Stability and Geometrical Nonlinear Analysis of Shallow Shell Structures

Project- 2 October’2004

by

Srihari Kurukuri

Supervisor

Prof. Dr. -Ing. habil. C. Könke
Dipl. -Ing. T. Luther

Advanced Mechanics of Materials and Structures
Graduate School of Structural Engineering
Bauhaus Universität, Weimar
Germany
Acknowledgement

I would like to thank my advisor, Professor Dr.-Ing. Habil. C.Könke for giving me the chance of working in such an excellent scientific environment and for his technical support and professional guidance, as well his encouragement. It has been a great pleasure working with him.

I am very thankful to the co-worker, Dipl.-Ing. Torsten. Luther, for his many valuable discussions, patience, technical assistance, advice, and encouragement. I really be pleased about his time and effort in been in the part of supervision. Other thanks are extended to the staff of Institute for Structural Mechanics for their support during my project.

Finally, I send my special thanks to my mother and father, for all their love and marvelous support in the most difficult times of my life. They are encouraging and supporting me all the time. Without them I would not be able to be what I am.

Millions of gratitudes to God for everything I have.....
Conceptual Formulation:

In the course of architectural evaluation, the modern design becomes more and more decisive influencing factor for construction of buildings. In this context engineers have to face the challenge of developing constructions on the limit of best available technology. Such a special challenge is the realization of shallow shell structures e.g. used for roofing of sports facilities. Often these structures are very sensitive for stability failure.

The project is dealing with the problem of highly geometrically non-linear behavior on the example of a real gym roofing which collapsed two times (in 1997 and 1998) in the construction state by loss of stability. The task in this project is to develop a mechanical model by abstraction of the real object to its main structural components and to define a critical load case for stability analysis. In a second step a geometrically non-linear finite element analysis for define load cases should be performed and evaluated. Due to often incorrect computational analysis in practice, the project is focused on the investigation of influence of support conditions, joints and imperfections on the critical loads. The aim of the project is to develop a sensibility for structural limits of stability and to gain experience in problems of non-linear finite element analysis.

The following problems have been addressed related to the Gymnasium in Halstenbek:

- Development of an abstract geometrical model of the steel glass roofing as a basis for finite element simulations in ANSYS.
- Discretization and definition of support conditions, stiffness in joints and load cases in ANSYS. Load cases to apply: dead load, wind load (simplified load case)
- Geometrical non-linear calculations in ANSYS for defined support conditions, joints and load cases under the assumption of linear elastic material behavior
- Investigation of sensitivity of stability behavior (critical load) due to changes in support conditions, stiffness in joints and prestressing in steel cables.
- Investigation of sensitivity of stability behavior (critical load) due to applied imperfections according to wind load (varying shape and size of imperfections)
- For the defined load cases and applied imperfections: Optimization of glass roofing due to reasonable changes e.g. in support conditions, stiffness in joints, arch rise of shell, prestressing in cables.
- Documentation and presentation of results.
List of Figures

**Figure-2.1:** Possible load \(Vs\) displacement behaviors of thin walled structure 6

**Figure-2.2:** Prebuckling and postbuckling of perfect and imperfect structures 8

**Figure-3.1:** Architectural design of gymnasium in Halstenbek, Germany 11

**Figure-3.2:** Extraction of geometry for roofing from the ellipsoid 11

**Figure-3.3:** plan and front views of roofing geometry 12

**Figure-3.4:** Creation of shallow arches 12

**Figure-3.5:** Creation of diagonal cables 12

**Figure-3.6:** schematic representation of type of joints used in the mechanical model 13

**Figure-3.7:** schematic representation of support conditions 13

**Figure-3.8:** simplified wind loading case considered in the analysis 15

**Figure-4.1:** Formation of complete geometry from \(1/4\) model 17

**Figure-4.2:** Discretized model of shell structure 19

**Figure-4.3:** Support conditions applied to shell structure 19

**Figure-4.4:** Dead load \& Wind load applied to the shell structure 20

**Figure-5.1:** 1st eigenmode shape due to application of dead \& wind load 22

**Figure-5.2:** Nonlinear buckling shape due to dead load of the structure 24

**Figure-5.3:** Nonlinear buckling shape due to dead \& wind load on the structure 24

**Figure-5.4:** Deflection plot due to applied symmetric imperfection on dead load case 26

**Figure-5.5:** Deflection plot due to applied unsymmetric imperfection on dead \& wind load case 26

**Figure-5.6:** Deflection plot due to applied unsymmetric imperfection on dead load case 27

**Figure-5.7:** Deflection due to applied unsymmetric imperfection on dead \& wind load case 27

**Figure-6.1:** Critical load \(Vs\) Prestressing in the cable 30

**Figure-6.2:** Critical load \(Vs\) Stiffness in the basis beam 31

**Figure-6.3:** Critical load \(Vs\) Stiffness in the basis beam 32

**Figure-6.4:** Critical load \(Vs\) arch rise of the shell 33

**Figure-6.5:** Critical load \(Vs\) thickness of the shell 34

**Figure-6.6:** Critical load \(Vs\) geometric imperfection scaling factor 36
List of Tables

Table-5.1: Critical Load factor from different type analyses 28
Table-6.1: Critical load factor Vs Prestressing in the cables 30
Table-6.2: Critical load Vs Stiffness in the joints 31
Table-6.3: Critical load Vs Stiffness in the basis beam 32
Table-6.4: Critical load Vs arch rise of the shell 33
Table-6.5: Critical load Vs thickness of the shell 34
Table-6.6: Application of geometric imperfections 35
Table-6.7: Critical load Vs geometric imperfection scaling factor 36
Table-6.8: Optimized design parameters 37
## Index

*Acknowledgement*  i

*Conceptual Formulation*  ii

*List of Figures*  iii

*List of Tables*  iv

1. **Introduction**  1

2. **Primary Background**  3

   2.1 Buckling Mechanism  3

   2.2 Characteristic Remarks on Buckling and Buckling Analysis  5

   2.3 Imperfection Sensitivity  8

3. **Construction Details**  11

   3.1 Geometrical details  11

      3.1.1 Glass roofing  11

      3.1.2 Shallow arches  12

      3.1.3 Diagonal cables  12

      3.1.4 Types of Joints  13

   3.2 Support conditions  13

   3.3 Material and geometrical data  14

   3.4 Loading  15

4. **Finite Element Discretization of Shell Structure**  17

   4.1 Computational consideration in geometrical modeling of shell structure  17

   4.2 Meshing  17

      4.2.1 Basis Beam & Shallow arches (BEAM4)  18

      4.2.2 Prestressed Cables (LINK10)  18

      4.2.3 Glass dome of the structure (SHELL43)  18

      4.2.4 Pin jointed truss elements (LINK8)  18

   4.3 Boundary Conditions:  19

5. **Numerical Simulation of Shell Buckling**  21

   5.1 Eigenvalue buckling analysis  21

   5.2 Geometrical nonlinear buckling analysis  22

   5.3 Nonlinear analysis of the imperfect shell  25
6. Investigation of Stability Behavior due to Changes in Design Parameters

6.1. Sensitivity of stability behavior due to changes in prestressing in cables

6.2. Sensitivity of stability behavior due to changes in bending stiffness of the joints

6.3. Sensitivity of stability behavior due to changes in stiffness of basis beam

6.4. Sensitivity of stability behavior due to changes in arch rise of the shell

6.5. Sensitivity of stability behavior due to changes in thickness of the shell

6.6. Sensitivity of stability behavior due to application of geometric imperfections

6.7. Imperfection Sensitivity due to changes in geometric imperfection scaling factor

6.8. Optimized design parameters of the structure

Conclusions

Appendix
A. Ansys Input Codes

References
1. Introduction

Shell structures are widely used in the fields of civil, mechanical, architectural, aeronautical, and marine engineering. Shell technology has been enhanced by the development of new materials and prefabrication schemes. In the era of architectural evaluation the modern design becomes more and more the decisive influencing factor for construction of buildings. In this context engineers have to face the challenge of developing constructions on the limit of best available technology. Such a special challenge is the realization of shallow shell structures e.g. used for roofing of sports facilities. Such shallow shells look gorgeous and can be very efficient for taking membrane forces. Often these shell structures are very sensitive to stability failure; snap through problems because of small arch rise. One more factor taken into consideration is imperfections, which have noteworthy influence on the buckling behavior of shell structures.

Stability is one of the very important properties of both static and dynamic systems in equilibrium. The investigation of stability concerns with what happens to the structure when it is slightly disturbed from its equilibrium position: ‘‘Does it return to its equilibrium position, or does it depart further?’’

In our project a geometrically nonlinear analysis for the glass roofing of a gymnasium in Halstenbek, Germany has been considered. It is a small city near Hamburg. It was collapsed two times during its construction stage. The interesting background information will show the importance of nonlinear and stability problems in reality. Construction was started in September’1995. At assembling state of steel-glass dome, first collapse of steel construction was occurred in February 1997 during a normal storm, because the diagonal cables were partially installed, but not prestressed which leads to unstable state of glass grid dome. The administration decided to rebuild the collapsed structure in January 1998. After 5 months again the next collapse occurred caused by an abrupt stability failure of steel glass dome. Investigation reveals that the failure is caused mainly due to the following reasons:

- Unfavourable support conditions for the membrane shell, which leads to no equilibrium of forces.
- Reduced stiffness in joints, because of smaller cross sectional area of mounting links which leads to higher bending forces than membrane forces.
• Imperfections which were distributed over the complete structure and exceeded the tolerances, which leads to decrease in ultimate load of roofing.

For proof of the load carrying capacity and the resulting safety margins of the structure, it is necessary to know the load limits for the stable equilibrium of the shell structure resulting from the basic design requirements of the application. The complexity of the buckling problem is a consequence of the cumulative effects of a number of influential variables resulting from the constructional details like shell geometry, support conditions, type of loading etc. The subsequent involved numeric simulation of the buckling problem is outlined in this work. The finite element method (FEM) is the optimal numerical analysis method to use for the simulation and an indispensable tool to perform an accurate computational three-dimensional analysis of the buckling behavior.

Predicting the buckling response of thin shells in structural simulations is difficult because most models do not include the physical characteristics of the problem that initiate instabilities. For shell structures the character of the buckling and load levels that lead to instability are governed by the non-uniformities or imperfections in either the structure or loading. A methodology should be developed to accurately predict the buckling response of the thin shell structures by incorporating either the measured imperfections in the structure and loading, or accurate statistical approximations to the imperfections.
2. Primary Background

2.1 Buckling Mechanism:

Buckling of bars, frames, plates, and shells may occur as a structural response to membrane forces. Membrane forces act along member axes and tangent to the shell midsurfaces. In general, a shell simultaneously displays bending stresses and membrane stresses. Bending stresses in a shell produce bending and twisting moments:

\[ M_x = \int_{-l/2}^{l/2} \sigma_x z \, dz; \quad M_y = \int_{-l/2}^{l/2} \sigma_y z \, dz; \quad M_{xy} = \int_{-l/2}^{l/2} \tau_{xy} z \, dz \]  
(2.1)

Membrane stresses correspond to stresses in a plane stress problem: they act tangent to the mid surface, and produce midsurface-tangent forces per unit length. The membrane forces:

\[ N_x, N_y \text{ and } N_{xy} \]  
are given by

\[ N_x = \int_{-l/2}^{l/2} \sigma_x z \, dz; \quad N_y = \int_{-l/2}^{l/2} \sigma_y z \, dz; \quad N_{xy} = \int_{-l/2}^{l/2} \tau_{xy} z \, dz \]  
(2.2)

Where \( x \) and \( y \) are orthogonal coordinates in the midsurface and \( z \) is a direction normal to the midsurface. Stresses in the shell are composed of a membrane component \( \sigma_m \) and a bending component \( \sigma_b \). A shell can carry a large load if membrane action dominates over bending, as a thin wire can carry a large load in tension but only a small load in bending. Practically, it is not possible to have only membrane action in a shell. Bending action is also present if concentrated loads are applied, if supports apply moments or transverse forces, or if a radius of curvature changes abruptly.

The property of thinness of a shell wall has a consequence as the following: The membrane stiffness is in general greater than the bending stiffness. A thin shell can absorb a great deal of membrane strain energy without deforming too much. It must deform much more in order to absorb an equal amount of bending strain energy. If the shell is loaded in such a way that most of its strain energy is in the form of membrane action, and if there is a way that this stored membrane energy can be converted into bending energy, the shell may fail rather dramatically in a process called "buckling", as it exchanges its membrane energy into bending energy. Large deflections are generally required to convert a given amount of membrane energy into bending energy.
One can also take the view that membrane forces alter the bending stiffness of a structure. Thus buckling occurs when compressive membrane forces are large enough to reduce the bending stiffness to zero for some physically possible deformation mode. If the membrane forces are reversed—that is, made tensile rather than compressive—bending stiffness is effectively increased. This effect is called stress stiffening. Stress stiffening (also called geometric stiffening, incremental stiffening, initial stress stiffening, or differential stiffening) is the stiffening (or weakening) of a structure due to its stress state. This stiffening effect normally needs to be considered for thin structures with bending stiffness very small compared to axial stiffness, such as cables, thin beams, and shells and couples the in-plane and transverse displacements. This effect also augments the regular nonlinear stiffness matrix produced by large strain or large deflection effects. The effect of stress stiffening is accounted for by generating and then using an additional stiffness matrix, hereinafter called the “stress stiffness matrix”, \([K_\sigma]\). The effect of membrane forces are accounted for by this stress stiffness matrix. The stress stiffness matrix is added to the regular stiffness matrix in order to give the total stiffness. This type of formulation is called classical stability formulation.

In what follows we emphasize classical buckling analysis, which uses \([K_\sigma]\). One begins by applying to the structure a reference level of loading \([P^0]\) and carrying out standard linear static analysis to obtain membrane stresses in elements. Hence, we generate a stress stiffness matrix \([K_\sigma^0]\) appropriate to \([P^0]\). For another load level, with \(\lambda\) a scalar multiplier,

\[
[K] = \lambda [K_\sigma^0]
\] (2.3)

When \(\{P\} = \lambda \{P^0\}\) (2.4)

The above equations imply that multiplying all loads \(P_i\) in \([P^0]\) by \(\lambda\) also multiplies the intensity of the stress field by \(\lambda\) but does not change the distribution of stresses. Then, since external loads do not change during an infinitesimal buckling displacement \(\{d\psi\}\),

\[
[[K_\sigma] + \lambda \{K_\sigma^0\}]\{\psi\} = [[K_\sigma] + \lambda \{K_\sigma^0\}]\{\psi + d\psi\} = \lambda \{P^0\}
\] (2.5)
Subtraction of the first equation from the second yields

\[
(\{K_c\} + \lambda_{cr}\{K_\sigma\})\{d\psi\} = 0
\]  

(2.6)

The above equation defines an eigenvalue problem whose lowest eigenvalue \(\lambda_{cr}\) is associated with buckling. The critical or buckling load is, from the equation (2.4),

\[
\{P_{cr}\} = \lambda\{P^0\}
\]  

(2.7)

The eigenvector \(\{d\psi\}\) associated with \(\lambda_{cr}\) defines the buckling mode. The magnitude of \(\{d\psi\}\) is indeterminate. Therefore \(\{d\psi\}\) identifies shape but not amplitude.

A physical interpretation of equation (2.6) as follows. Terms in parentheses in equation (2.6) comprise a total or net stiffness matrix \([K_{net}]\). Since forces \([K_{net}]\{d\psi\}\) are zero, one can say that membrane stresses of critical intensity reduce the stiffness of the structure to zero with respect to buckling mode \(\{d\psi\}\). Numerous computational methods for determining \(\lambda_{cr}\) are available in the literature. Eigenvalue extraction methods are widely used to calculating the critical or buckling load.

2.2 Characteristic Remarks on Buckling and Buckling Analysis:

A real structure may collapse at a load quite different than that predicted by a linear bifurcation buckling analysis. Figure-2.1(a) illustrates some of the ways a structure may behave. Here \(P\) is either the load or is representative of its magnitude, and \(D\) is displacement of some d.o.f. of interest. In Figure -2.1(a) the primary or prebuckling path happens to be linear. At bifurcation, either of two adjacent and infinitesimally closed equilibrium positions is possible. Thereafter, for \(P > P_{cr}\), a real (imperfect) structure follows the secondary path. The secondary (postbuckling) path rises, which means that the structure has postbuckling strength. In this case \(P_{cr}\) characterizes a local buckling action that has little to do with the overall strength. This structure finally collapses at a limit point, which is defined as a relative maximum on the Load Vs Displacement curve for which there is no adjacent equilibrium position. General terminology may refer to the limit point load as a buckling load. The action at collapse becomes dynamic,
because the slope of the curve becomes negative and the structure releases elastic energy (which is converted into kinetic energy).

![Diagram](image)

**Figure-2.1:** Possible load Vs displacement behaviors of thin walled structure

At different type of behavior is depicted in Figure-2.1(b). Here the perfect means (Idealized) structure has a nonlinear primary path. The postbuckling path falls, so there is no postbuckling strength. If the primary path is closed to a falling secondary path, the structure is called *imperfection sensitive*, which means that the collapse load of the actual structure is strongly affected by small changes in direction of loads, manner of support, are changes in geometry. The actual structure, which has imperfections, displays a limit point rather than bifurcation, as shown by dashed line.

The buckling and postbuckling behavior of columns, plates and shells under axial compression will be discussed for perfect and imperfect structures as a very former background. The motivation is visualize the mechanism and imperfection sensitivity of different structures.

The buckling loads of perfect structures are characterized by bifurcation of the equilibrium states. For a column after buckling the load remains constant while increasing by a smaller slope for increasing axial shortening in case of a plate as in Figure-2.2. A column exhibits constant *postbuckling load*–type behavior and plate has an *increasing load carrying capacity* in postbuckling region. In case of a thin–walled cylinder, after buckling the load decreases
suddenly in Figure-2.2(c). The wall of the cylinder jumps into a new equilibrium configuration consisting of buckles of finite amplitude and reaches to a load far below the buckling load. The post critical load decreases indicating a *decreasing load carrying capacity*.

Several different postbuckling patterns as one–tier, two–tiers, three–tiers, diamond pattern and corresponding postbuckling curves are possible due to the different loading case e.g. axial compression, external/internal/hydrostatic pressure.

The buckling load of perfect thin–walled structures is different than the thin–walled structures with initial geometrical imperfections. Although the buckling and postbuckling behavior of shells show huge differences for perfect and imperfect ones, while for columns and plates this difference is slight. This is due to the fact that thin–walled shells are sensitive to initial imperfections. In contrast to the perfect column a buckling load is not obviously defined for the imperfect column.

The load–deformation curve which approximates asymptotically to the curve which belongs to the perfect column by an increasing axial shortening and continuously varying slope. The behavior of the imperfect plate is similar to the imperfect column’s response. The reason of the reduction in buckling load is the bending stresses coming due to the existence of initial imperfections. This decrease is less in case of a plate since the postbuckling loads of the perfect plate reach the buckling load.

The response of imperfect thin–walled cylindrical shell is completely different from that of the imperfect column and plate. After a maximum axial load the curve passes a point with a vertical tangent meaning a maximum of axial shortening. This maximum load is the buckling load of the imperfect cylinder which is always lower than that of the perfect one. At this point, the equilibrium becomes unstable and the wall of the cylinder snaps through into a new equilibrium state consists of large deformations and a lower axial load than before.
In the buckling process, the difference between the buckling pattern and the postbuckling pattern should be distinguished. The radial displacements of the buckling pattern are infinitesimally small and cannot be seen by naked eye. Moreover, the buckling pattern is usually unstable. Therefore as soon as the buckling load is reached, the shell snaps into a stable periodic postbuckling pattern with visible buckles of finite depth.

2.3. Imperfection Sensitivity:

In reality, many shells fail at a smaller load than the theoretical critical load. The reason of this early collapse is the effect of small, unavoidable geometrical imperfections. After reaching the critical states, the load rapidly decreases while deflections increase meaning that the structure undergoes softening. Furthermore, a small disturbance can cause the shell to jump to a postbuckling state at which the carrying capacity is greatly reduced. Imperfections may come from
The fact that no member of a structure can be constructed perfectly.

The inability to ensure that the load actually act geometrically perfect.

In Fig. 2.2 (a–c) the load $N_x$ is plotted against the buckling displacement $w$ which is perpendicular to the shell surface. Fig. 2.2 (a) shows that after reaching the critical load, the load carrying capacity of the axially compressed structure remains constant. Initial imperfections increase the deformations, but curves will have no peak points, approaching to the horizontal line of the perfect one asymptotically.

Fig. 2.2 (b) shows the increasing load carrying capacity in the postbuckling region. In case of a perfect plate there is a definite critical load at which buckling occurs but for an imperfect plate there exists no certainly defined critical buckling load. The buckling deformation increases with increasing load. As a consequence this structure is imperfection–insensitive.

For axially compressed cylinders described in Fig. 2.2 (c), the load carrying capacity decreases after reaching a certain critical load and snaps through a lower load level. These kinds of structures are very imperfection–sensitive.

As Koiter showed that the imperfection sensitivity of cylindrical shell structures is closely related with their initial postbuckling behavior. In case of a plate, i.e. when early postbuckling path has a positive curvature the structure is able to develop considerable postbuckling strength, and loss of stability of the primary path does not result in structural collapse as in Fig. 2.2 (b).

For a cylinder which has a negative slope in the initial postbuckling region, buckling will occur violently and buckling load depends on the initial imperfections. Consequently, sensitivity of the structure to imperfections is affected by early postbuckling behavior and the shape of the postbuckling equilibrium path plays a central role in determining the influence of initial imperfections.

Thicker shells appear to be less sensitive to imperfections than thinner shells. As the structure becomes thinner the sensitivity to imperfections increases. Shells subjected to external pressure are less sensitive to imperfections than are shells subjected to axial compression because the wave lengths of buckles in axial direction are longer in the former case and eigenvalues do not cluster around the critical value. Hence, very small local imperfections do not affect the critical pressure as much as they affect the critical axial load and, a very small local imperfection can tend to trigger premature failure. One of the reasons of high imperfection sensitivity of these highly symmetrical systems comes from the facts that many different buckling modes are associated with more or less same eigenvalues. For simulations of imperfect shells it is an
important issue to decide about the imperfection amplitude and form to simulate an almost real behavior. Initial geometric imperfections can be either axi-symmetric or asymmetric and either in shape of classical buckling mode or in shape of diamond buckling pattern and as combinations of different possibilities.
3. Construction Details

At first it is required to develop a mechanical model for the steel glass roofing by abstraction of the real roof to its significant structural elements. To reduce the computational effort it is required to simplify the model as much as possible, but without disregarding important characteristics of the basic system.

3.1. Geometrical details:

In the Figure-3.1 we can see the architectural design of the gymnasium in Halstenbek, Germany. The glass roofing has an elliptical horizontal projection with a length of 46m and a width of 28m. The shell structure is parabolic in both directions with a height of 4.6m. The roof is composed of shallow steel arches at intervals of about 1.2m and prestressed steel cables, which are diagonal, and attached to arches as well as the glass elements, which form the exterior shell.

![Figure-3.1: Architectural design of gymnasium in Halstenbek, Germany](image)

3.1.1. Glass roofing:

The roofing has been approximated by ellipsoidal segment as shown in Figure-3.2

![Figure-3.2: Extraction of geometry for roofing from the ellipsoid](image)
- see details in the attached plan and front views of the roofing

Figure-3.3: plan and front views of roofing geometry

3.1.2. Shallow arches:
- shallow arches perpendicular to x-direction at intervals of 1,20m (38 arches)
- shallow arches perpendicular to z-direction at intervals of 1,12m (25 arches)

41 arches perpendicular to x-direction      25 arches perpendicular to z-direction

Figure-3.4: Creation of shallow arches

3.1.3. Diagonal cables:
- Diagonal cables 45° and -45° to the x-direction

47 diagonals under 45°         47 diagonals under –45°

Figure-3.5: Creation of diagonal cables
3.1.4. Types of Joints:
In the nonlinear and stability analysis of this shallow shell structure we are considering three types of joints for horizontal and vertical shallow arches with the prestressing cables:
Joint-1: Bending resistant joints
Joint-2: pin joints
Joint-3: joints with decreased stiffness

![Bending resistant jointed beam elements](image1)

![Joints with decreased stiffness](image2)

*Figure-3.6:* schematic representation of type of joints used in the mechanical model

3.2. Support conditions:
- surrounding basic beam is sliding supported on concrete shell substructure in vertical direction at intervals of 1.2-1.5 m; the basic beam can not rotate in the supports
- in horizontal direction the surrounding basic beam is supported only in angular points
- see support conditions in Figure-6

![Support conditions](image3)

*Figure-3.7:* schematic representation of support conditions
### 3.3. Material and geometrical dimensions of components:

- **Shallow arch elements:**
  
  **(Rectangular section)**
  
  - flat bar: Steel S 355
  
  - dimensions: 40 x 60 [mm]
  
  - $A = 2400 \text{ mm}^2$
  
  - $I_z = 720000 \text{ mm}^4$
  
  - $I_y = 320000 \text{ mm}^4$
  
  - $I_T = 755000 \text{ mm}^4$
  
  - $E = 210000 \text{ N/mm}^2$
  
  - $\sigma_y = 360 \text{ N/mm}^2$
  
  - $\nu = 0.3$
  
  - $\rho = 7850 \text{ kg/m}^3$
  
  - $\alpha_T = 1.2 \times 10^{-5} \text{ K}$

  - **decreased stiffness**
    
    - **joints:** 2 x 60 x 7.5
    
    - $A' = 900 \text{ mm}^2$
    
    - $I_z' = 270000 \text{ mm}^4$
    
    - $I_y' = 241900 \text{ mm}^4$

  - **(Schlaich)**

- **Basis beam:**
  
  **(Rectangular section)**
  
  - flat bar: Steel S 355
  
  - dimensions: 220 x 40 [mm]
  
  - $A = 8800 \text{ mm}^2$
  
  - $I_z = 1170000 \text{ mm}^4$
  
  - $I_y = 35490000 \text{ mm}^4$
  
  - $I_T = 125430000 \text{ mm}^4$
  
  - $E = 210000 \text{ N/mm}^2$
  
  - $\sigma_y = 360 \text{ N/mm}^2$
  
  - $\nu = 0.3$
  
  - $\rho = 7850 \text{ kg/m}^3$
  
  - $\alpha_T = 1.2 \times 10^{-5} \text{ K}$

- **Steel cable:**
  
  - 1 x 37 lacing
  
  - dimensions: 2 x $\varnothing 6$ [mm]
  
  - $A = 44.8 \text{ mm}^2$
  
  - $E = 160000 \text{ N/mm}^2$
  
  - $\sigma_y = 1570 \text{ N/mm}^2$
  
  - $\rho = 7850 \text{ kg/m}^3$
  
  - $\alpha_T = 1.2 \times 10^{-5} \text{ K}$

  - **prestress** = 800
- glass roof: 
  \( t = 18 \text{ mm (thickness)} \)
  \( E = 70000 \text{ N/mm}^2 \)
  \( \nu = 0.23 \)
  \( \rho = 2500 \text{ kg/m}^3 \)
  \( \alpha_T = 0.9 \times 10^{-5} \text{ K} \)

### 3.4. Loading:
(according to German standard)

- **Load case 1: dead load**
  
  **shallow arch elements:**
  
  \( G = 78.5 \text{ kN/m}^3 \)
  \( 0.04 \text{ m} \times 0.06 \text{ m} = 0.1884 \text{kN/m} \)
  
  \( g = 2 \times 0.1884 \text{kN/m} / 1.20 \text{ m} = 0.314 \text{kN/m}^2 \)

  **steel cable:**
  
  \( G = 78.5 \text{ kN/m}^3 \)
  \( 44.8 \times 10^{-6} = 0.0035 \text{kN/m} \)
  
  \( g = 2 \times [0.0035 \text{kN/m} \times (\sqrt{2} \times 1.2 \text{ m})] / (1.2 \text{ m})^2 \)
  
  \( = 0.008 \text{kN/m}^2 \)

  **glass roof:**
  
  \( g = 25 \text{ kN/m}^3 \times 0.018 \text{ m} \)
  
  \( = 0.450 \text{kN/m}^2 \)

  **sum:**
  
  \( = 0.772 \text{kN/m}^2 \)

  **2% addition for small parts:**
  
  \( 1.02 \times 0.774 \text{kN/m}^2 = 0.788 \text{kN/m}^2 \)

- **Load case 2: wind (simplified)**
  
  **Compression coefficient \( c_p \):**

  *Figure-3.8: simplified wind loading case considered in the analysis*

  - **building height above site:** \( < 8 \text{ m} \)
  - **impact pressure:** \( q = 0.5 \text{kN/m}^2 \)
  - **circumscribed rectangle ground plan:** \( h/b < 5; h/a < 5 \)
  - **force coefficient:** \( c_f = 1.3 \)
- compression: 
  \[ q_w = 0.3 \cdot c_r \cdot q = 0.195 \text{kN/m}^2 \]
- suction: 
  \[ q_w = 0.7 \cdot c_r \cdot q = 0.455 \text{kN/m}^2 \]

- Imperfections:
  applied according to eigenforms

- Load combination: dead load + wind
  
  1.35 \cdot \text{dead load} + 1.5 \cdot \text{wind}
4. Finite Element Discretization of Shell Structure

4.1. Computational consideration in geometrical modeling of shell structure:

In order to develop the mechanical model for the glass steel grid dome, like creation of shallow arches, prestressing cables, joints, basis beam, basis shell and shell elements for roofing we need to use a lot of boolean operations in ANSYS/Multiphysics. Generally boolean operations in the modeling need high computational effort. By taking this fact into consideration, computational effort is drastically reduced by considering the 1/4\textsuperscript{th} part of the glass dome and apply all the necessary boolean operations to obtain the mechanical model. Now in order to get the complete model, by simply reflecting this quarter part to other directions as shown in Figure-4.1. Because stability analysis requires the complete geometry.

![Figure-4.1: Formation of complete geometry from ¼ model](image)

4.2. Meshing:

The point of interest is to know the global structural stability behavior of the steel glass grid dome. To obtain the accurate results, we should choose appropriate finite element for the respective structural member in order to represent the exact physical behavior of the structural component in the FE model. To obtain good meshing, smooth transition of elements is maintained in different components of the structure, i.e. the node connectivity should be appropriate, which leads to better results.
4.2.1. Basis Beam & Shallow arches (BEAM4):

BEAM4 might be a suitable element for nonlinear and buckling finite element modeling of Basis beam and Shallow arches, because it is a uniaxial element with tension, compression, torsion, and bending capabilities. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes. Stress stiffening and large deflection capabilities are included. A consistent tangent stiffness matrix option is available for use in large deflection analyses.

4.2.2. Prestressed Cables (LINK10):

LINK10 element could be best suitable for FE modeling of prestressing cables. LINK10 is a three-dimensional spar element having the unique feature of a bilinear stiffness matrix resulting in a uniaxial tension-only (or compression-only) element. With the tension-only option, the stiffness is removed if the element goes into compression (simulating a slack cable or slack chain condition). This feature is useful for static cable wire applications where the entire cable wire is modeled with one element. LINK10 has three degrees of freedom at each node: translations in the nodal x, y, and z directions. No bending stiffness is included in either the tension-only (cable) option or the compression-only (gap) option. The element is not capable of carrying bending loads. The stress is assumed to be uniform over the entire element.

4.2.3. Glass Dome of the Structure (SHELL43):

SHELL43 could be well suited to model the Shallow Shell member, because it is used for linear, warped, moderately-thick shell structures. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes. The deformation shapes are linear in both in-plane directions. For the out-of-plane motion, it uses a mixed interpolation of tensorial components. The element has plasticity, creep, stress stiffening, large deflection, and large strain capabilities.

4.2.4. Pin Jointed Truss Elements (LINK8):

In the case of pin jointed structures, we are assuming that the structural members do not carry the bending loads. In such situations, LINK8 could be the best choice. LINK8 is a spar which may be used in a variety of engineering applications. This element can be used to model trusses,
sagging cables, links, springs, etc. The three-dimensional spar element is a uniaxial tension-compression element with three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element is not capable of carrying bending loads. The stress is assumed to be uniform over the entire element.

**Figure-4.2:** Discretized model of shell structure

### 4.3. Boundary Conditions:

An accurate modelling of the boundary conditions is very important in order to get the correct results from the FE-analysis. The boundary conditions should be modelled as close to the real-life situation as possible. In order to predict the global structural stability behaviour of the shell structure, applied support conditions are as follows. Surrounding basic beam is sliding supported on concrete shell substructure in vertical direction at intervals of 1.2-1.5 m, and the basic beam can not rotate in the supports and in horizontal direction the surrounding basic beam is supported only in angular points as shown in Figure-

**Figure-4.3:** Support conditions applied to shell structure
In the stability analysis of the shallow shell structure we have been considered two types of loads cases as dead load and simplified wind load. First we apply dead load as shown by red colored arrows and as a wind load we have been applied compressive pressure load on half of the structure as shown by blue colored contour and on the other side suction pressure load as shown by red colored contour in the Figure-4.4.

*Figure-4.4:* Dead load & Wind load applied to the shell structure
5. Numerical Simulation of Shell Buckling

Within the last decade, it is reviewed that, with the evaluation of finite element analysis of stability problems, the advance of powerful computers and highly competent numerical techniques has come to a state where any given shell structure can be calculated—no matter how complicated the geometry, how dominant the imperfection influence and how nonlinear the load carrying behavior is. However, the main task of the design engineer is, more than ever, to model his shell problem properly and to convert the numerical output into the characteristic buckling strength of the “real” shell which is needed for an equally safe and economic design.

The following observations are concerned with different kinds of numerical approaches to shell buckling when using a commercial FEM package like ANSYS.

5.1. Eigenvalue buckling analysis:

Eigenvalue buckling analysis predicts the theoretical buckling strength (the bifurcation point) of an ideal linear elastic structure. This method corresponds to the textbook approach to elastic buckling analysis: for instance, an eigenvalue buckling analysis of a column will match the classical Euler solution. However, imperfections and nonlinearities prevent most real-world structures from achieving their theoretical elastic buckling strength. Thus, eigenvalue buckling analysis often yields unconservative results, and should generally not be used in actual day-to-day engineering analyses. The appropriate eigenvectors \( \{\psi\} \) are representing the buckling shapes (deformation pattern which is characterizing the alternative response paths). These eigenvectors are giving \textit{qualitative information about the buckling shape} but not quantitative information. This restriction to qualitative information is a fundamental characteristic of eigenvalue problems.

The lowest eigenvalue load of the perfect shell is needed as critical buckling resistance \( P_{cr} \) for the simplest numerically based design approach. It is generally no problem to produce this eigenvalue for a given FE model. However, it should be very cautious about the elementary mistakes which are made again and again when defining the FE model of a given shell buckling, such as the level of discretization i.e. the number of finite elements used in the analysis, type of boundary conditions, loading cases etc.,
Points to be considered in eigenvalue buckling analysis with ANSYS:

- Only linear behavior is valid. Nonlinear elements, if any, are treated as linear. If we include contact elements, for example, their stiffnesses are calculated based on their status after the static prestress run and are never changed.
- Stiffness in some form must be defined. Material properties may be linear, isotropic or orthotropic, and constant or temperature-dependent. Nonlinear properties, if any, are ignored.

Figure-5.1: 1st eigenmode shape due to application of dead & wind load

5.2. Geometrical nonlinear buckling analysis:

Usually the linear bifurcation analysis discussed above is sufficient to produce a reliable critical buckling resistance $P_{cr}$ as the numerical basis for a traditional reduction factor shell buckling design. However, in some cases a geometrically nonlinear elastic analysis (i.e. large deflection theory) should be used to calculate the prebuckling path of the perfect shell configuration to which the eigenvalue search is applied. Nonlinear buckling analysis is usually the more accurate approach and is therefore recommended for design or evaluation of actual structures. This technique employs a nonlinear static analysis with gradually increasing loads to seek the load
level at which our structure becomes unstable. Using the nonlinear technique, our model can include features such as initial imperfections, plastic behavior, gaps, and large-deflection response.

In nonlinear buckling analysis, we have to consider the complete nonlinear part of the tangential stiffness matrix, i.e. $[K_n]$. For nonlinear buckling analysis large computation power is required, because in this analysis complete nonlinear tangential stiffness matrix is considered and the problem should be solved iteratively to know the load level at which the structure has to be collapsed. Such a nonlinear bifurcation FE analysis or nonlinear buckling analysis requires considerable experience of the user.

**Points to be considered in nonlinear buckling analysis with ANSYS:**

- A nonlinear buckling analysis is a static analysis with the consideration of large deflections, extended to a point where the structure reaches its limit load or maximum load. Other nonlinearities such as plasticity may be included in the analysis.

- The basic approach in a nonlinear buckling analysis is to constantly increment the applied loads until the solution begins to diverge. It is required to use a sufficiently fine load increment as our loads approach the expected critical buckling load. If the load increment is too coarse, the buckling load predicted may not be accurate.

- It is important that an unconverged solution does not necessarily mean that the structure has reached its maximum load. It could also be caused by numerical instability, which might be corrected by refining our modeling technique. Track the load-deflection history of your structure's response to decide whether an unconverged load step represents actual structural buckling, or whether it reflects some other problem.

- If the loading on the structure is perfectly in-plane (that is, membrane or axial stresses only), the out-of-plane deflections necessary to initiate buckling will not develop, and the analysis will fail to predict buckling behavior. To overcome this problem, apply a small out-of-plane perturbation, such as a modest temporary force or specified displacement, to begin the buckling response. (A preliminary eigenvalue buckling analysis of your structure may be useful as a predictor of the buckling mode shape, allowing you to choose appropriate locations for applying perturbations to stimulate the desired buckling
response.) The imperfection (perturbation) induced should match the location and size of that in the real structure. The failure load is very sensitive to these parameters.

**Figure-5.2:** Nonlinear buckling shape due to dead load of the structure

**Figure-5.3:** Nonlinear buckling shape due to dead & wind load on the structure
5.3. Nonlinear analysis of the imperfect shell:

An “exact” FE analysis of the shallow shell configuration, i.e. including all nonlinear geometry, support conditions and loading influences, but with perfect geometry, has become quite popular among researchers. The reasons are in the author’s view: it is a challenge to master all the complicated nonlinear techniques; the computational power to achieve this is available these days; and the unpleasant problems of realistic imperfections are avoided. However, real shell structures are imperfect—unfortunately.

From the foregoing explanations it may be concluded that, for a specific design case, a fully nonlinear analysis of the perfect shell is physically reasonable only for a relatively thick-walled shell for which a pronouncedly yield-induced collapse mechanism is expected.

The behavior of an imperfect shell can only be simulated by analyzing the imperfect shell itself. However, this logical perception leads inevitably to the question of how the imperfections of a real shell structure look. Without going into any detail, it may be stated that, no matter how sophisticated a numerical imitation of an imperfection field may be, it still represents merely a “substitute imperfection” because certain components of the real imperfections (e.g. residual stresses, inhomogenities, anisotropies, loading and boundary inaccuracies) are eventually not included and must therefore be “substituted” in the simulated imperfection model. Usually these substitute imperfections are introduced in the form of equivalent geometric imperfections. In literature numerous techniques are published by so many researchers on how to consider the imperfections in numerical simulations.

Among all of these techniques, in our nonlinear imperfection simulations, worst geometric imperfections technique has been considered. The idea to find the “worst possible” geometric imperfection pattern for a given, to-be-designed shell structure and to introduce it into the nonlinear analysis is as old as the discovery of the detrimental influence of imperfections. It was common practice from the beginning, supposedly taken over from column and plate buckling experience, to consider that imperfection pattern to be the worst which is affine to the lowest eigenmode.
Figure-5.4: Deflection plot due to applied symmetric imperfection on dead load case

Figure-5.5: Deflection plot due to applied symmetric imperfection on dead & wind load case
**Figure-5.6:** Deflection plot due to applied unsymmetric imperfection on dead load case

**Figure-5.7:** Deflection plot due to applied unsymmetric imperfection on dead & wind load case
<table>
<thead>
<tr>
<th>Analysis type</th>
<th>Dead Load Factor</th>
<th>Wind Load Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Eigen buckling analysis</td>
<td>-</td>
<td>0.88930</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.85502</td>
</tr>
<tr>
<td>Nonlinear buckling analysis</td>
<td>-</td>
<td>0.64646</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.60287</td>
</tr>
<tr>
<td>Imperfections</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Scaling factor: 0.002</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Symmetric</td>
<td>0.60844</td>
<td>0.56428</td>
</tr>
<tr>
<td>Unsymmetric</td>
<td>0.60100</td>
<td>0.56695</td>
</tr>
</tbody>
</table>

*Table-5.1: Critical Load factor from different type analyses*
6. Investigation of Stability Behavior due to Changes in Design Parameters

In reality the gymnasium located in Halstenbek, Germany which we considered in our simulations, was collapsed two times. First collapse was happened at construction stage. Reason for this collapse was, that the diagonal cables were partially installed, but not prestressed which leads to unstable state of glass grid dome during a normal storm. So the steel frame requires a large number of temporary columns to support the structure in the assembly state. But in reality only 10% of the flat bar joints were supported and this fact led to large imperfections, structure was not able to resist moderate wind loads and was collapsed during a storm.

The second collapse occurred when the glass roofing was completely built. Investigations of experts in structural design reveal that, the failure of the structure is mainly due to the following reasons:

- Unfavourable support conditions for the membrane shell, which leads to no equilibrium of forces. In reality vertical support was used. But in an ideal membrane we have only normal forces. Because of this vertical support, we obtain a reaction force component perpendicular to the membrane layer, which is therewith not in equilibrium.

- Reduced stiffness in joints, because of smaller cross sectional area of mounting links which leads to lower bending stiffness in the joints.

- Imperfections which were distributed over the complete structure and exceeded the tolerances, which leads to decrease in ultimate load of roofing.

From the foregoing discussions it may be concluded that, the structure was collapsed due to unfavorable design parameters of the structure. During this work, we have been trying to optimize the design of glass roofing of gymnasium in Halstenbek with the reasonable changes in support conditions, prestressing in the cables, stiffness in joints, stiffness in basis beam and arch rise of the shell.

6.1. Sensitivity of stability behavior due to changes in prestressing in cables:

The application of load to a structure so as to deform it in such a manner that the structure will withstand its working load more effectively or with less deflection is called prestressing a structure. We can able to increase the stability of the steel shell structure, by the application of prestressing in the cables, which are attached to the steel shallow arches. To finding out the
optimum value of prestressing in the cables we can carry out parametric studies by keeping all other design parameters to default, and changes are made only with the prestress in the cables. Results are shown in Table-6.1

<table>
<thead>
<tr>
<th>Prestress in Cables [N/mm$^2$]</th>
<th>Dead Load Factor</th>
<th>Dead &amp; Wind Load Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.72916</td>
<td>0.71324</td>
</tr>
<tr>
<td>200</td>
<td>0.93048</td>
<td>0.89528</td>
</tr>
<tr>
<td>400</td>
<td>0.91689</td>
<td>0.88206</td>
</tr>
<tr>
<td>600</td>
<td>0.90297</td>
<td>0.86842</td>
</tr>
<tr>
<td>800</td>
<td>0.8893</td>
<td>0.85502</td>
</tr>
<tr>
<td>1000</td>
<td>0.87588</td>
<td>0.84185</td>
</tr>
<tr>
<td>1200</td>
<td>0.8627</td>
<td>0.82892</td>
</tr>
<tr>
<td>1500</td>
<td>0.84333</td>
<td>0.80995</td>
</tr>
</tbody>
</table>

*Table-6.1: Critical load factor Vs prestressing in the cables*

*Figure-6.1: Critical load Vs Prestressing in the cable*

**Observations:** Stability behavior of the shallow shell structure is very sensitive with the changes in prestressing in the cables. Observations have been taken by keeping other parameters to default and changes are made only with the prestress in the cables. With zero prestressing the critical load factor is very low and is optimum with 200N/mm$^2$. 
6.2. Sensitivity of stability behavior due to changes in bending stiffness of the joints:

<table>
<thead>
<tr>
<th>Dimensions [mm]</th>
<th>Area[mm$^2$]</th>
<th>Iy[*1000m$^4$]</th>
<th>Iz[*1000m$^4$]</th>
<th>Dead Load</th>
<th>Dead &amp; Wind</th>
</tr>
</thead>
<tbody>
<tr>
<td>L = 60 B = 7.5</td>
<td>900</td>
<td>241</td>
<td>270</td>
<td>0.88930</td>
<td>0.85502</td>
</tr>
<tr>
<td>L = 60 B = 9.375</td>
<td>1125</td>
<td>333.4</td>
<td>337.5</td>
<td>0.92727</td>
<td>0.89305</td>
</tr>
<tr>
<td>L = 75 B = 7.5</td>
<td>1125</td>
<td>302</td>
<td>527</td>
<td>0.94562</td>
<td>0.91130</td>
</tr>
<tr>
<td>L = 80 B = 7.5</td>
<td>1200</td>
<td>322.5</td>
<td>640</td>
<td>0.96844</td>
<td>0.93470</td>
</tr>
<tr>
<td>L = 75 B = 9.375</td>
<td>1406</td>
<td>417</td>
<td>659</td>
<td>0.98246</td>
<td>0.94907</td>
</tr>
<tr>
<td>L = 80 B = 9.375</td>
<td>1500</td>
<td>444.5</td>
<td>800</td>
<td>0.99561</td>
<td>0.96215</td>
</tr>
<tr>
<td>L = 85 B = 10.5</td>
<td>1785</td>
<td>547</td>
<td>1075</td>
<td>1.0173</td>
<td>0.98378</td>
</tr>
<tr>
<td>L = 90 B = 10.5</td>
<td>1890</td>
<td>596</td>
<td>1276</td>
<td>1.0271</td>
<td>0.99349</td>
</tr>
</tbody>
</table>

Table-6.2: Critical load $Vs$ Stiffness in the joints

![Critical Load Factor Vs Bending Stiffness (Z-direction) in Joints](image)

Figure-6.2: Critical load $Vs$ Stiffness in the basis beam

Observations: The stability of the structure is very sensitive with the bending stiffness in the joints. It is observed that for the same cross-sectional areas, the structure is more stable for higher values of bending stiffness in $z$-direction i.e. length direction, compared with the stiffness in width direction.
6.3. Sensitivity of stability behavior due to changes in stiffness of basis beam:

<table>
<thead>
<tr>
<th>Area of the Basis Beam</th>
<th>Dead Load Factor</th>
<th>Wind Load Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>8000</td>
<td>0.88856</td>
<td>0.85413</td>
</tr>
<tr>
<td>8800</td>
<td>0.8893</td>
<td>0.85502</td>
</tr>
<tr>
<td>9500</td>
<td>0.88986</td>
<td>0.8591</td>
</tr>
<tr>
<td>10500</td>
<td>0.89054</td>
<td>0.85673</td>
</tr>
<tr>
<td>13200</td>
<td>0.89508</td>
<td>0.85905</td>
</tr>
<tr>
<td>13750</td>
<td>0.89623</td>
<td>0.86001</td>
</tr>
<tr>
<td>19800</td>
<td>0.90222</td>
<td>0.87032</td>
</tr>
<tr>
<td>26950</td>
<td>0.89839</td>
<td>0.86781</td>
</tr>
<tr>
<td>35200</td>
<td>0.89931</td>
<td>0.87054</td>
</tr>
</tbody>
</table>

Table-6.3: Critical load Vs Stiffness in the basis beam

Figure-6.3: Critical load Vs Stiffness in the basis beam

Observations: Dimensions of basis beam are also influence the stability behavior of the shell structure. We have the optimum value of critical load factor by increasing the dimensions of the rectangular cross-section to a factor of 1.5.
6.4. Sensitivity of stability behavior due to changes in arch rise of the shell:

<table>
<thead>
<tr>
<th>Height of the Shell [m]</th>
<th>Dead Load Factor</th>
<th>Dead &amp; Wind Load Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.6</td>
<td>0.8893</td>
<td>0.85502</td>
</tr>
<tr>
<td>4.8</td>
<td>0.9533</td>
<td>0.91004</td>
</tr>
<tr>
<td>5.0</td>
<td>1.0124</td>
<td>0.96572</td>
</tr>
<tr>
<td>5.2</td>
<td>1.0711</td>
<td>1.01990</td>
</tr>
<tr>
<td>5.4</td>
<td>1.1296</td>
<td>1.07340</td>
</tr>
<tr>
<td>5.6</td>
<td>1.1913</td>
<td>1.12900</td>
</tr>
</tbody>
</table>

*Table-6.4:* Critical load Vs arch rise of the shell

*Figure-6.4:* Critical load Vs arch rise of the shell

**Observations:** It is observed that the stability of the structure is sensitive with the height of the shell as shown in figure. If we are increasing the height, then structure is more stable.
6.5. Sensitivity of stability behavior due to changes in thickness of the shell:

<table>
<thead>
<tr>
<th>Thickness of the Shell [mm]</th>
<th>Dead Load Factor</th>
<th>Dead &amp; Wind Load Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>0.79156</td>
<td>0.81987</td>
</tr>
<tr>
<td>18</td>
<td>0.8893</td>
<td>0.85502</td>
</tr>
<tr>
<td>20</td>
<td>0.91302</td>
<td>0.87609</td>
</tr>
<tr>
<td>25</td>
<td>0.96956</td>
<td>0.92814</td>
</tr>
</tbody>
</table>

**Table-6.5:** Critical load Vs thickness of the shell

**Figure-6.5:** Critical load Vs thickness of the shell

**Observations:** The thickness of the shell also has the influence on the stability behavior of the structure. By increasing the thickness of the shell we have higher critical load factors.
### 6.6. Sensitivity of stability behavior due to application of geometric imperfections:

<table>
<thead>
<tr>
<th>Type of Imperfection</th>
<th>Scaling Factor</th>
<th>Dead Load</th>
<th>Dead &amp; Wind Load</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Symmetrical</strong></td>
<td>0.002</td>
<td>0.608442</td>
<td>0.56428</td>
</tr>
<tr>
<td></td>
<td>0.01</td>
<td>0.51909</td>
<td>0.46242</td>
</tr>
<tr>
<td><strong>Unsymmetrical</strong></td>
<td>0.002</td>
<td>0.60100</td>
<td>0.56695</td>
</tr>
<tr>
<td></td>
<td>0.01</td>
<td>0.49938</td>
<td>0.48100</td>
</tr>
<tr>
<td><strong>With out Imperfection</strong></td>
<td>-</td>
<td>0.60287</td>
<td>0.64646</td>
</tr>
</tbody>
</table>

**Table-6.6:** Application of geometric imperfections

Observations:

1. Load factor decrease with increases in Imperfection
2. Max deflection increases in case of symmetric loading with symmetric imperfection increases as in case of Dead load case
3. With wind, the result is difficult to predict (unsymmetrical imperfections)
4. Imperfection plays a vital role in determining the critical load
5. Imperfection should be included in the stability analysis of the structure
6.7. Imperfection Sensitivity due to changes in geometric imperfection scaling factor:

<table>
<thead>
<tr>
<th>Scaling Factor</th>
<th>Symmetrical Imperfections</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Dead Load Factor</td>
<td>Dead &amp; Wind Load Factor</td>
<td></td>
</tr>
<tr>
<td>0.002</td>
<td>0.60844</td>
<td>0.56695</td>
<td></td>
</tr>
<tr>
<td>0.004</td>
<td>0.58120</td>
<td>0.56018</td>
<td></td>
</tr>
<tr>
<td>0.006</td>
<td>0.56085</td>
<td>0.50558</td>
<td></td>
</tr>
<tr>
<td>0.008</td>
<td>0.53788</td>
<td>0.48012</td>
<td></td>
</tr>
<tr>
<td>0.010</td>
<td>0.51909</td>
<td>0.48100</td>
<td></td>
</tr>
</tbody>
</table>

*Table-6.7: Critical load Vs geometric imperfection scaling factor*

![Imperfection Sensitivity with Different Scaling Factors](image)

*Figure-6.6: Critical load Vs geometric imperfection scaling factor*

6.8. Optimized design parameters of the structure:
<table>
<thead>
<tr>
<th>Component</th>
<th>Original Dimensions</th>
<th>Proposed Dimensions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Basis Beam</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Area ( \text{m}^2 )</td>
<td>8.8e-3</td>
<td>19.8e-3</td>
</tr>
<tr>
<td>( I_{zz} \text{m}^4 )</td>
<td>117.3e-8</td>
<td>594e-8</td>
</tr>
<tr>
<td>( I_{yy} \text{m}^4 )</td>
<td>3549.3e-8</td>
<td>1796.85e-6</td>
</tr>
<tr>
<td>( T_x \text{m} )</td>
<td>4e-2</td>
<td>6e-2</td>
</tr>
<tr>
<td>( T_y \text{m} )</td>
<td>22e-2</td>
<td>33e-2</td>
</tr>
<tr>
<td><strong>Reduced stiffness in joints</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Area ( \text{m}^2 )</td>
<td>0.9e-3</td>
<td>1.89e-3</td>
</tr>
<tr>
<td>( I_{zz} \text{m}^4 )</td>
<td>27e-8</td>
<td>1275.75e-9</td>
</tr>
<tr>
<td>( I_{yy} \text{m}^4 )</td>
<td>24.2e-8</td>
<td>596e-9</td>
</tr>
<tr>
<td>( T_x \text{m} )</td>
<td>6e-2</td>
<td>9e-2</td>
</tr>
<tr>
<td>( T_y \text{m} )</td>
<td>1.5e-2</td>
<td>2.1e-2</td>
</tr>
<tr>
<td>Prestressing in the Cables ( \text{N/mm}^2 )</td>
<td>800</td>
<td>200</td>
</tr>
<tr>
<td>Height of the Shell ( \text{m} )</td>
<td>4.6</td>
<td>5.4</td>
</tr>
</tbody>
</table>

**Table-6.8**: Optimized design parameters

With the new proposed optimum values for geometry, the stability behavior of the structure is improved drastically:

Classical buckling load factor:

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Dead Load case</td>
<td>1.4251</td>
</tr>
<tr>
<td>Dead &amp; Wind Load case</td>
<td>1.3678</td>
</tr>
</tbody>
</table>

With imperfections also it bares up to 0.01* first eigenmode shape from the classical stability analysis.
Conclusions

In the present work, static buckling of shell structures including eigenvalue buckling, nonlinear buckling and imperfection sensitivity due to applied load, support conditions, stiffness in joints and prestressing in the cables are discussed. Finally the design has been optimized with reasonable design parameters: Optimum values for the design parameters have been obtained by keeping other parameters to default and changes are only made with in the required design parameter. It should keep in mind that, in this work all the calculations are carried out with the assumption of linear elastic material behavior. So care should be taken while calculating the optimum design parameters, since the stresses developed in the structural members may some times crosses the yield point.

The prestressing in cables opens more advantages since the initial strain is anticipated and larger stiffness is obtained. And also prestressing in the cables alters the compressive membrane forces in to tensile; ultimately bending stiffness in the structure might effectively increase. It has been observed that, with zero prestressing the critical load factor is very low and is optimum with 200N/mm².

In a thin walled structure such as a shell, membrane stiffness is typically orders of magnitude greater than bending stiffness. Accordingly, small membrane deformations can store a large amount of strain energy, but comparatively large lateral deflections are needed to absorb this energy in bending deformations leads to buckling failures. In reality, cross section at the joints is very low; consequently bending stiffness in joints is also smaller and structure is prone to stability failure. So by increasing the bending stiffness in joints and basis beam, stability has been increased effectively.

The arch rise of the shell structure is also a very big influencing factor on the structural stability. As we know shell can carry a large load if membrane action dominates over bending, so by increasing the height of the shell we can get large membrane stiffness which leads to high snap through point (critical point) for geometrically perfect structures.

Out look: Buckling of structures is in reality a dynamic process. It can physically be defined as the collapse of structures under loads that are less than those causing material failures and usually results in a sudden catastrophic collapse of the structure. As a result, it may be more realistic to approach buckling and loss of stability from a dynamic point of view.
Appendix-A

ANSYS Input Codes

1. Geometry Input File

FINISH
/CLEAR, START
!****************Geometry Parameters***!

A=72.20932139  !x-dimension of ellipsoid
B=96.0  !y-dimension of ellipsoid
C=43.95349998  !z-dimension of ellipsoid
H=5.0  !height of ellipsoidal segment
KZ=24  !Number of shallow arches spanned perpendicular to z-direction decreased by one (must be even)
KX=40   !Number of shallow arches spanned perpendicular to x-direction decreased by one (must be even)
DX=23   !x-distance (in numbers of shallow arches) of starting diagonal cable
E=1.125   ! Distance between arches

/PREP7
!* Creation of shell surface by an ellipsoid***!
Sph4,,1
Vlscale,,a,b,c,,1
Block,-a-1,a+1,-b-1,b-h,-c-1,c+1,
Vovlap,,1,2
!* Erasure of no longer used volumes, areas, lines and key points***!
Numcmp,Area
Agen,2,7
Vdele,all
Adele,1,10
Numcmp,Area
Asel,,s,,1
Lsla,all
Cm,lines,line
Alls
Lsel,all
Lsel,all,lines
Ldele,all
Cmdele,lines
Alls
Lsel,all
Ksl,all
Cm,keypoints,kp
Alls
Ksel,all
Ksel,all,keypoints
Kdele,all
Cmdele,keypoints
Aplot
Numcmp,line

!* Generation of shallow archs in x-direction (longer archs)***!
Rectng,-a,a,b-h-1,b+1
/Replot
Agen,Kz/2+1,2,,E
Adele,2,1
Numcmp,Area
*Get,Number,Area,,Num,Max
Asba,1,Number,,Delete,Delete
Numcmp,Area
Asba,Number,1,,Delete,Delete
Numcmp,Area
Asba,Number,1,,Delete,Delete
Numcmp,Area
N=Number-4
/Replot
*Do,i,1,N
Numcmp,Area
Asba,Number-1,1,,Delete,Delete
Numcmp,Area
*Enddo

!* Generation of shallow archs in z-direction (shorter archs)***!
Local,11,0,E*Kx/2,,90
Wpcsys,-1
Rectng,-c,c,b-h-1,b+1
Agen,(Kx/2)+1,Number+1,,E
*Get,Number2,Area,,Num,Max
N=Number2-Number
*Do,i,1,N
Asba,All,Number+i,,Delete,Delete
*Enddo
Numcmp,Area
Adele,22,,1
Alls
/Replot
!**Creation of components for shallow arches in x- and z-direction***!
csys,0
lsel,s,loc,y,b-h,b-h+0.01
cm,basis,line
alls
lsel,all
lsel,u,,,basis
*DO,i,-kz/2,kz/2
lsel,u,loc,z,e*i
*ENDDO
cm,lattice1,line
alls
lsel,all
lsel,u,,,basis
lsel,u,, lattice1
cm,lattice2,line
cmdele,basis
alls

!***Generation of diagonal cables***!

!Cables in first direction!
*GET,NUMBER2,AREA,,NUM,MAX
local,12,0,e*dx,,,,45
wpcsys,-1
rectng,-c*2,c*2,b-h-1,b+1
csys,0
agen,dx+1,22,,,-e

*GET,NUMBER3,AREA,,NUM,MAX
n=NUMBER3-NUMBER2
asba,all,22,,keep,delete
*ENDDO
adele,241,,,1
numcmp,area

!Creation of component for diagonals in first direction!
lsel,s,loc,y,b-h,b-h+0.01
cm,basis,line
alls
lsel,all
lsel,u,,,basis
lsel,u,, lattice1
lsel,u,, lattice2
lsel,u,,cable1_1
lsel,u,, 1,2
cm,cable2_2,line
cmdele,basis
alls

!Erasure of no longer used areas!
alls
asel,all
asel,u,,1,NUMBER2
adele,all,,
alls
aplot

!Creation of component for diagonals in second direction!
lsel,s,loc,y,b-h,b-h+0.01
cm,basis,line
alls
lsel,all
lsel,u,,,basis
lsel,u,, lattice1
lsel,u,, lattice2
lsel,u,, cable1_1
lsel,u,, 1,2
cm,cable2_2,line
cmdele,basis
alls
numcmp,area

!Erasure of no longer used areas!
alls
asel,all
asel,u,,1,NUMBER2
adele,all,,
alls
aplot

!Creation of component for shell areas!

!Creation of component for diagonals in second direction!
lsel,s,loc,y,b-h,b-h+0.01
cm,basis,line
alls
lsel,all
lsel,u,,,basis
lsel,u,, lattice1
lsel,u,, lattice2
lsel,u,, cable1_1
lsel,u,, 1,2
cm,cable2_2,line
cmdele,basis
alls
numcmp,area

!Erasure of no longer used areas!
alls
asel,all
asel,u,,1,NUMBER2
adele,all,,
alls
aplot

!Creation of component for shell areas!

!Creation of component for diagonals in first direction within help geometry!
alls
numcmp,area
*GET,NUMBER4,AREA,,NUM,MAX
local,14,0,e*dx,,b,,,45
wpcsys,-1
rectng,-c*2,c*2,b-h-1,b+1
csys,0
agen,dx+1,NUMBER4+1,,,-e
alls
*DO,i,1,n
asba,all,NUMBER4+i,,delete,delete
*ENDDO
adele,460,,,1
numcmp,area

!Generation of identical help geometry!
agen,2,shell1,,,b

!Creation of component for basis lines of shell areas!

lsel,all,shell1
lsel,r,loc,y,b-h,b-h+0.01
cm,shell1,line

!Erasure of no longer used lines in original geometry!
lsel,s,loc,y,b-h,b-h+0.01
lsel,u,,basis_shell1
ldele,all

!Copying of new splitted basis lines into the original geometry!

lsel,s,loc,y,-h,-h+0.01
lgen,2,all,,,,b

!Creation of component for basis lines belonging to basis beam!

lsel,s,loc,y,b-h,b-h+0.01
lsel,u,,basis_shell1
cm,basis_beam1,line

!Erasure of no longer used lines in original geometry!
lsel,s,loc,y,b-h,b-h+0.01
lsel,u,,basis_shell1
ldele,all

!Copying of new splitted basis lines into the original geometry!
lsel,s,loc,y,-h,-h+0.01
lgen,2,all,,,,b

!Creation of component for basis lines belonging to basis beam!

lsel,s,loc,y,b-h,b-h+0.01
lsel,u,,basis_shell1
cm,basis_beam1,line
all

!**********Erasure of help geometry!
asell,all
asell,u,,shell1
adell,all
alls
lsell,all
lsell,u,, lattice1
lsell,u,, lattice2
lsell,u,, cable1_1
lsell,u,, cable2_2
lsell,u,, basis_shell1
lsell,u,, basis_shell2
ldeell,all
alls
plot

!**********Creation of Full geometry**************
csys,0
lsell,loc,y,b-h,b-h+0.01
cm,basis,line
alls
kplot
KSYMM,X,all,,1,0
KSYMM,Z,all,,1,0
lsell,all
lsell,u,,basis
lsell,u,, lattice1
lsell,u,, lattice2
lsell,u,, cable1_1
lsell,u,, cable2_2
LSYMM,X,all,,1,0
LSYMM,Z,all,,1,0
CM,arches1,line
alls
lsell,all
lsell,u,,basis
lsell,u,, lattice1
lsell,u,, lattice2
lsell,u,, cable1_1
lsell,u,, cable2_2
lsell,u,, arches1
LSYMM,X,all,,1,0
LSYMM,Z,all,,1,0
CM,arches2,line
alls
lsell,all
lsell,u,,basis
lsell,u,, lattice1
lsell,u,, lattice2
lsell,u,, cable1_1
lsell,u,, cable2_2
lsell,u,, arches1
lsell,u,, arches2
LSYMM,X,all,,1,0
LSYMM,Z,all,,1,0
CM,arches1,line
alls
lsell,all
lsell,u,,basis
lsell,u,, lattice1
lsell,u,, lattice2
lsell,u,, cable1_1
lsell,u,, cable2_2
lsell,u,, arches1
lsell,u,, arches2
LSYMM,X,all,,1,0
LSYMM,Z,all,,1,0
CM,arches2,line
alls
lsell,all
lsell,u,,basis
lsell,u,, lattice1
lsell,u,, lattice2
lsell,u,, cable1_1
lsell,u,, cable2_2
cm,a, line
lsell,,cable2_2
LSYMM,X,all,,1,0
LSYMM,Z,all,,1,0
cm,b, line
lsell,,a
LSYMM,Z,all,,1,0
lsell,,a
cm, line
alls
lsell,all
lsell,u,,basis
lsell,u,, lattice1
lsell,u,, lattice2
lsell,u,, cable1_2
lsell,u,, cable1_1
lsell,u,, arches1
lsell,u,, arches2
lsell,u,, arches1
lsell,,a

CMSEL,S,BASIS_BEAM1
LSYMM,X,all,,1,0
LSYMM,Z,all,,1,0
Cm,basis_beam,line
alls
CMSEL,S,BASIS_SHELL1
LSYMM,X,all,,1,0
LSYMM,Z,all,,1,0
Cm,basis_shell,line
Cmdele,lattice1
Cmdele,lattice2
Cmdele,cable1_1
Cmdele,cable2_2
Cmdele,basis_beam1
Cmdele,basis_shell1
Cmdele,basis
Cmdele,shell1

ARSYM,X,all,,1,0
ARSYM,Z,all,,1,0
Aset,all
Cm,shell,area
alls
Nummg,kp
Cm,arches1,line
Cm,lattice1,line
Cm,arches2,line
Cm,lattice2,line
Cmdele,arches1
Cmdele,arches2
alls
aplo
FINISH
2. Discretization Input File:

/PREP7

!joint=2          !joint=0 ... bending resistant jointed
!joint=1 ... pin-jointed
!joint=2 ... joints with decreased stiffness
shell=1          !shell=0 ... without shell elements

!shell=1 ... with shell elements

!***Material Parameters***!
!***Units [kN],[m],[K]***!

!******Basis Beam!*****!
ET,1,BEAM4
KEYOPT,1,9,1
MP,EX,1,2.1e8
MP,NUXY,1,0.3
MP,ALPX,1,1.2e-5
MP,DENS,1,7.85
R,1,8.8e-3,117.3e-8,3549.3e-8,4e-2,22e-2 !Area,Izz,Iyy,Tz,Ty

!******Shallow archs - Beam elements*****!
ET,2,BEAM4
KEYOPT,2,9,1
MP,EX,2,2.1e8
MP,NUXY,2,0.3
MP,ALPX,2,1.2e-5
MP,DENS,2,7.85
R,2,2.4e-3,72e-8,32e-8,6e-2,4e-2  !Area,Izz,Iyy,Tz,Ty

!******Shallow arches - Link elements*****!
ET,5,LINK8
MP,EX,5,2.1e8
MP,ALPX,5,1.2e-5
MP,DENS,5,7.85
R,5,2.4e-3                   !Area

!Shallow archs - Beam elements with decreased stiffness!
R,6,0.9e-3,2.7e-8,24.2e-8,6e-2,1.5e-2  !Area,Izz,Iyy,Tz,Ty

!******Prestressed Cables!
ET,3,LINK10,...,0          !Keyopt(3)=0 ...
Tension only
MP,EX,3,1.6e8
MP,ALPX,3,1.2e-5
MP,DENS,3,7.85
R,3,4.48e-6,6.5e-3                   !Area,InitialStrain

!****Shell elements!
ET,4,SHELL43
MP,EX,4,7e5
!MP,EX,4,7e7
MP,NUXY,4,0.23
MP,ALPX,4,0.9e-5
MP,DENS,4,2.5
R,4,1.8e-2                   !Thickness

!****Discretisation****!
!******Basis Beam! alls!
alls
lsel,s,,basis_beam
lesize,all,,,1
mat,1
real,1
lmesh,all
esll,s
cm,beam_el,elem
alls

!Shallow archs!

!*if,joint,eq,0,then
lsel,s,,lattice1
lsel,a,,lattice2
lesize,all,,,1
type,2
mat,2
real,2
lmesh,all
lsel,s,,lattice1
lsel,a,,lattice2
esll,s
cm,lattice_el,elem
alls
!endif

!*if,joint,eq,1,then
lsel,s,,lattice1
lsel,a,,lattice2
ksel,all
ksel,r,loc,y,b-h,b-h+0.01
lsk,s,,all
lsel,u,,basis_beam
lsel,u,,basis_shell
cm,beam_line,line
lesize,all,,,1
type,2  !bending resistant jointed basis beam joints
mat,2
real,2
lmesh,all
lsel,s,,lattice1
lsel,a,,lattice2
lsel,u,,beam_line
lesize,all,,,1
type,2  !pin jointed truss joints
mat,2
real,
lmesh,all
cmdele,beam_line
lsel,s,,lattice1
lsel,a,,lattice2
esll,s
cm,lattice_el,elem
alls
!endif

!*if,joint,eq,2,then
lsel,s,,lattice1
lsel,a,,lattice2
ksel,all
ksel,r,loc,y,b-h,b-h+0.01
lsk,s,,all
lsel,u,,basis_beam
lsel,u,,basis_shell
cm,beam_line,line
lesize,all,,,1
type,2  !bending resistant jointed basis beam joints
mat,2
real,2
lmesh,all
esll,s
cm,new_lines1
lsel,s,,new_lines1
lsel,a,,joint_lines1
**3. Boundary Conditions Input File**

```
/PREP7

**Boundary Conditions**!

Sup=0

Sup=0 ... vertical supported
Sup=1 ... perpendicular to shell surface supported
Sup=2 ... perpendicular and in shell surface dir. supported

*if,sup,eq,0,then
  alls
  esel,s,,lattice_el
  nsle,all
  nsel,r,loc,y,b-h-0.01,b-h+0.01
  d,all,uy
  d,all,rotx
  d,all,rotz
  !smallest and highest x- and y-coordinate!
  nsel,r,loc,x,-0.01,0.01
  d,all,ux
  esel,s,,lattice_el
  nsle,all
  nsel,r,loc,y,b-h,b-h+0.01
  nselr,loc,z,-0.01,0.01
  d,all,uz
*endif

*if,sup,eq,1,or,sup,eq,2,then
  alls
  esel,s,,lattice_el
  nsle,all
  nsel,r,loc,y,b-h-0.01,b-h+0.01
  *GET,NUMBER,NODE,,COUNT
  *DO,ii,1,NUMBER
    *GET,Nr,NODE,,NUM,MIN
    *GET,xCor,NODE,Nr,LOC,X
    *GET,yCor,NODE,Nr,LOC,Y
    *GET,zCor,NODE,Nr,LOC,Z
    *if,xCor,EQ,0,then
      n,100000,xCor+1,yCor,zCor
    *endif
    *if,yCor,EQ,0,then
      n,100000,xCor,yCor+1,zCor+1
    *endif
    *if,zCor,GT,0,then
      *SET,z1,-14*xCor/(23*23)/SQRT(1-xCor*xCor/(23*23))
      n,100000,xCor+1,yCor,zCor-z1
    *else
      *SET,z1,-14*xCor/(23*23)/SQRT(1-xCor*xCor/(23*23))
      n,100000,xCor+1,yCor,zCor+z1
    *endif
    *if,zCor,LT,0,then
      *SET,z1,-14*xCor/(23*23)/SQRT(1-xCor*xCor/(23*23))
      n,100000,xCor+1,yCor,zCor+z1
    *else
      *SET,z1,-14*xCor/(23*23)/SQRT(1-xCor*xCor/(23*23))
      n,100000,xCor+1,yCor,zCor-z1
    *endif
  *endif
  *if,xCor,GT,0,then
    CLOCAL,11,0,,,,,,180
  *endif
*endif

**only for consideration of stiff shell elements**!

**Shell elements!**

*if,shell,eq,1,then
  alls
  asel,s,,shell
  type,4
  mat,4
  real,4
  esize,1
  amesh,all
  esla,s
  cmshell_el,elem
  alls
*endif

**only for consideration of stiff shell elements**!

**NUMMRG,NODE,0.0001, , , LOW**

numcmp,node
alls
eplot
FINISH
```
3. Loading Input File:

/PREP7

***Loading Conditions***!

dl=1   !lc=0 ... dead load off
f_dl=1.35 !factor for dead load
wi=0     !wi=1 ... wind load
f_wi=1.5 !factor for wind
sn=0     !sn=1 ... snow load
f_sn=1.5 !factor for snow
sl=0     !sl=1 ... single load
f_sl=1.5 !factor for single load

********dead load***********!
*if,dl.eq,1,then
  esel,s,,lattice_el
  nsle,all
  nsel,r,loc,y,b-h,b+h=0.01
  cm,beam_node,node
  esel,s,,lattice_el
  nsle,all
  nsel,u,,beam_node
  cm,lattice_node,node
  F,all,FY,-0.99*f_dl
  cmdele,beam_node
  cmdele,lattice_node
  alls
*endif

*******wind load************!
*if,wi.eq,1,then
  asel,s,loc,z,0,15
  esla,s
  SFE,all,0,PRES,-0.195*f_wi,,
  asel,s,loc,z,-15,0
  esla,s
  SFE,all,0,PRES,0.455*f_wi,,
*endif

*******snow*************!
*if,sn.eq,1,then
  esel,s,,lattice_el
  nsle,all
  nsel,r,loc,y,b-h,b+h=0.01
  cm,beam_node,node
  esel,s,,lattice_el
  nsle,all
  nsel,u,,beam_node
  cm,lattice_node,node
  F,all,FY,-0.94*f_sn
  cmdele,beam_node
  cmdele,lattice_node
  alls
*endif

*******single load*********
*if,sl.eq,1,then
  esel,s,,lattice_el
  nsle,all
  nsel,r,loc,x,-0.1,0.1
  nsel,r,loc,z,9.52,10.63
  F,all,FY,-0.5*f_sl
  alls
*endif

4. Solution Input File

/SOLU

sol=1       !sol=0 ... linear solution
           !sol=1 ... geometrically nonlinear solution
           !sol=2 ... classical stability analysis

***********Linear Solution************!
*if,sol.eq,0,then
  ANTYPE,0  !Static analysis
  OUTRES,ALL,ALL  !Save results
  solve
*endif

***Geometrically Nonlinear Solution***!
*if,sol.eq,1,then
  ANTYPE,0  !Static analysis
  NROPT,FULL,,OFF
  !Full Newton-Raphson method without !adaptive descent
  NLGEOM,ON  !Large deformation considered
  SSTIF,ON  !Stress stiffening considered
  DELTIM,0.2,1e-3,0.2 !Definition of time steps
  OUTPR,ALL,ALL  !Write data to output file
  OUTRES,ALL,ALL  !Save results of all sub steps
  solve
*endif

*****Classical Stability Analysis*****!
*if,sol.eq,2,then
  ANTYPE,0  !Static analysis
  NROPT,FULL,,OFF
  !Full Newton-Raphson method without !adaptive descent
  PSTRES,ON  !Prestress effects included
  DELTIM,0.2,1e-3,0.2 !Definition of time steps
OUTRES,ALL,ALL                  !Save results of all sub steps 
solve 
FINISH 

/SOLU
ANTYPE,BUCKLE,NEW 
BUCOPT,SUBSP,10
OUTPRES,NSOL,ALL 
solve 
FINISH 

/SOLU
EXPASS,ON
MXPAND
OUTPR,ALL,ALL
OUTRES,ALL,ALL 
solve 
FINISH 
/POST1
SET,,1
PLDISP,0
*endif 
FINISH 

5. Post processing Input File

/POST1 
/DSCALE,1,10    
/CONT,1,10,AUTO
PLNSOL,U,SUM,0,1

!***Definition of element table and print out of results****!
!Items can be found in element description in the ANSYS help!

ETABLE,STRESS,LS,1
!axial stress of cable_el, lattice_el, beam_el
PLETAB,STRESS,NOAV
PRETAB,STRESS
/CVAL,1,-20000,0,20000,30000,50000,750000,850000
!definition of no uniform contour plot
/REPLOT
ETABLE,STRAIN,LEPEL,1
!axial strain of cable_el, lattice_el, beam_el
PLETAB,STRAIN,NOAV
/CONT,1,10,AUTO
/REPLOT

45
References

[1] ANSYS Help version 7.0