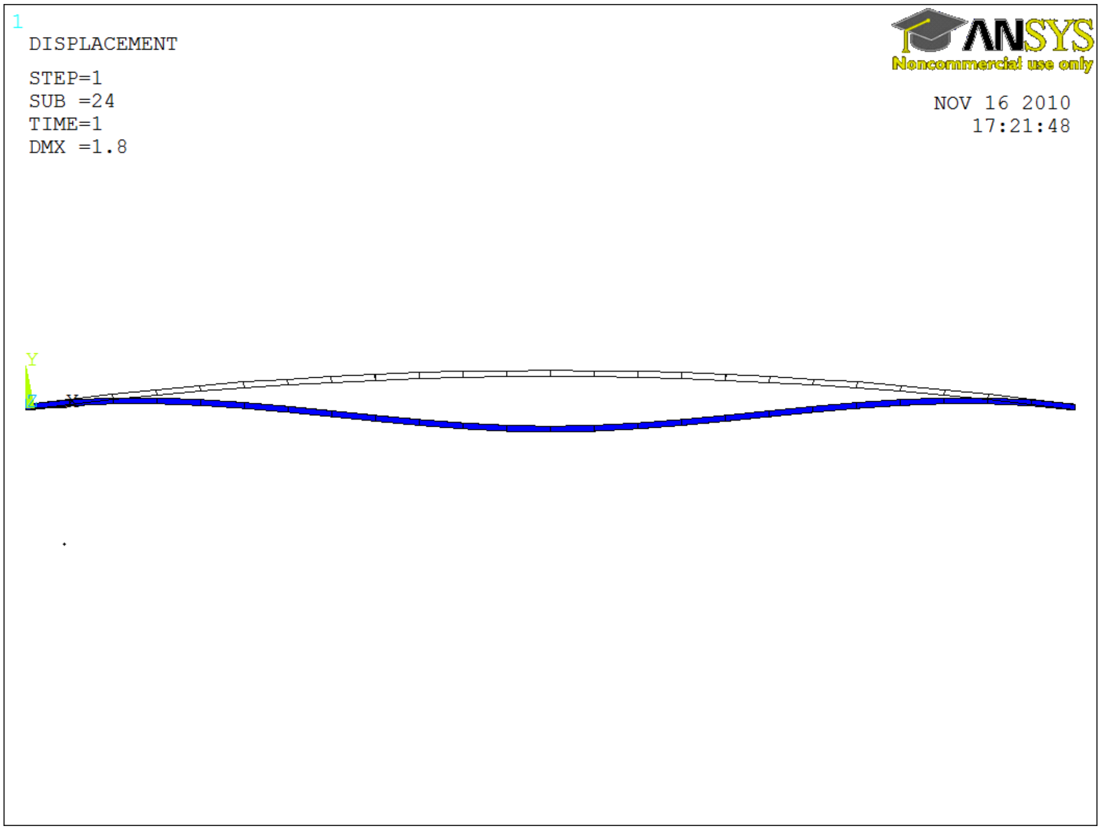


Figure 4.19 Deformed Shape of Shallow Arch with central displacement = -1.8 in

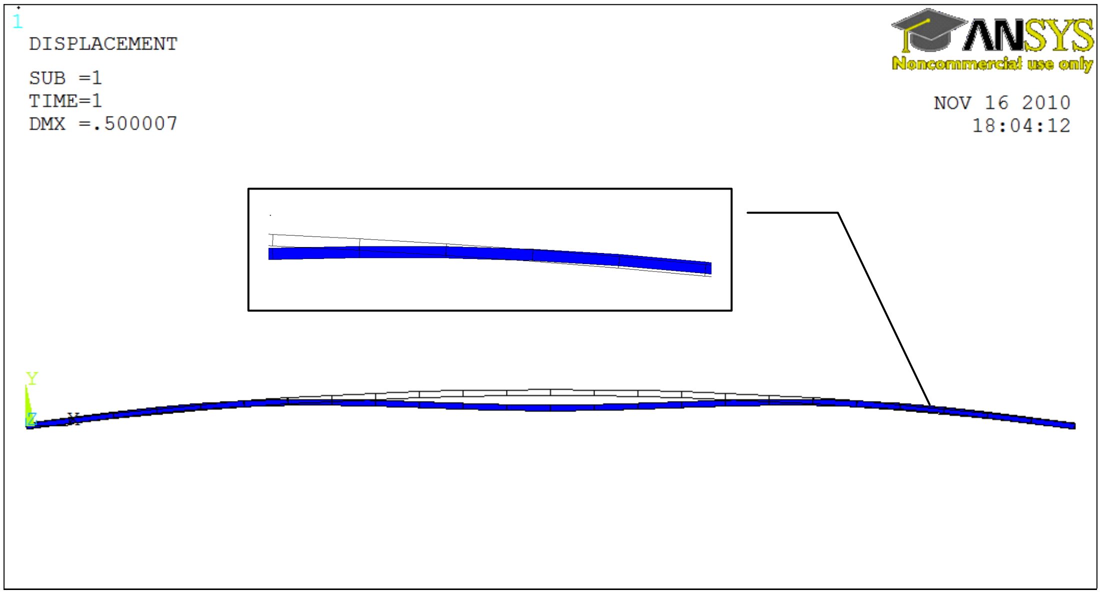


Figure 4.20 Deformed Shape of Shallow Arch with central displacement = -0.5 in

The vertical load reaction at the central node is plotted against the deflection at the central node,

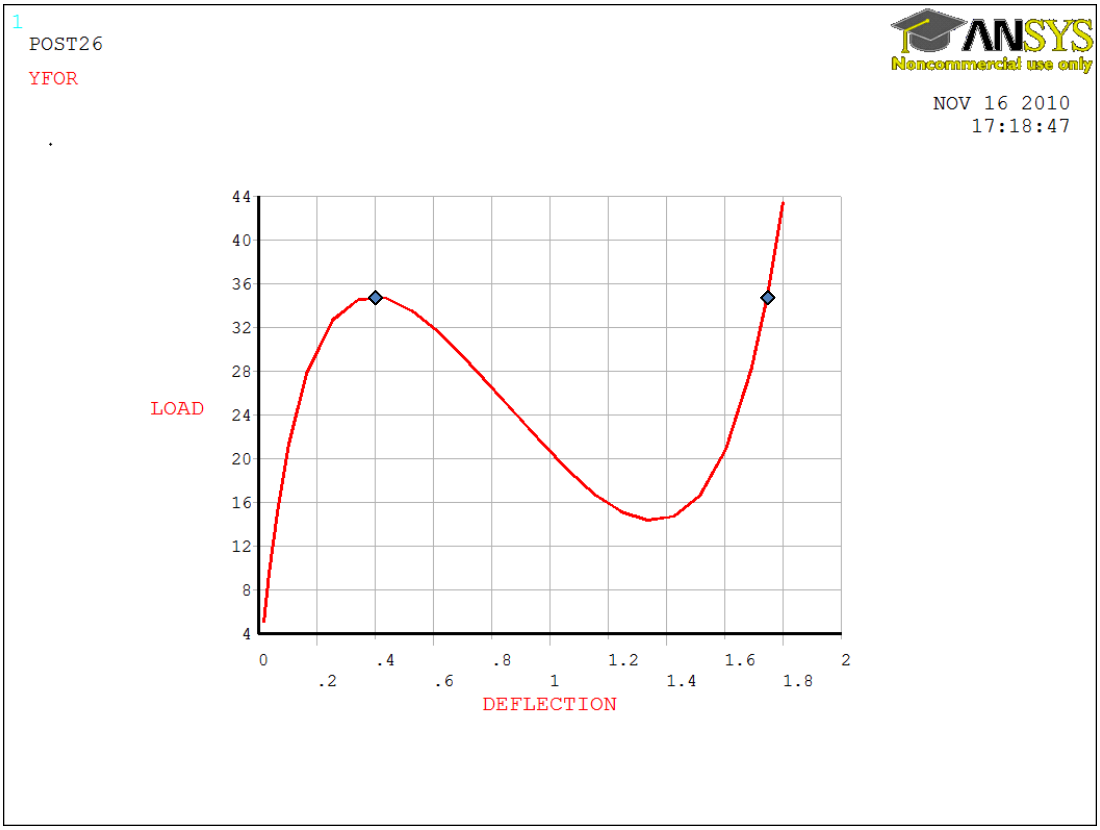


Figure 4.21 Load (lb) Vs Displacement (in)

The limit point of the truss occurs at a load value of 35 lb and a deflection value of 0.4 in. As the point of critical load is reached the structure goes into the non linear i.e. post buckling region. The region continues till the load of 35 lb is reached at a corresponding displacement value of 1.75 in.

The results match with those given in reference [15], where half of the arch is analyzed using 12 equal length beam elements.

CHAPTER 5

SHELL STRUCTURES

5.1 Cylindrical panel

The cylindrical panel is meshed by employing Shell 181 elements. Shell 181 is a 4-node element with six degrees of freedom at each node: translations in the x, y, and z directions, and rotations about the x, y, and z-axes. Its features include plasticity, elasticity and large deflection [18].

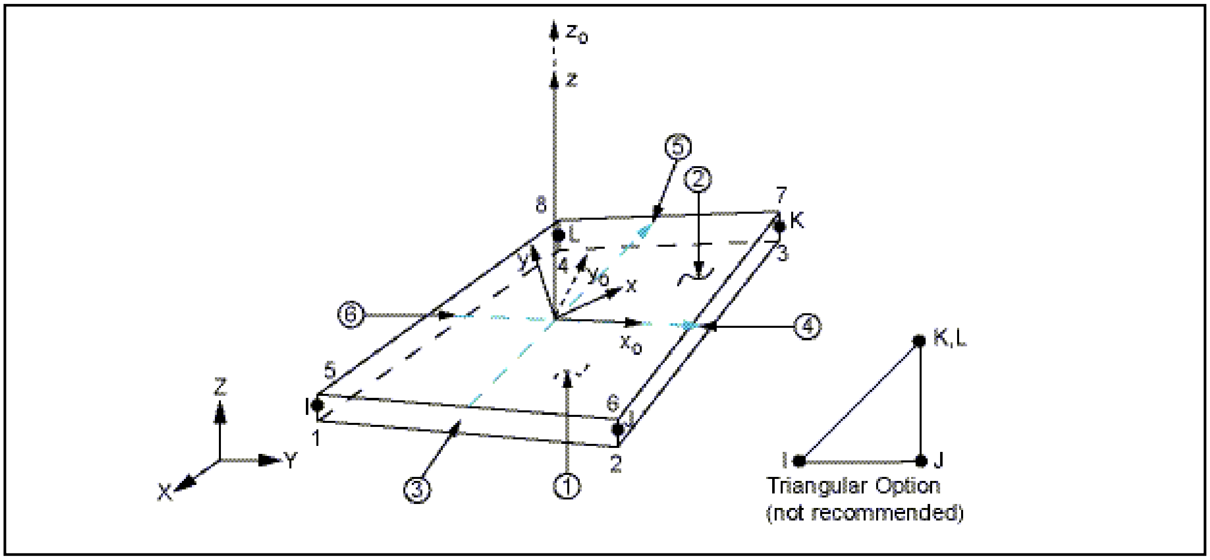


Figure 5.1 Element Shell 181

5.1.1 ANSYS Simulation

a) Geometry: The panel is a part of a cylinder that has the following dimensions,

Table 5.1 Dimensions of Panel

Radius, R	2540 mm
Length	580 mm
Thickness	12.7 mm
Included angle of panel	0.2 radian

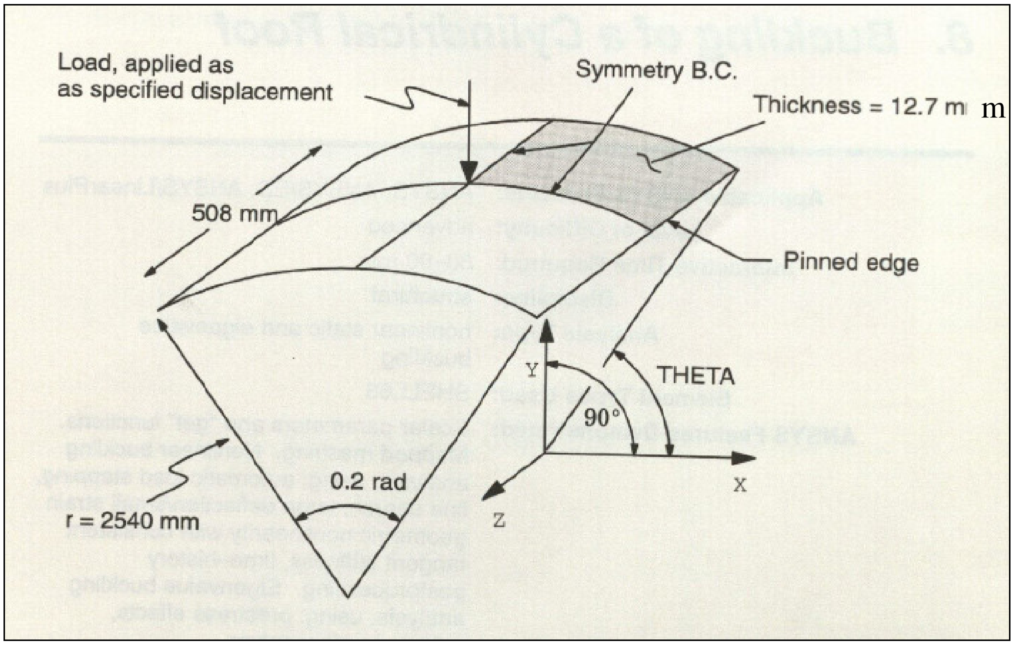


Figure 5.2 Geometry of Cylindrical panel

Due to symmetry a quarter portion of the panel is modeled and analyzed. The geometry of the modeled part and the mesh density is shown in the following figure 5.3.

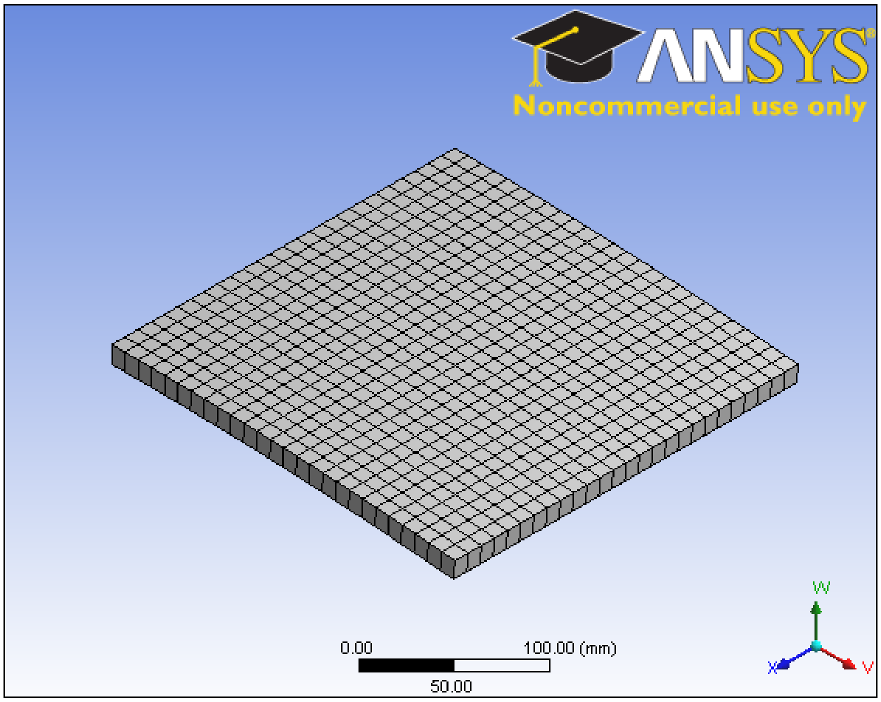


Figure 5.3 Mesh of Cylindrical Panel

b) Material Properties

The panel possesses linear elastic material properties:

Table 5.2 Material Properties of panel

Young’s Modulus	3102.75 MPa
Poisson’s Ratio	0.3

c) Boundary Conditions

Symmetry boundary conditions are applied on the inner edges. The cylindrical panel is simply supported on the outer edges. The problem is solved for the following cases of loading on the central node,

c.i) Applied displacement of 100 mm

c.ii) Applied force of 1000 N

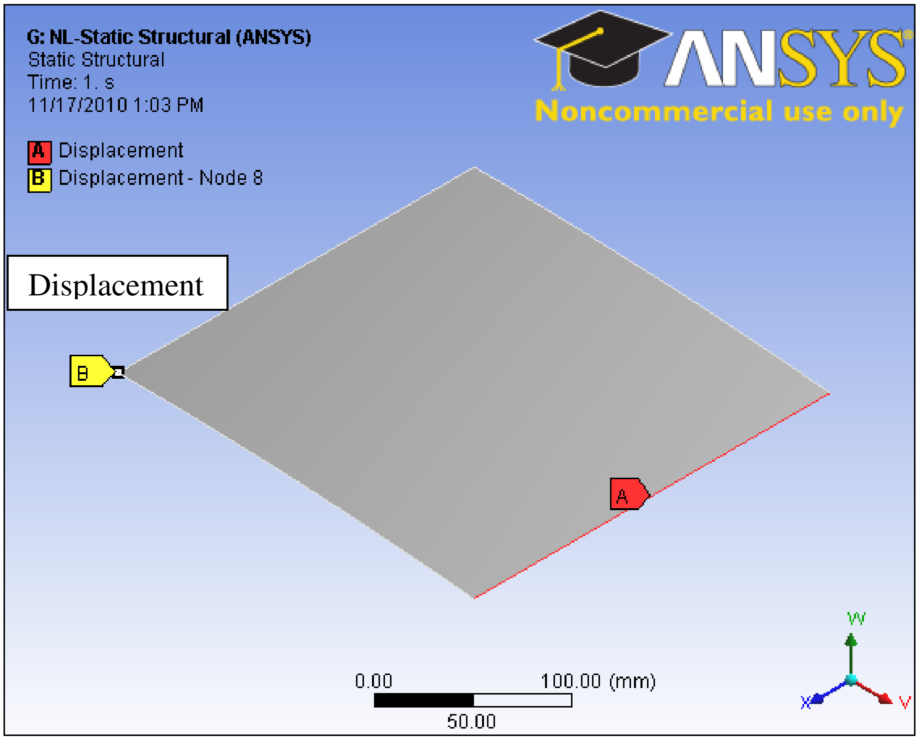


Figure 5.4 Applied Displacement at central Node

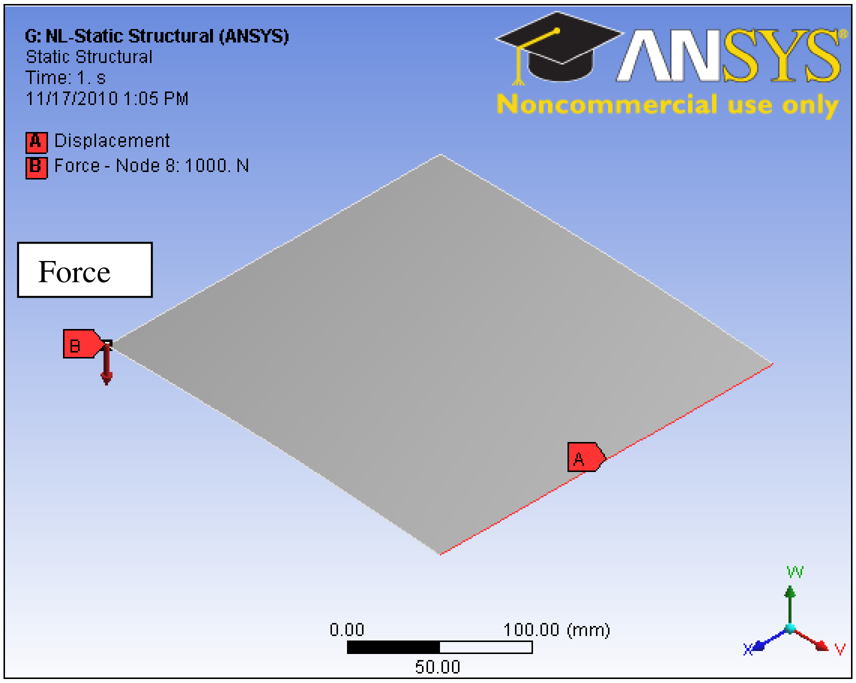


Figure 5.5 Applied Force at Central node

5.1.2 Results

With applied displacement at the central node, the panel deforms as in the below figure 5.5

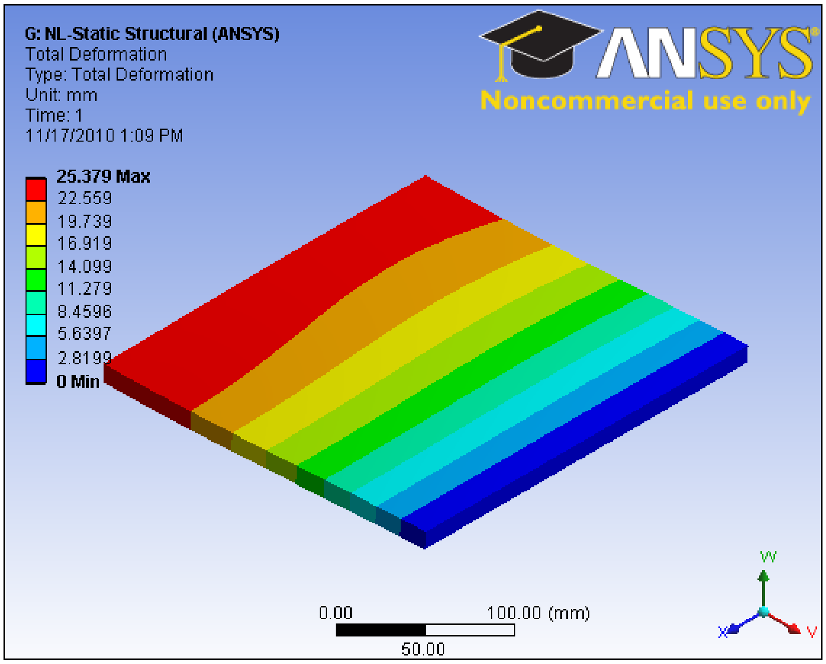


Figure 5.6 Deformed Shape of Cylindrical Panel

When displacement is applied at the center of the panel, the panel experiences a reaction force against the applied displacement. The reaction force at the central node is evaluated as shown in Figure 5.7,

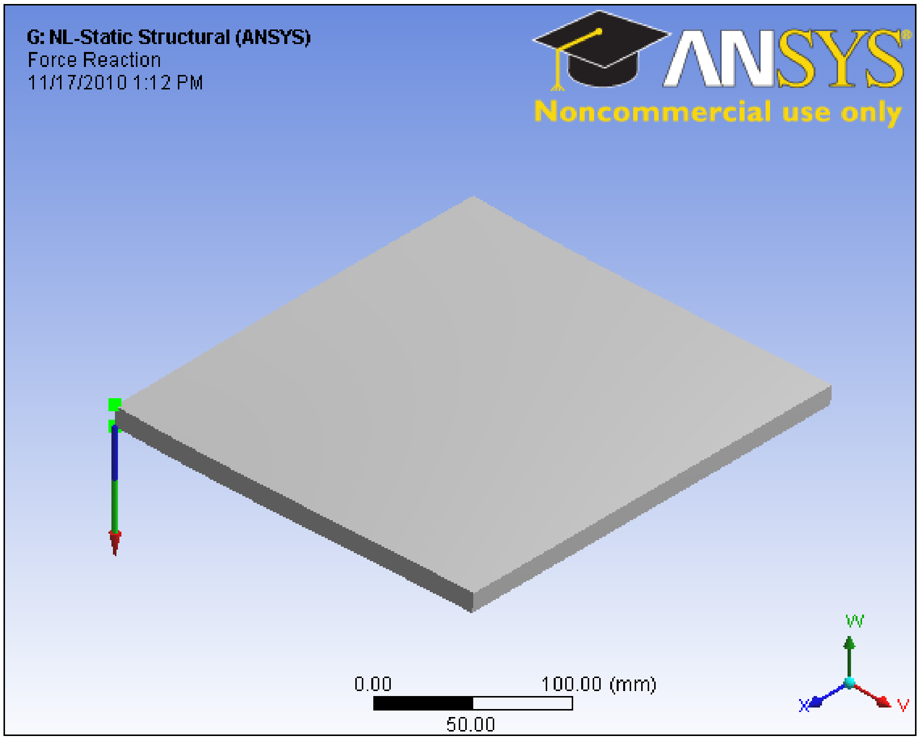


Figure 5.7 Force Reaction at Central Node

The equilibrium path for the panel is plotted below in Figure 5.7. After the limit load, the path enters the non-linear region and gradually approaches the new stable point which would have the critical load equal to the first one but corresponding to a new structural shape.

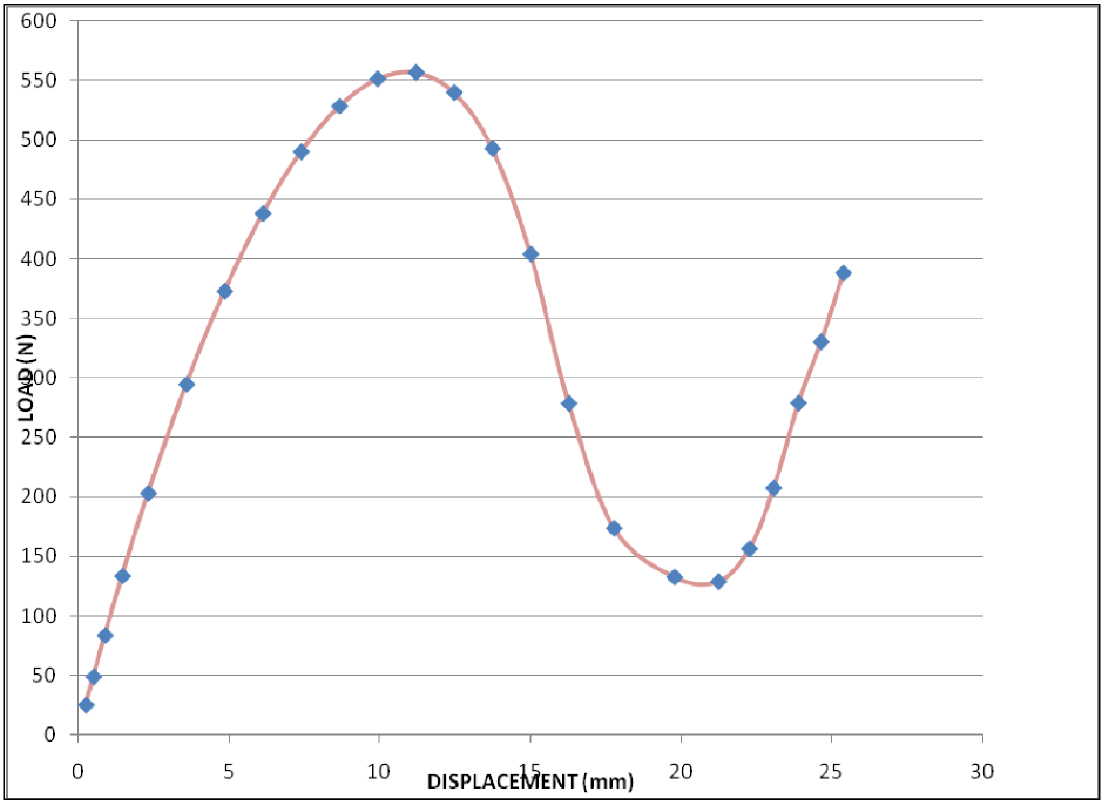


Figure 5.8 Load Vs Displacement

From the above results we observe that the limit point for the panel occurs at a load of 556.59 N. Since, a quarter portion of the panel is considered, the actual limit point for the whole panel occurs at 2226 N.

These results are in agreement with the results in reference [3] which uses a 16 element mesh for the quarter model.

When a force is applied at the central node, a maximum displacement of 30.7 mm in the negative Y direction occurs at the central node of the panel. The deformed shape of the panel is shown in figure 5.8.

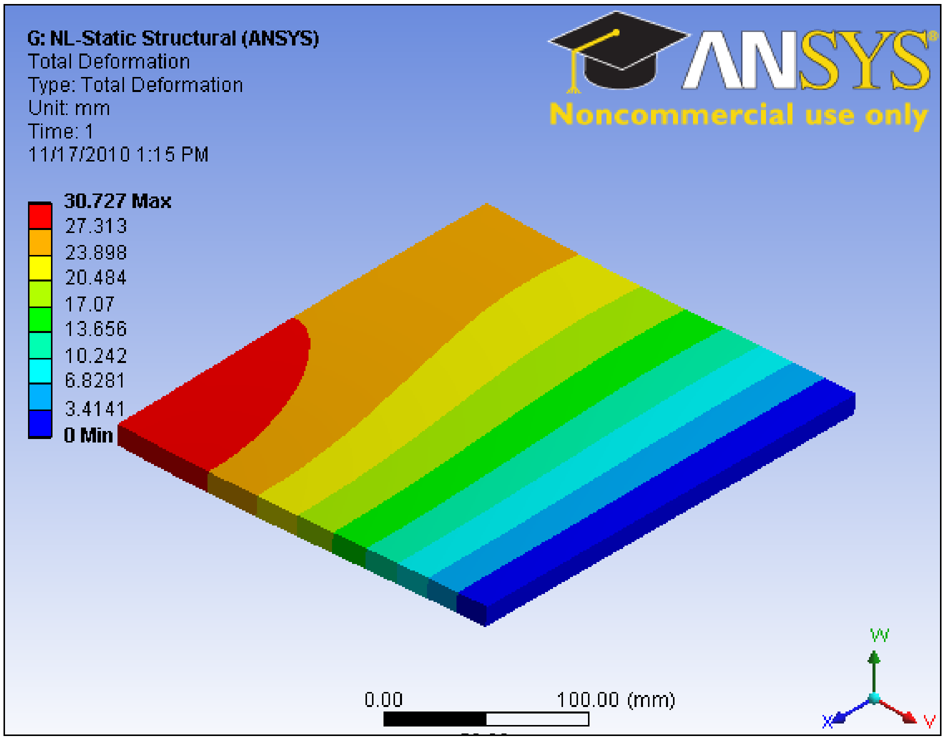


Figure 5.9 Deformed Shape of panel due to applied load

5.2 Conical Frustrum

Conical frusta find wide applications as energy absorbers. Hence, buckling and post buckling is an important area of study for these.

The buckling and post buckling behavior of thin walled conical frusta subjected to axial compression is analyzed. The component is assumed to be subjected to geometric and material non-linearity. The conical frusta in the following problem are simulated by crushing them between rigid plates and hence experience contact non linearity [2].

5.2.1 ANSYS Simulation

a) Geometry

A number of conical specimens were analyzed using ANSYS Workbench. Following are the dimensions of the specimens,

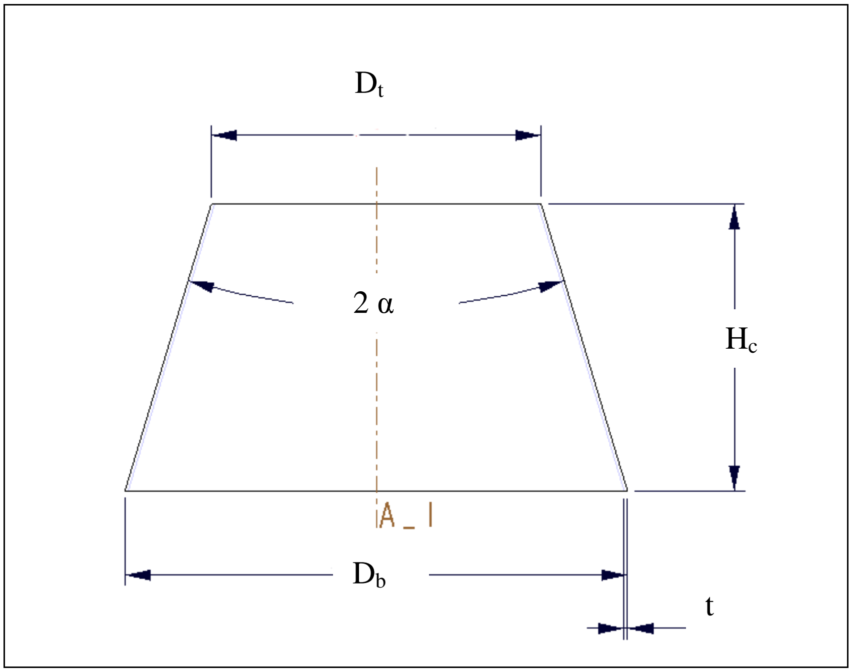


Figure 5.10 Geometry of Frustum

Table 5.3 Dimensions of Conical frusta

Specimen	α(˚)	D_b	D_t	t	H_c
C6	18	135	88	0.9	68
C25	23	165	89	0.65	88.1
C25	23	165	89	0.7	88.1
C18	24	152	90	0.9	68
C2	16	128	88	0.9	69

All dimensions are in mm

The frustum and the plate are meshed using the Shell 181 element. While performing the analysis the plate on top of the cone is assumed to be rigid. Hence, when the 3D assembly is meshed, for the plate, a single mass element is created at its centroid. For such cases the mesh is created only on the faces in the defined contact [16].

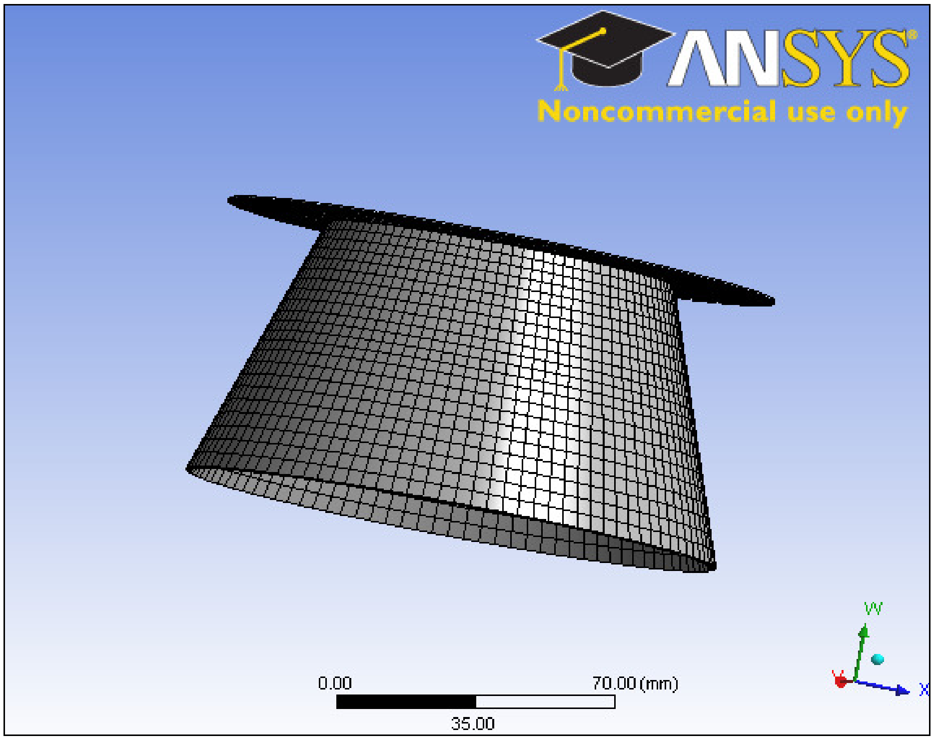


Figure 5.11 Mesh and geometry of specimen C6

b) Material Properties

The material assumed for this problem is Aluminum.

Table 5.4 Material Properties of Frusta

Young's Modulus	70 GN/m²
Poisson's Ratio	0.35
Yield Stress	55 MPa
Tangent Modulus	4213 N/mm²

The plate is made of structural steel and assumed to possess linear material properties,

Table 5.5 Material Properties of Rigid Plate

Young's Modulus	200 GN/m²
Poisson's Ratio	0.3

c) Boundary Conditions

The cone is subjected to axial compression between rigid platens. Hence, while modeling, the bottom edge of the cone is fixed and the top of the cone is subjected to compression by the application of remote displacement of 15 mm on the rigid plate. With remote displacement, workbench applies the specified displacement to the geometric center of the rigid body.

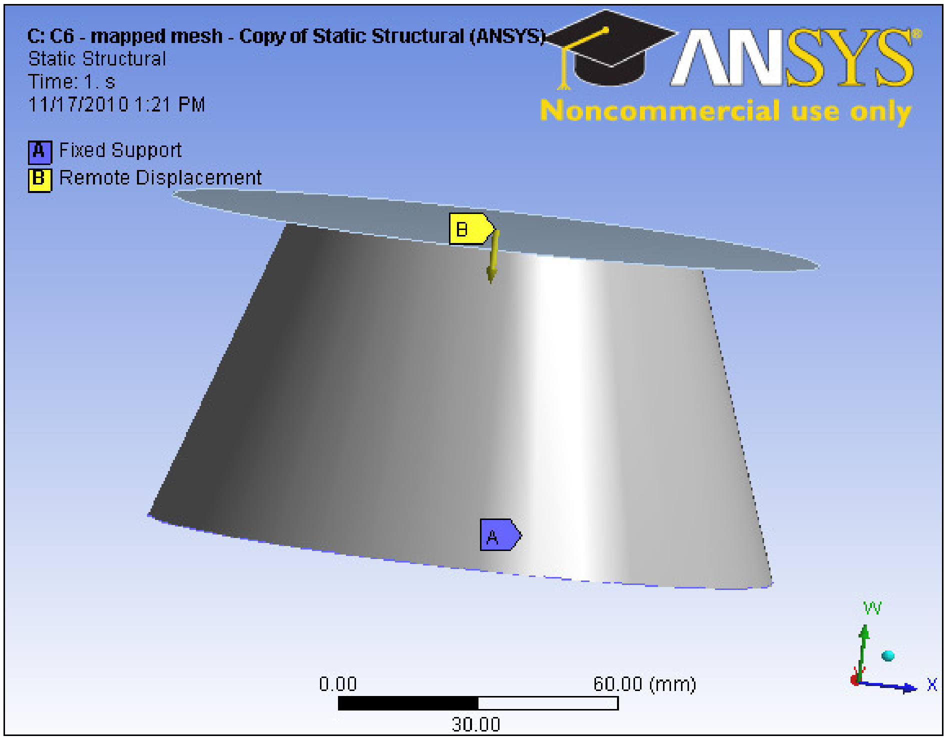


Figure 5.12 Boundary conditions applied to specimen C6

5.2.2 Results

The applied displacement causes the conical frustum to deform. The deformed shape of the frustum is shown in the figure 5.13.

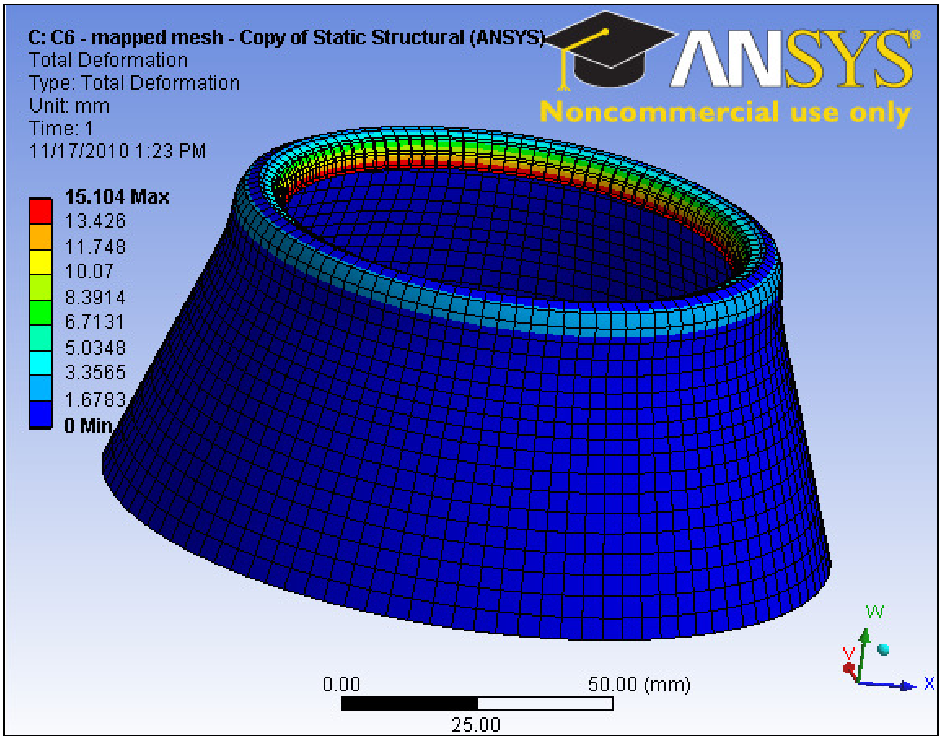


Figure 5.13 Deformed Shape of frustum

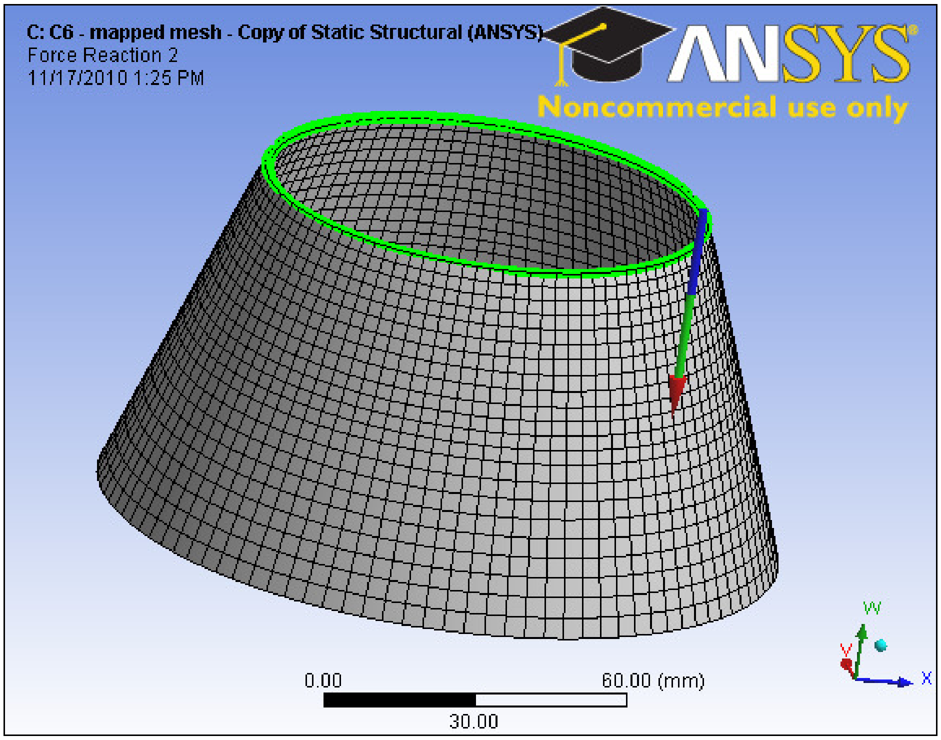


Figure 5.14 Reaction Force evaluated at contact region

The Force Vs displacement graph for specimen C6 is shown in fig (5.14). Aluminum represents a ductile type of failure. The peak load observed is 9.3 kN for a given displacement of 15 mm. The initial peak is observed for load value of 7.1 kN and then the load drops to 4.3 kN which is then followed by a rising trend.

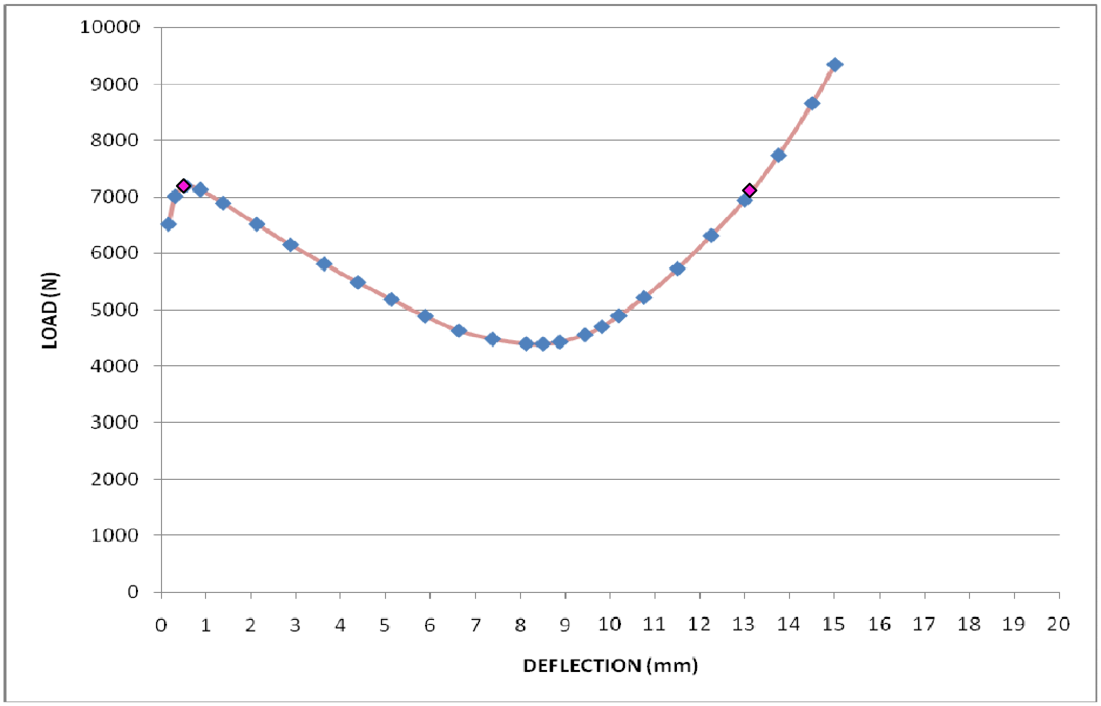


Figure 5.15 Load Vs Displacement

The results obtained from the analysis in ANSYS workbench by using shell 181 element are fairy comparable with the ones in [2]. The reference uses element type Shell 143 to mesh the frusta and the software used to accomplish the analysis is ANSYS APDL.

Table 5.6 Force Values for Frusta

	ANSYS Workbench		ANSYS APDL		Experimental
Element Type	Shell 181		Shell 143		
Specimen	Max Force (kN)	Avg Force (kN)	Max Force (kN)	Avg Force (kN)	Max Force (kN)
C6	9.3	5.9	8.2	5.5	3.4
C25(0.65)	5.8	3.1	4.4	13.7	4.9
C25 (0.7)	6.3	3.5	4.4	13.7	4.9
C18	7.7	5.1	12.7	8.35	5.5
C2	14.7	7.7	9.2	7.2	4

The observed differences in the force values are the cumulative result of the difference in Mesh density and the element type implemented. The Mesh used in this work has larger mesh density. Also, under bending loads, for Shell 143, inferior results can be produced leading to a need of a refined mesh [13] [18].

5.3 Eigen Value Buckling Load

The Eigen value buckling analysis is performed for all cases of structures and the Eigen value buckling load is compared to the limit load obtained from the non-linear buckling simulation. From the results in Table 5.7 it is observed that the eigen value buckling load is higher in each case.

Eigen value buckling load is the load at which the structure experiences instability and fails immediately.

Table 5.7 Eigen Value Buckling Load compared to the Non-linear Buckling Limit Load

Structural Component	Software	Non-Linear Buckling Limit Load	Eigen-value Buckling Load
Shallow Truss - Clamped	ANSYS APDL	33.3 lb	93.5 lb
Shallow Truss - Hinged	ANSYS APDL	18 lb	35.8 lb
Diagonal Truss	ANSYS APDL	18 kN	26 kN
Shallow Arch	ANSYS APDL	35 lb	53.713 lb
Cylindrical Panel	ANSYS Workbench	2.2 kN	2.7 kN
Conical frustum – Specimen C6	ANSYS Workbench	9.3 kN	17.84 kN
Conical frustum – Specimen C25	ANSYS Workbench	6.3 kN	7.5 kN
Conical frustum – Specimen C18	ANSYS Workbench	7.7 kN	13.65 kN
Conical frustum – Specimen C25	ANSYS Workbench	14.7 kN	20.4 kN

In case of Non-linear buckling analysis, limit load is the critical load after which the structure goes into a post-buckling stage, from one stable point to other, both having equal critical load values but different structural shapes. Hence Eigen value is predicted by ANSYS is higher.

CHAPTER 6

CONCLUSION AND RECOMMENDATIONS

The main objective of the thesis is to investigate the buckling and post buckling behavior of structural components that are bound to be subjected to heavy loads. Their complex equilibrium paths are explained to inform engineers of possible nonlinear behavior in designs and that instability may occur before a design bifurcation limit is reached. Understanding the large elastic displacement of these types of structures can prevent sudden buckling failures from applied operational and construction loads.

Structural components like the shallow truss, shallow arch, diagonal truss and shells like conical frusta and cylindrical panel are chosen for examination. These are realistic configurations that are directly usable in practical structural designs. These are simple metallic structural components from which buckling and post buckling can be studied.

The investigated models displayed snap-through behavior after reaching a critical point. The nonlinear behavior was identified by plotting the equilibrium paths of the center nodes. Post buckling region is that area of these equilibrium plots, that lies between the two stable points that correspond to the same critical load value but different displacement values.

This behavior of the analyzed structures signifies that under loaded condition the structure does not become unstable and buckle immediately. These load values are compared to the Eigen buckling load values and Eigen Value buckling load is found to be higher. At this load the structure becomes unstable and buckles immediately.

The post buckling study can be extended to micro sized structures that are used in micro electromechanical systems, for example, miniature trusses can be used as switches that can be integrated into circuits. Also, sensitivity analysis of the post buckling behavior of these structures can be accomplished. The Finite Element Methods software ANSYS APDL and workbench can be successfully implemented for the same.

APPENDIX A

DEFORMED SPECIMENS OF CONICAL FRUSTA

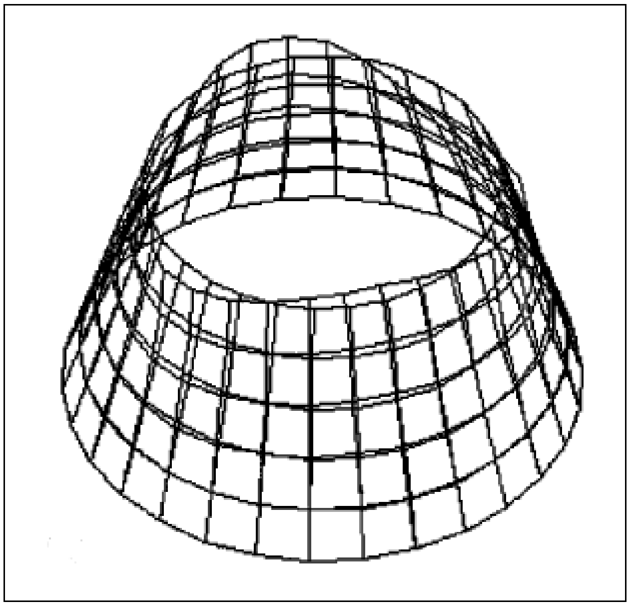
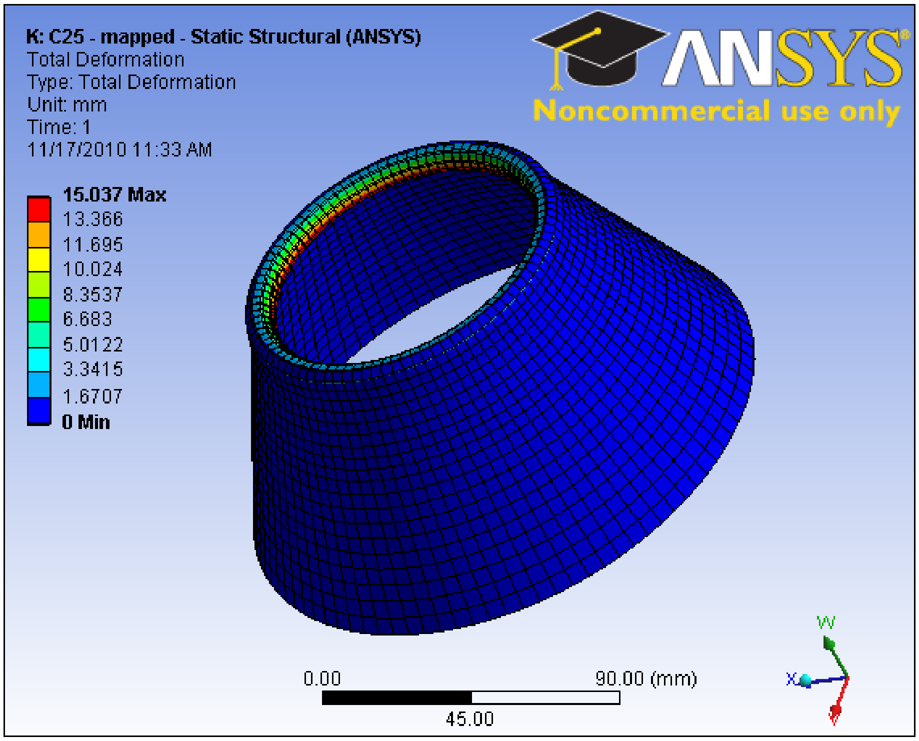


Figure A.1 Example of Mesh density for frusta in reference [2]



K: C25 - mapped - Static Structural (ANSYS)
Total Deformation
Type: Total Deformation
Unit: mm
Time: 1
11/17/2010 11:33 AM

Figure A.2 C25 Deformed Shape for thickness of 0.7 mm

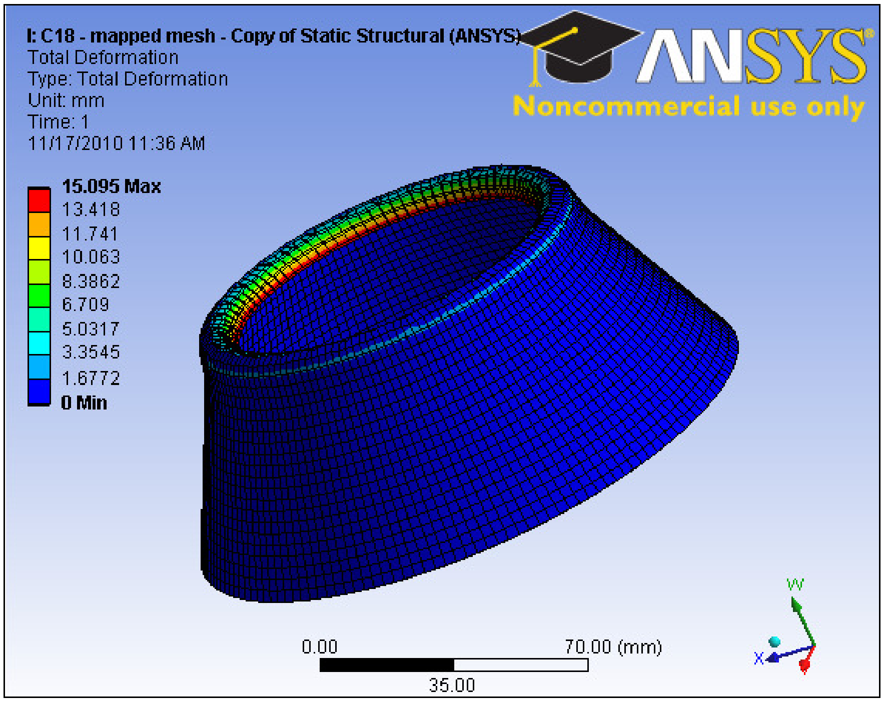


Figure A.3 C18 Deformed Shape

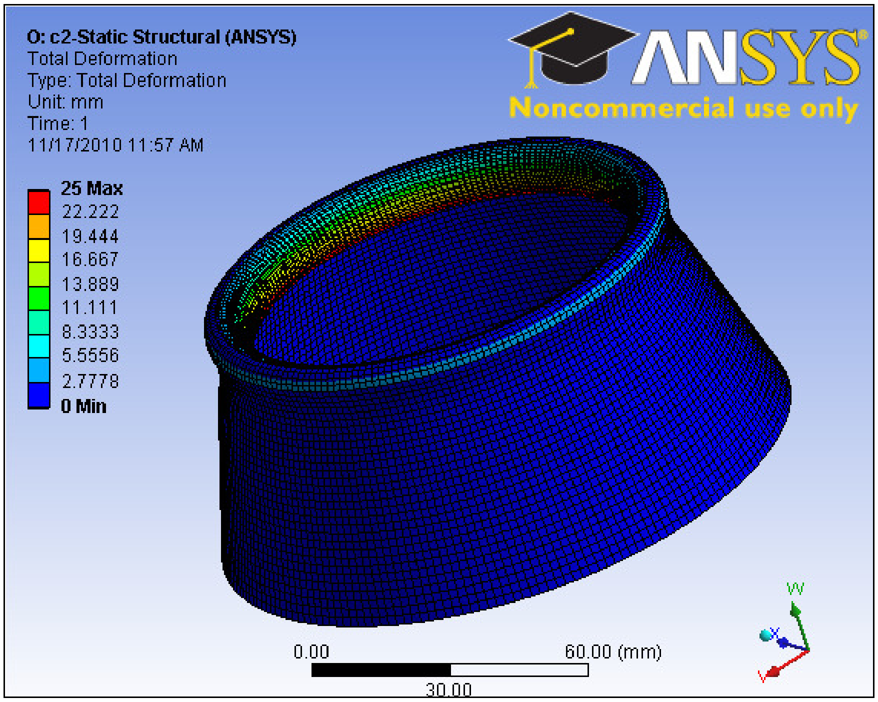


Figure A.4 C2 Deformed Shape

APPENDIX B

EIGEN VALUE BUCKLING MODE SHAPES

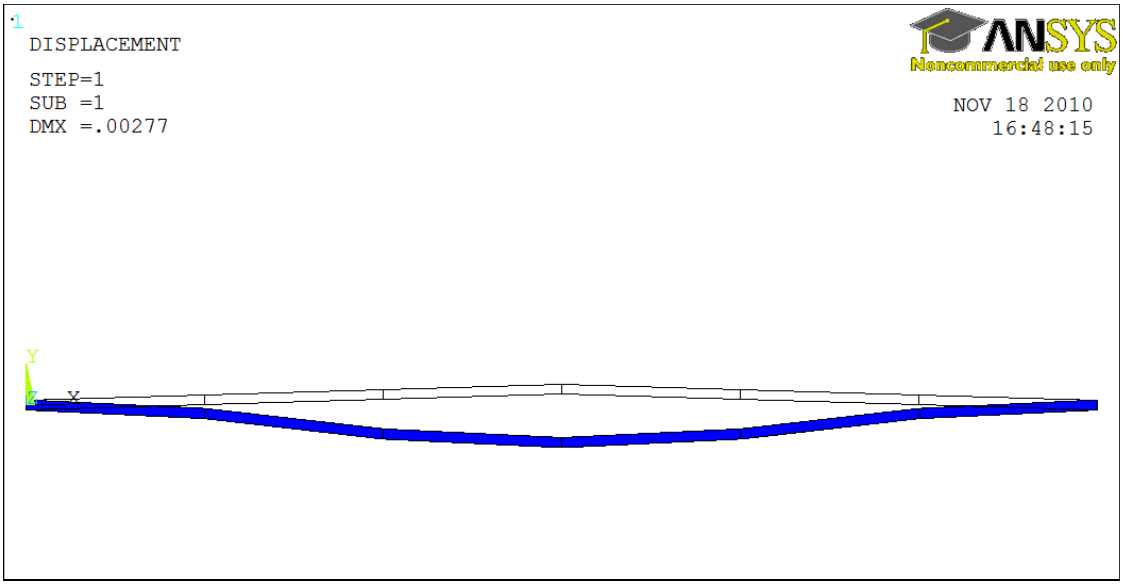


Figure B.1 Shallow Truss – Clamped

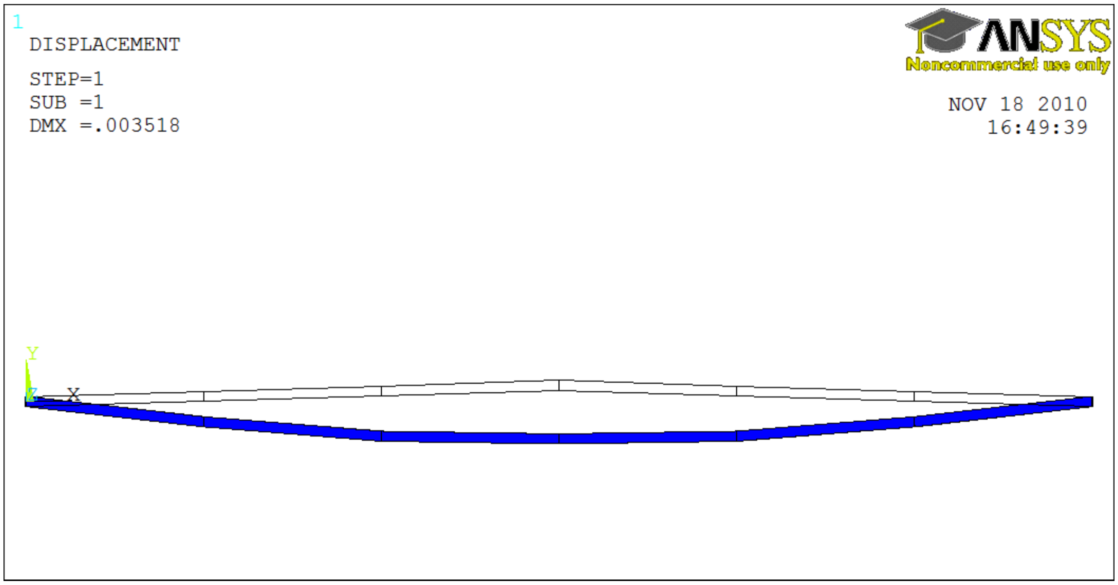


Figure B.2 Shallow Truss - Hinged

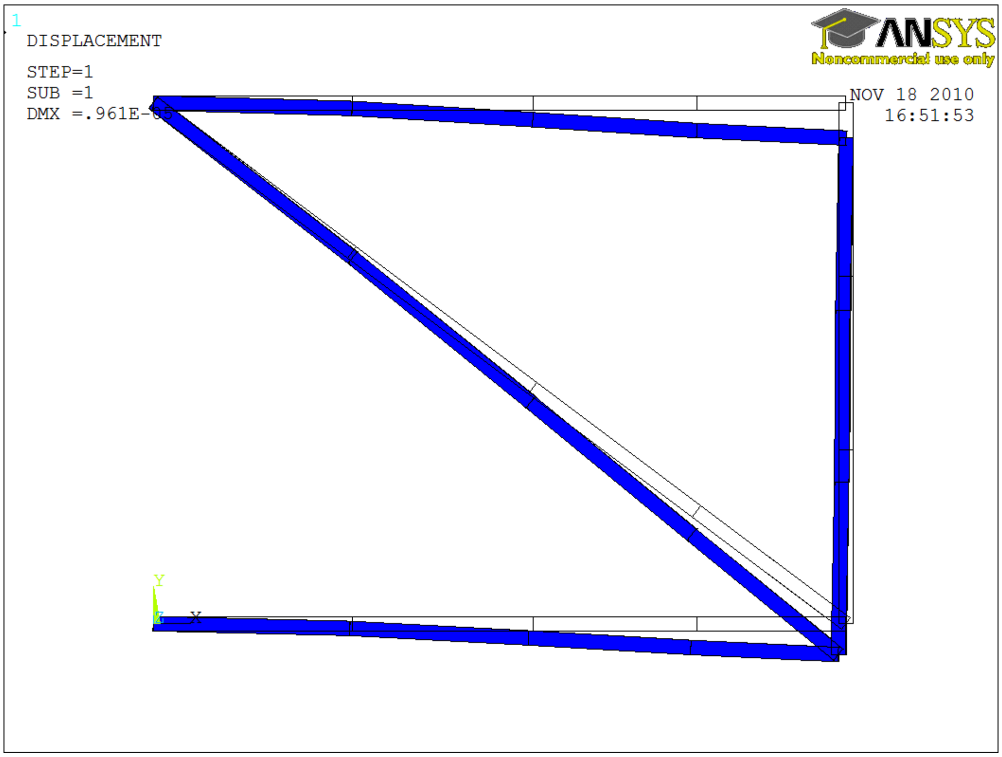


Figure B.3 Diagonal Truss

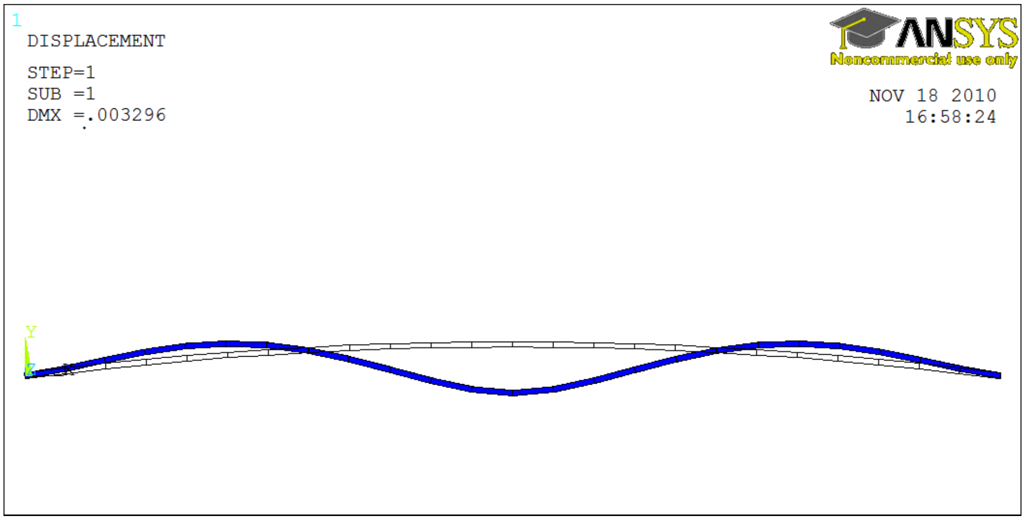


Figure B.4 Shallow Arch

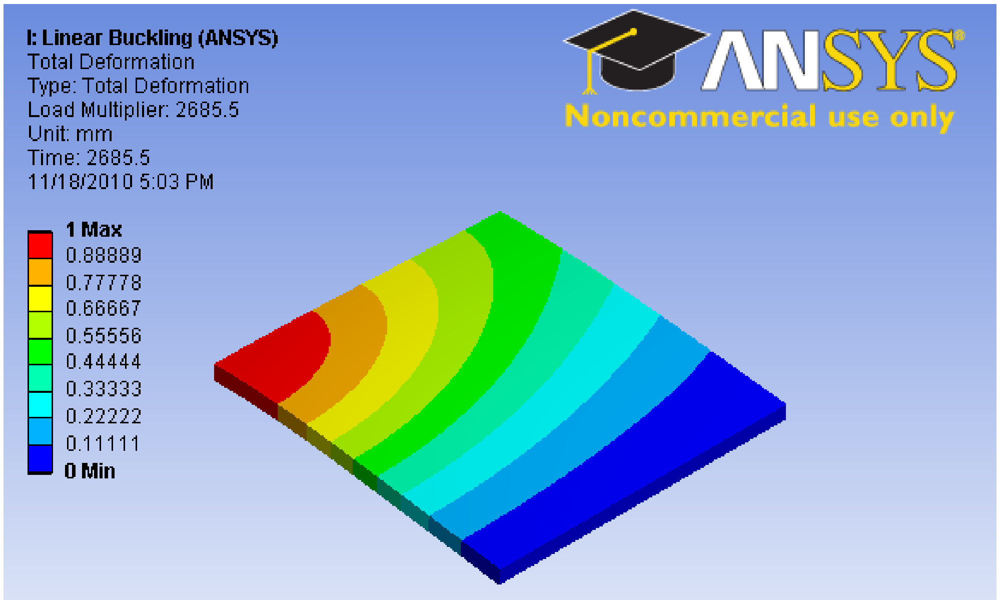


Figure B.5 Cylindrical Panel

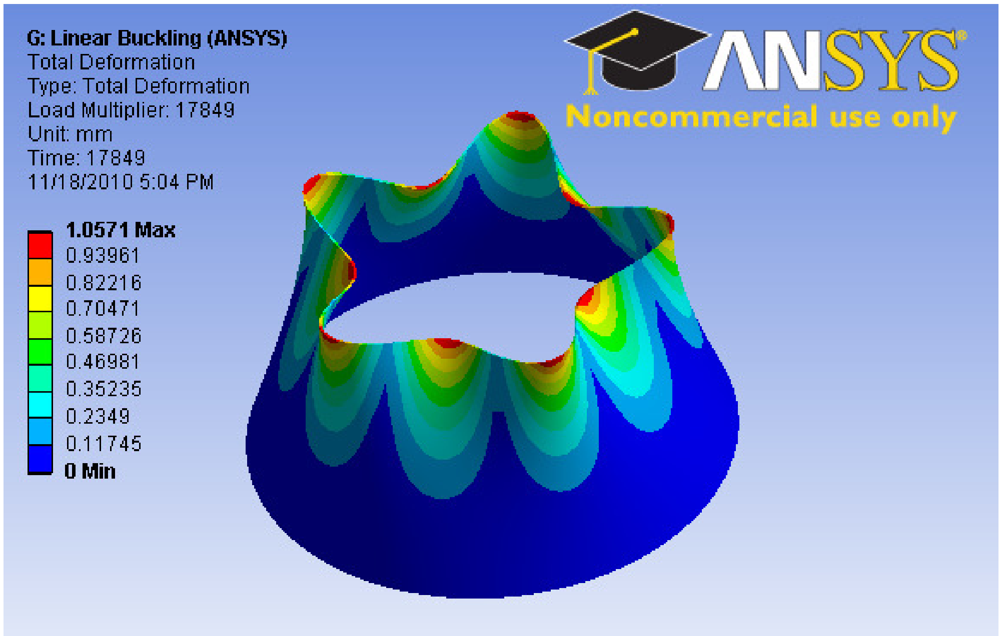


Figure B.6 Conical Frustum Specimen C6

REFERENCES

1] Buckling and post-buckling behavior of thin walled cylindrical steel shells with varying thickness subjected to uniform external pressure; S Aghajari, K Abedi and H Showkati, Thin Walled Structures, 44, (2006) 904-909

2] Buckling of thin conical frustum under axial loads; N. K. Gupta, N. Mohamed Sheriff, R. Velmurugan, Thin Walled Structures, 44, (2006) 986-996

3] Post buckling behavior of plates and shells using a mindlin shallow shell formulation; A. Pica and R. D. Wood, Computers and Structures, Vol. 12, pp 759-768

4] Buckling of Axially compressed conical shells of linearly variable thickness using structural model; Chawalit Thinvongpituk and Pisit Techarungpaisarn, The 17th annual conference of Mechanical Engineering Network of Thiland, Oct 2003

5] The Inelastic post buckling analysis of space steel trusses using the generalized displacement control method; Huu Tai Thai and Seung Eock Kim, 5th International Symposium on Steel Structures

6] Plastic buckling of conical shells under axial compression; H Ramsey, J. of Mechanical Science, Vol 19, pp 257-270, 1977

7] Axisymmetric axial crushing of thin frusta; N. K. Gupta and H. Abbas, Thin Walled Structures, 36 (2000) 169-179

8] Elastic buckling and post buckling behavior of widely stiffened conical shells under axial compression; A. Spagnoli, M. K. Chryssanthopoulos, Engineering Structures 21 (1999) 845-855

9] The application of Finite Element Methods to the Large Deflection Geometrically Non Linear behavior of Cylindrical Shells, A.B. Sabir and A. C. Lock, Variational methods in Engineering, pp 66-75.

10] Buckling of thin walled conical shells under uniform external pressure, B S Golzan and H. Showkati, Thin Walled Structures 46 (2008) 516-529

11] Hill C.D., Blandford G.E. and Wang S.T. (1989) Post-buckling analysis of steel space trusses, Journal of Structural Engineering, Vol. 115, No. 4, pp. 900-919.

12] Non-Linear Buckling and post buckling analysis of shells of revolution; Tao Hong, Thesis, Sep 1999

13] www.kxcad.net

14] Sensitivity Analysis of the Post Buckling Behavior of Elastic Structures; H. A. Mang, G Hofinger and X. Jia

15] Solutions for Structural problems solved using ADINA.

16] Finite Element Simulations with ANSYS Workbench 12

17] ANSYS tutotial: University of Alberta

18] ANSYS Help Manual

19] The Finite Element Method and Applications in Engineering using ANSYS, Madenci, Erdogan.

BIOGRAPHICAL INFORMATION

Shruti Deshpande has completed her Bachelor's degree in Production Engineering from Pune university in 2005. She worked with BOSCH for two years after earning the degree. She started her Master's in Mechanical Engineering in the Spring of 2009 and completed her degree in Fall 2010. During her tenure as a Master's student, she served as a Graduate teaching Assistant for Dr. Kent Lawrence. Her thesis is based on the study of buckling and post buckling of Structural Components, incorporating the use of Finite Element Analysis tools such as ANSYS APDL and ANSYS Workbench. Her work interests are Finite Element Methods and Analysis and Computer Aided Design.