BUCKLING ANALYSIS OF LARGE DIAMETER CONCRETE SPHERICAL SHELL DOMES

Ahmed Hadi Abood
Pumps Engineering Dept. – Technical College – Al-Musaib

ABSTRACT

Shell domes are largely applied nowadays for their aesthetic shape, architectural aspect, covering large spans, high strength capacity to apply loads and low costs, like domes of mosques, industrial building, auditoriums, nuclear reactors, space vehicles , … etc. Modern concrete shell domes can be built to the ratio (thickness -to- radius) of 1:800 constructed with concrete and wire mesh and they are safe and beautiful.

The main aim of this study is to investigate the behavior and strength of modern thin spherical domes made of concrete with and without ribs through buckling modes.

The work includes spherical domes with large diameter 50 m with rib and without, with thickness 5 cm.

All models were analyzed, using modal structural analysis and design program ANSYS code 12 is used to consider the influence of the large diameter on mode shape and to determine the critical load.

KEYWORDS: Dome, Finite Element Analysis, Buckling , concrete, ANSYS.

الخلاصة

تَطبق القبب القشرية بشكل كبير في الوقت الحاضر لشكلِها الجمالي وذات سمة معمارية، وتَغطّي مجالات كبيرةَ، ولها القدرة العاليةِ لتَطبيق الأحمالِ الكبيرة وتكون ذات كلفِ منخفضةِ، مثل قببِ المساجدِ، البنايات الصناعية، الصالات، المفاعلات النووية، عربات فضاءِ ، … الخ. القبب القشرية الخرسانيةِ الحديثةِ يُمْكِنُ أَنْ تُبْنَى بنسبةِ (سُمك إلى نصفِ القطر) 1:800 وتتضمن شبكةِ الاسلاكَ والخرسانةَ وتكون آمنة وجميلة.

إنّ الهدفَ الرئيسيَ لهذه الدراسةِ أَنْ تَتحرّى السلوكَ ومقاومة القببِ الكرويةِ الرقيقةِ الحديثةِ صَنعا من الخرسانةِ مَع وبدون أضلاعِ من خلال معرفة اطوار الانبعاج لهذه القبب.

يَتضمّنُ العملُ القببِ الكرويةِ بالقطرِ الكبيرِ (50) متر ، بدون اضلاعِ ومع وجود الاضلاع ، بسُمكِ (5) سم كُلّ النماذج تم تحليلها باستخدام برنامج التصاميم والتحليل الهندسي ANSYS 12 حيث استخدم لدراسة تاثير القطرِ الكبيرِ على شكلِ النمطِ ولحساب الحملِ الحرجِ.

INTRODUCTION

In general, Dome structures are increasingly used nowadays to roof large spans and to provide column –free space in industrial buildings, auditoriums, domes, space vehicle, nuclear reactor and pressure vessels … etc[1,2,3] .

From the point of view of architecture, the development of shell structure offers unexpected possibilities and opportunities for the combined realization of functional, economic and aesthetic aspects[1]. Chen, et al (1979)[4],studied and tested conical concrete-shell specimens with widely varying material properties and traced their load-deformation response, internal stresses and crack propagation through the elastic, inelastic, and ultimate stress ranges. It is possible to vary the stress strain properties of concrete and the behaviour of conical concrete shells by: (1). Acrylic impregnation (2) steel reinforcement as ring stiffening; and (3) general mesh reinforcement.[3]

The stress-strain and fracture behaviour for both reinforced and unreinforced concrete is tailored to range from strong linear elastic but brittle to tough and ductile by various combinations of rubbery methyle methacrylate (MMA)[5]. Such composite specimens are found ideal for the purpose of comparison with various material models now available in the extended NONSAP program .

Zweilfel, (1997)[6] presented a small diameter concrete dome with alternate forming system. He built a dome of 16 ft in diameter and 8 ft in height. It covers 200 square feet. Manasrah, (1998)[3] investigated the effectiveness of connection elements of the segmental cylindrical shell units subjected to knife-edge load at the crown of the shell. A total of eight full scale shell unit models with four types of connections were constructed and tested. All the models were of 450 mm width, 35 mm thickness, 1000mm head of the crown and 4000 mm covered span. Jackson, Mississippi (2002)[7], studied structural evaluation of the 5 meter diameter observatory dome structure constructed by Observa-Dome laboratories, Inc. Hani (2010)[8] studied the impact of diameter, number of ribs, percentage of steel, compressive strength and cover thickness on the large concrete dome.

Regarding the presentation of literature review, it should be emphasized that no investigation related to the buckling of large concrete domes is found. In this studied the critical load and mode shape of the domes are presented.

ISLER'S BUCKLING EQUATION

Isler, (1996)[2] , a Swiss engineer who in the course of thirty-five years has built more than 1500 thin-shell concrete domes, has synthesized his experience into an equation:

$$P_k = C\,\alpha.\beta.\delta\ .\ E\ .\ (t/r)^{x} \geq 5.\ P_{eff} \qquad (1)$$

It was stated that the critical buckling load (**P_k**) is equal to constant (C) times the modulus of elasticity [E] of the concrete times the (t/r) ratio , raised to a power, of the thickness of the shell (r) which defines the curvature of the shell . The ratio will always be a very small number. The constant (C) is the product of three numbers: α unknown at first, is the result of the computation and must be solved between (0.2) and (1.2); β is a long-studied and well-known margin-of-error factor due to inaccuracies in the shape, about (0.3); and γ is a number, learned from experience with models or previous structures, that is specific to each shape, of shell and, in case of sphere is equal to (1). Most powerful is (x), the exponent, which also dependent on the specific shape and has been learned and catalogued by Isler from experience; its value ranges from (3) for a cylinder to (2) for a sphere. For a stable shell, the critical buckling load must be greater than or equal to an effective load P_{eff} (e.g. snow), times a safety factor, in Isler's case a factor of (3).

CONCRETE MODELS ADOPTED IN THIS STUDY:

In this study, concrete material models that deal with the nonlinear three dimensional analysis of reinforced concrete members under static increasing load are considered. These models treat the concrete as being a linear elastic-perfectly plastic-brittle-fracture material, and therefore they constitute the following:

- Stress-strain relationship model.
- Failure criterion to simulate cracking and crushing of concrete.
- Cracking model.
- Crushing model.

These models are implemented in ANSYS program

Stress-Strain Relationship Model [8]:

In this study, the concrete is assumed to be homogeneous and initially isotropic. The stress-strain relation is described by an elastic-perfectly plastic-brittle fracture model, as shown in **Fig.1**. The concrete under a triaxial stress state is assumed to crush or crack completely once the fracture surface is reached. The complete stress-strain relationship for a perfectly plastic-brittle fracture model is developed in three parts:

(1)Before yielding, (2) During plastic flow, (3) After fracture.

This stress-strain relationship is expressed by a single value of Young's modulus, E, and a constant Poisson's ratio, υ. So this relation can be written in matrix form as:

$$\{\sigma\} = [D_c]\ \{\varepsilon\} \tag{2}$$

where:

$\{\sigma\}$= stress vector

$[D_c]$= constitutive matrix for concrete

$\{\varepsilon\}$= strain vector

The matrix $[D_c]$ for uncracked elastic concrete can be defined by[8,9]:

$$[D_c] = \frac{E}{(1+\upsilon)(1-2\upsilon)} \begin{bmatrix} (1-\upsilon) & \upsilon & \upsilon & 0 & 0 & 0 \\ \upsilon & (1-\upsilon) & \upsilon & 0 & 0 & 0 \\ \upsilon & \upsilon & (1-\upsilon) & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{(1-2\upsilon)}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{(1-2\upsilon)}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{(1-2\upsilon)}{2} \end{bmatrix} \tag{3}$$

The equivalent uniaxial stress-strains in the various stages are given by:

1) For $\sigma \le f'_c$ then $\sigma = E\varepsilon$

2) For $\varepsilon \ge \frac{f'_c}{E}$ then $\sigma = f'_c$

The incremental stress-strain relationship can be expressed as:

$$\{d\sigma\} = [D_c]\ \{d\varepsilon\} \tag{4}$$

where:

$\{d\sigma\}$= stress increment vector

$\{d\varepsilon\}$= strain increment vector

Determination of the Model Parameters

A total of five strength parameters are needed to define the failure surface as well as an ambient hydrostatic stress state (f'_c, f_t, f_{cb}, f_1, f_2 and σ_h^a), these are shown in **Fig.2**.

f'_c and f_t can be specified from two simple tests. The other three constants can be determined from [8,9]:

$$f_{cb} = 1.2\, f'_c \quad (5)$$

$$f_1 = 1.45\, f'_c \quad (6)$$

$$f_2 = 1.725\, f'_c \quad (7)$$

However, these values are valid only for stress states where the following condition is satisfied:

$$|\sigma_h| \le \sqrt{3}\, f'_c \quad (8)$$

where:

$$\sigma_h \text{ = hydrostatic stress state = } \frac{1}{3}\left(\sigma_{xp} + \sigma_{yp} + \sigma_{zp}\right) \quad (9)$$

Condition(8) applies to stress situations with a low hydrostatic stress component[8,9].

In **Fig.2**, the lower curve represents all stress states such that ($\theta = 0°$), while the upper curve represents stress states for ($\theta = 60°$). Here θ is defined as the angle of symmetry. The axis (ξ) represents the hydrostatic length.

The materials properties of our research is taken as in **Table 1**.

Table 1: Material properties used Reinforcing concerted ribbed domes[8]

Concrete		
E_c	Young's modulus (MPa)	25800
$f'c$	Compressive strength (MPa)	30.36
f_t	Tensile strength (MPa)	3.1
υ	Poisson's ratio	0.2
Reinforcing steel		
E_s	Young's modulus (MPa)	200000
f_y	Yield stress (MPa)	344.8
υ	Poisson's ratio	0.3

FINITE ELEMENT MODEL OF CONCRETE

In this study, three dimensional 8-node solid elements are used to model the concrete. The element has eight corner nodes, and each node has three degrees of freedom "u, v and w" in the "x, y and z "directions respectively, as shown in **Fig.3**.(Solid element 65 in ANSYS)

Reinforcement Idealization [8]:

In developing a finite element model for reinforced concrete members three alternative representations of reinforcement can usually be used, these are given as follows:

1. Distributed Representation:

   In this approach, the reinforcement is assumed to be distributed in a layer over the element in any specified direction. To derive the constitutive relation of a composite concrete-reinforcement, perfect bond is assumed, **Fig.4.a**.

2. Discrete Representation:

   One dimensional bar element may be used in this approach to simulate the reinforcement. Discrete representation has been widely used due to its versatility and capability to adequately account for the bond-slip and dowel action phenomena, **Fig.4.b**.

3. Embedded Representation:

The embedded representation is often used with high order isoparametric elements. The bar elements are assumed to be built into the brick elements. In this approach perfect bond is assumed between the reinforcing bars and the surrounding concrete. The stiffness of steel bars is added to that of the concrete to obtain the global stiffness matrix of the element. It is assumed that the bars are restricted to be parallel to the local coordinate axes ξ, η and ζ of the brick element, **Fig.4.c**.

In this work, the reinforcement is included within the properties of the 8-node brick elements (embedded representation) to include the reinforcement effect in the concrete structures, excluding the reinforcing bars that are crossing the joint, which are represented by using "bar elements" (Discrete representation). In the two manners, the reinforcement is assumed to be capable of transmitting axial forces only, and perfect bond is assumed to exist between the concrete and the reinforcing bars.

Bar Element (Link8) [10]

The element has three degrees of freedom at each node, (nodal translation in x, y and z-directions). The elements used to represent shear connectors are also used in resisting uplift separation. The axial normal stress is assumed to be uniform over the entire element. The "bar elements", have been used in this study.

Beam Element (beam188)[10]

BEAM188 is suitable for analyzing slender to moderately stubby/thick beam structures. This element is based on Timoshenko beam theory. Shear deformation effects are included.
BEAM188 is a linear (2-node) or a quadratic beam element in 3-D. BEAM188 has six or seven degrees of freedom at each node, with the number of degrees of freedom depending on the value of KEYOPT(1). When KEYOPT(1) = 0 (the default), six degrees of freedom occur at each node. These include translations in the x, y, and z directions and rotations about the x, y, and z directions. When KEYOPT(1) = 1, a seventh degree of freedom (warping magnitude) is also considered. This element is well-suited for linear, large rotation, and/or large strain nonlinear applications.

BEAM188 can be used with any beam cross-section defined via SECTYPE. The cross-section associated with the beam may be linearly tapered.

Section Type of Beam188

The command is SECTYPE associates with element beam188, it has several sections (**Fig.5**) , section type T-shaped is used in our research .

Nonlinear Solution Techniques [10,11]:

The nonlinear response of most reinforced concrete structures can be conceived from the examination of the load-deflection history, and it will appear that all response exhibits the fundamental characteristics of nonlinear structural behavior by "changing structural stiffness". The nonlinear structural behavior arises from a number of causes, which can be grouped into three principal categories:

- Changing status (Including contact).
- Geometric nonlinearities.
- Material nonlinearities.

In this study, both changing status and material nonlinearities are considered. The use of finite element discretization in nonlinear structural problems results in a set of nonlinear algebraic equations of the form given earlier in Equation(10):

$$[K]\ \{a\} = \{F^a\} \tag{10}$$

where:

$[K]$= stiffness matrix (with element depending on current state of strains and stresses)

$\{a\}$= vector of nodal displacements

$\{F^a\}$= vector of applied loads (external load vector)

The solution for the equation of linear elastic structural problem can be obtained directly. But in nonlinear problems, a direct solution is no longer possible since the stiffness matrix [K] depends on the unknown displacement level. Therefore, it cannot be exactly calculated before the determination of the unknown nodal displacements {a}. For the solution of a nonlinear structural problem, the state of equilibrium of the structural system corresponding to the applied load must be found, and the equilibrium equations can be expressed as:

$$\{R\} = \{F^{in}\} - \{F^a\} \tag{11}$$

where:

$\{R\}$= out of balance force vector

$\{F^{in}\}$ = internal nodal forces vector

where $$\{F^{in}\} = \int_v [B]^T \{\sigma\} \cdot dv \tag{12}$$

The internal forces also depend on the unknown displacement level, and they have to be approximated in successive steps until Equation(11) is satisfied with small out of balance. Several techniques exist for solving the nonlinear Equation(12) and they can be classified into:

Incremental Techniques.

Iterative Techniques.

Incremental-Iterative Techniques.

In this study, a maximum number of iterations for last increment of load is specified to stop the nonlinear solution if the convergence tolerance has not been achieved. A maximum number of iterations of range (150-250) have been used, because it is observed and found that this range is generally sufficient to predict the solution's divergence or failure.

MODEL GENERATION

The ultimate purpose of a finite element analysis is to re-create mathematically the behavior of an actual engineering system. In other words, the analysis must be an accurate mathematical model of a physical prototype.

In the broadest sense, the model comprises all the nodes, elements, material properties, real constants, boundary conditions and the other features that used to represent the physical system. In ANSYS terminology, the term model generation usually takes on the narrower meaning of generating the nodes and elements that represent the special volume and connectivity of the actual system. Thus, model generation in this discussion will mean the process of defining the geometric configuration of the model's nodes and elements. The program offers the following approaches to model generation [10,11] :

a) Creating a solid model
b) Using direct generation
c) Importing a model created in a computer-aided design CAD system.

Two different methods used to generate a model: Solid model and direct generation. In solid modeling some one can describe the boundaries of the model, establish controls over the size and desired shape elements automatically, by contrast. In the direct generation method, determine the location of every node and size, shape and connectivity of every element prior to defining these entities in ANSYS model.

Solid modeling is usually more powerful and versatile than direct generation. In this study the solid modeling and direct method are used together to constructed the APDL program.

ANSYS Modeling

In this study the dome with large diameter 50 m is analysis with rib and without, firstly the dome is plotted and meshed using solid65 with different thickness 5 cm as shown in **Fig.6**. Then the dome is reinforcement using steel element link8 as shown **Fig.7**.

The study is concerned on the effect of the ribs on the deflection of the dome. The section of the rib is T.

Two cases are studied, the first one is the dome with one rib, the second is the dome with two ribs. in case of the one rib ,the rib is located half on the dome and in the second case the two rib was crossed as shown in **Fig.8**.

Several difficulties was attacked in the building the model, there are two major cases are studied, the first one is the rib embedded in the dome and in this case the rib is done by using direct generation using beam188 with square-section while in the second case the square- section as an area and then dragging with path to form the rib, this steps is done with one and two ribs. To connect the rib and the dome, the dowel is used which represented by link8.

Buckling Analysis By ANSYS Finite Element Techniques [10]

Two techniques are available in the ANSYS/Mechanical, programs for predicting the buckling load of a structure: nonlinear buckling analysis and eigenvalue (or linear) buckling analysis. Since these two methods frequently yield quite different results.

Eigenvalue buckling analysis predicts the theoretical buckling strength (the bifurcation point) of an ideal linear elastic structure. This method corresponds to the approach of 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.

Linear buckling analysis in ANSYS finite elements software is performed in two steps. In the first step a static solution to the structure is obtained . In this analysis the prebuckling stress of the structure is calculated. The second step involves solving the eigenvalue problem given in the form of the following equation :

$$([K]+\lambda_i[S])\{\psi_i\}=\{0\} \tag{13}$$

Where :
[K] = stiffness matrix
[S] = stress stiffness matrix
λ_i = ith eigenvalue (used to multiply the loads which generated [S])
ψ_i = ith eigenvector of displacements

The buckling analysis is defined as the analysis type (ANTYPE, Buckle) and the analysis option (Bucopt) in which the solution methods is chosen either subspace iteration method (Subspac) which is generally recommended for eigenvalue buckling because it uses the full system matrices, and the other method is the Householder method (Reduced).

After that, the no. of eigenvalue to be extracted is chosen by activating the no. of modes to expand, the command (MXPAND).

APDL program is made to build the model and analyze the concrete dome with buckling load by ANSYS12 software, as following steps:

```
/prep7
/view,1,1,1,1
/vup,1,y
! This Program is done by AHMED HADI ABOOD
! Technical College Al-Musaib
*ask,R,Raduis of the Dome,25
*ask,T,thickness of the Dome,0.05
sphere,R,R-T,0,180
et,1,solid65  ,    r,1,,,,,
mp,ex,1,34480e6  , mp,prxy,1,.15
TB,concr,1 ! Willam and Warnik
tbdata,1,0.2,0.7,2.2e6,45e6
lesize,13,,2  ,  lesize,14,,2  ,  lesize,1,,2  , lesize,2,,2
lesize,3,,2  ,  lesize,4,,2  ,  lesize,5,,2 ,  lesize,6,,2
lesize,7,,2  , lesize,8,,2  ,  lesize,9,,2 , lesize,10,,2
lesize,11,,2 ,  lesize,12,,2 , mshape,1,3d , mshkey,0
vmesh,1 ,  nsel,s,loc,y,0 , D,all,all , allsel,all
! reinfoce
et,2,link8  ,  r,2,78.5e-6 , mp,ex,2,200e9
mp,prxy,2,.3 , tb,biso,2
tbdata,1,520e6,10e6
type,2 , mat,2  , real,2
*do,I,4622,4705,1
E,I,I+1  , *enddo
*do,I,362,445,1
E,I,I+1  , *enddo
allsel,all! rib  T section
et,3,beam188 , keyopt,3,1,0
mp,ex,3,26998e6 , sectype,1,beam,T
secdata,0.6,0.6,0.1,0.3
type,3  , mat,3 , real,3
*do,I,48,90,1 , E,I,I+1 , *enddo
*do,I,3,45,1 , E,I,I+1 , *enddo
E,2,46   ,  E,2,48 , allsel
finish
! Solution
/solu
antype,static  , pstres,on
! applied load
  F,2,fy,1   ,    allsel,all
Solve
Finish
/solu
antype,buckle        !buckling analysis
```

```
bucopt,subsp,4       ! 2 no. of mode shape
mxpand,4             ! 2 no. of mode shape
solve
finish
```

RESULTS AND DISCUSSION

The behavior of the structure of dome from the beginning of loading down to its ultimate is shown in **Fig.9**.

This work covers the Finite Element analysis of reinforcement concrete rib dome. Three cases used to calculate the buckling mode shapes. The first case without ribs and with embedded ribs and the third case was for two embedded ribs. All models are concrete dome with diameter 50m and thickness (5 cm) and with one and two ribs.

The ANSYS finite element program is used to analyze the above study cases to determine the critical load at reinforcement concrete dome and its buckling mode shapes. The ANSYS results are comparison with the published theoretical results and getting achieved good agreement.

Dome Without Rib

The results of reinforced concrete dome without rib is modeled by 8-node brick elements soid65 shown in **Table 2**.

Table 2 : Result of Dome without Rib

Diameter of dome(m)	ANSYS Max. Stress MPa	Theoretical stress[12] MPa	Max, deflection Mm	critical load KN
50	1.51	1.65	1.49	184

The results of the buckling mode shapes shown in **Fig.10**.

Dome With One Embedded Rib

The results of reinforced concrete dome with one embedded rib is modeled by 8-node brick elements soid65 shown in **Table 3**.

Table 3: Result of dome with one embedded rib

Diameter of dome(m)	ANSYS Max. Stress MPa	Max, deflection mm	critical load kN
50	1.55	1.82	245

The results of the buckling mode shapes shown in **Fig.11**.

Dome With Two Embedded Rib

The results of reinforced concrete dome with two embeded rib is modeled by 8-node brick elements soid65 shown in **Table 4**.

Table 4: Result of dome with two embedded rib

Diameter of dome(m)	ANSYS Max. Stress MPa	Max, deflection mm	critical load kN
50	1.68	1.87	269

The results of the buckling mode shapes shown in **Fig.12**.

Effect of Number of Ribs

The effect of the number of ribs can be shown in **Table 5**.

Table 5: Explain the effect of number of ribs on the max. deflection and load

Rib	ANSYS Max. Stress (MPa)	Max, deflection (mm)	critical load (kN)
Without rib	1.51	1.49	184
one rib	1.55	1.82	245
Two ribs	1.68	1.87	269

REFERENCES

1. Chandrashekhra, k., Analysis of Thin Concrete Shells.(2nd ed.).new age international publishers , India (1995).
2. Chen ,W.F. "Experiments On Axially Loaded Concrete Shells " Journal of The Structural Division , Proceeding of The American Society of Civil Engineers , Vol.105, No.ST8 , August ,1979.
3. Ford, E. ,The Theory and Practice of Impermanence, Report from website. " Harvard Design magazine" , number (3) , Harvard colledge (2001).
4. Hani Aziz Ameen “The impact of diameter, number of ribs, percentage of steel, compressive strength and cover thickness on the large concrete dome”, American Journal of Scientific and Industrial Research, 2010, 1(3): 472-495
5. Jackson , Mississippi, Report " The Finding of Structural Evaluation of the 5 meter Diameter Dome"(2002).
6. Manasrah, A., " Ferrocement Segmented Shell Structures" M.S.c. Thesis University of Technology , Building and Construction Department . Baghdad – Iraq , September (1993).
7. Nanette South “ Finite Element analysis of Monolithic Dome”, M.Sc. thesis, Idaho State University, 2005.
8. Saeed Mouveni “ Finite element analysis” , theory and application
9. Tony, R., Engineering a new Architecture. (Quebecor - Eusey, Leominster, Massachusetts, USA (1996).
10. User’s manual of FEA/ANSYS/ Version 12 , 2009 .
11. W.F. Chen “ Plasticity in Reinforced Concrete” McGraw-Hill Co. ,1982.
    with ANSYS, Prentice Hall Press,1999.
12. Zweilfel, CH., "Harvard Design Magazine", Harvard Colledge (1997).

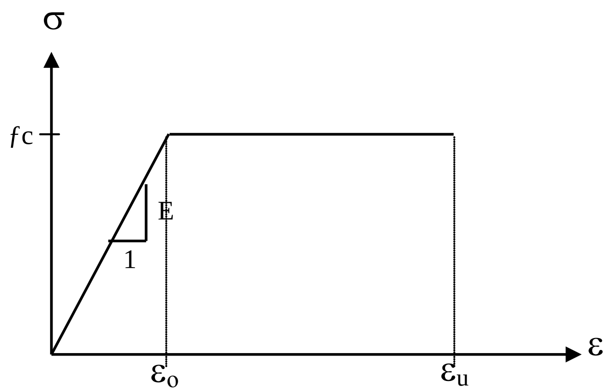


Fig. 1: Uniaxial stress-strain relationship used for concrete.

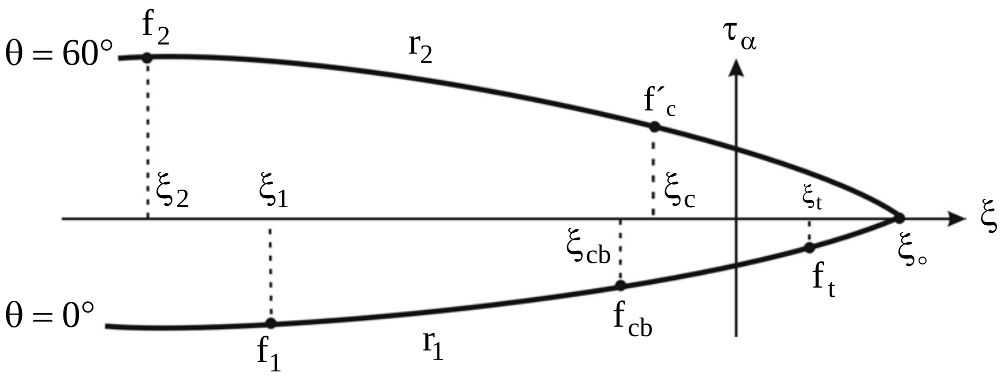


Fig. 2 : A profile of the failure surface as a function of five parameters[8,9].

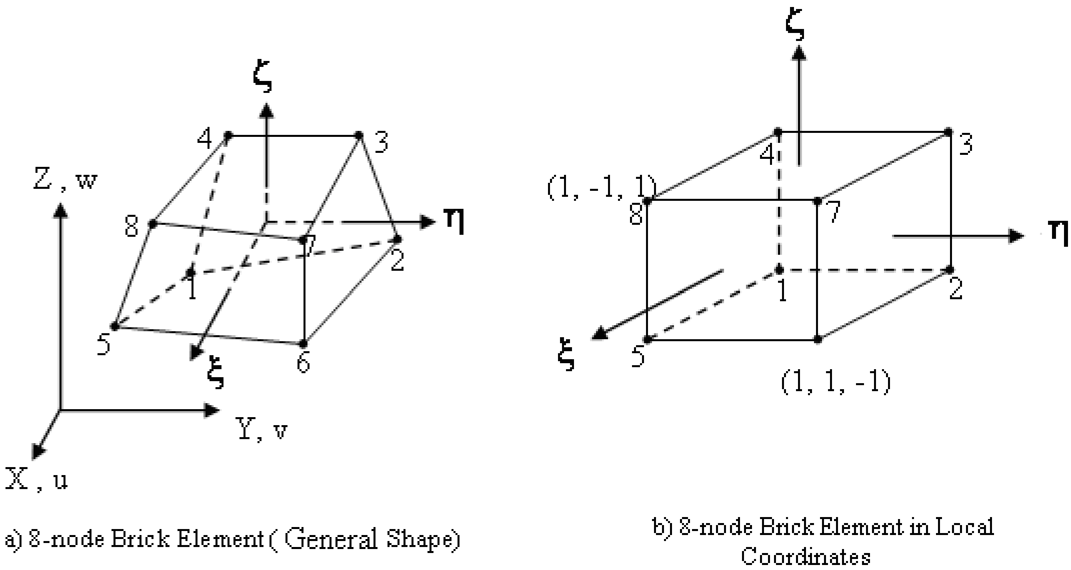
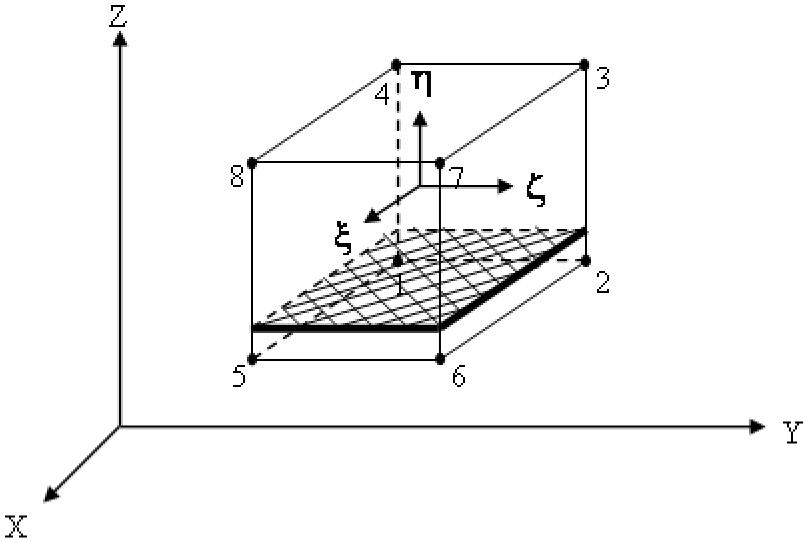
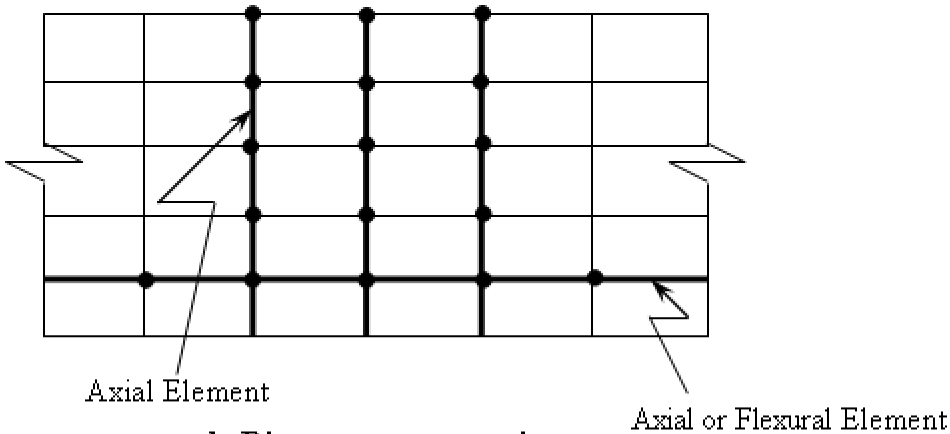


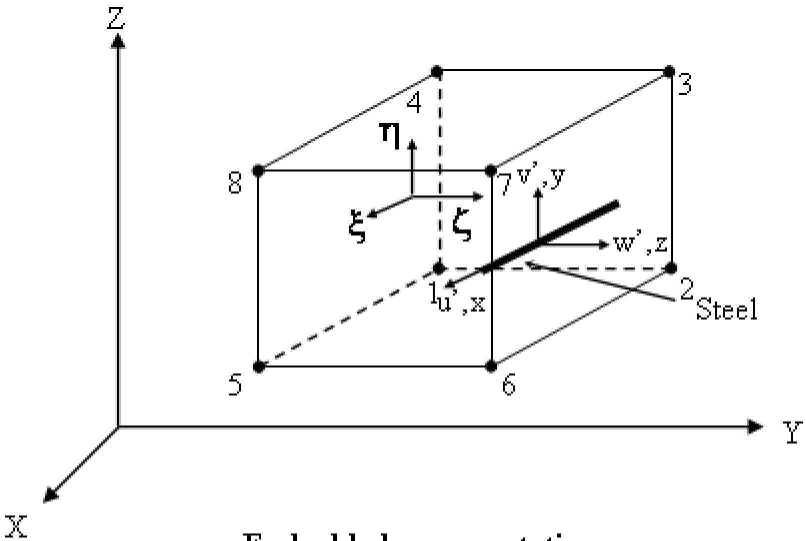
Fig.3 : Three dimensional 8-node brick element[10].



a. Distributed representation



b. Discrete representation



c. Embedded representation

Fig. 4: Reinforcement representation type

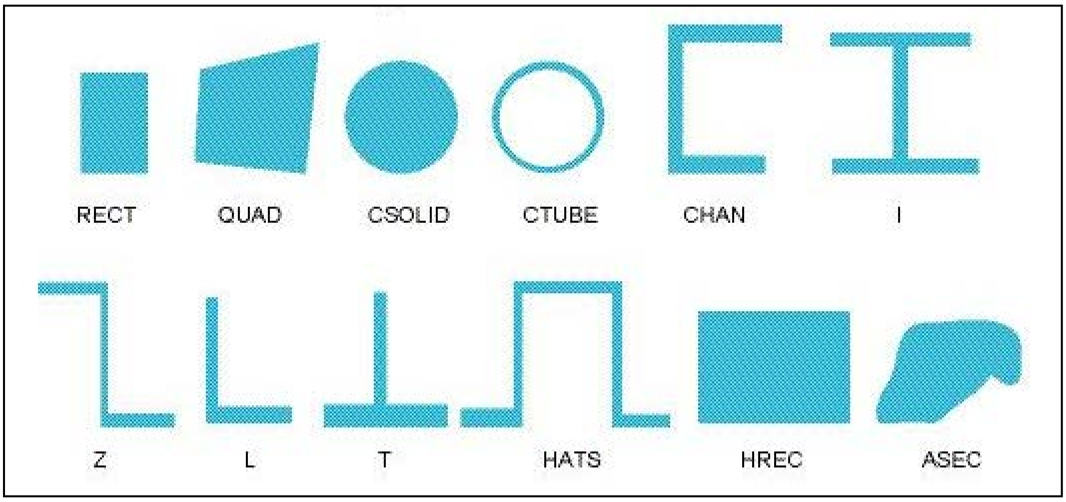


Fig.5: section type of element beam188[10]

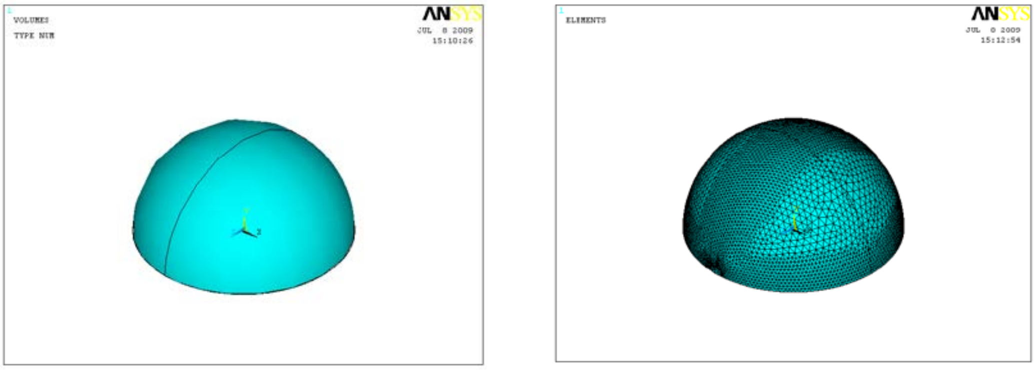


Fig.6 : ANSYS model of Dome

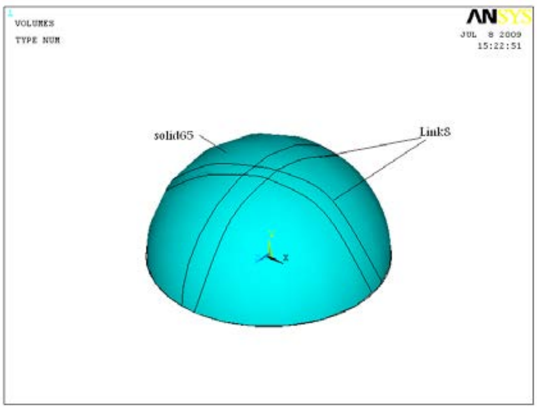


Fig.7 : Reinforcement of Dome

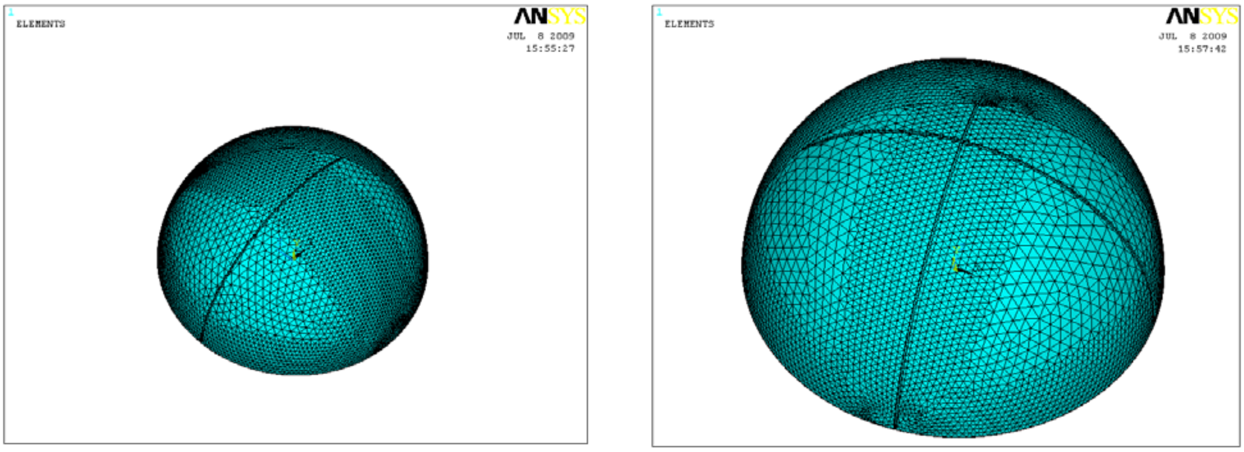


Fig.8 : Dome with one rib and two rib

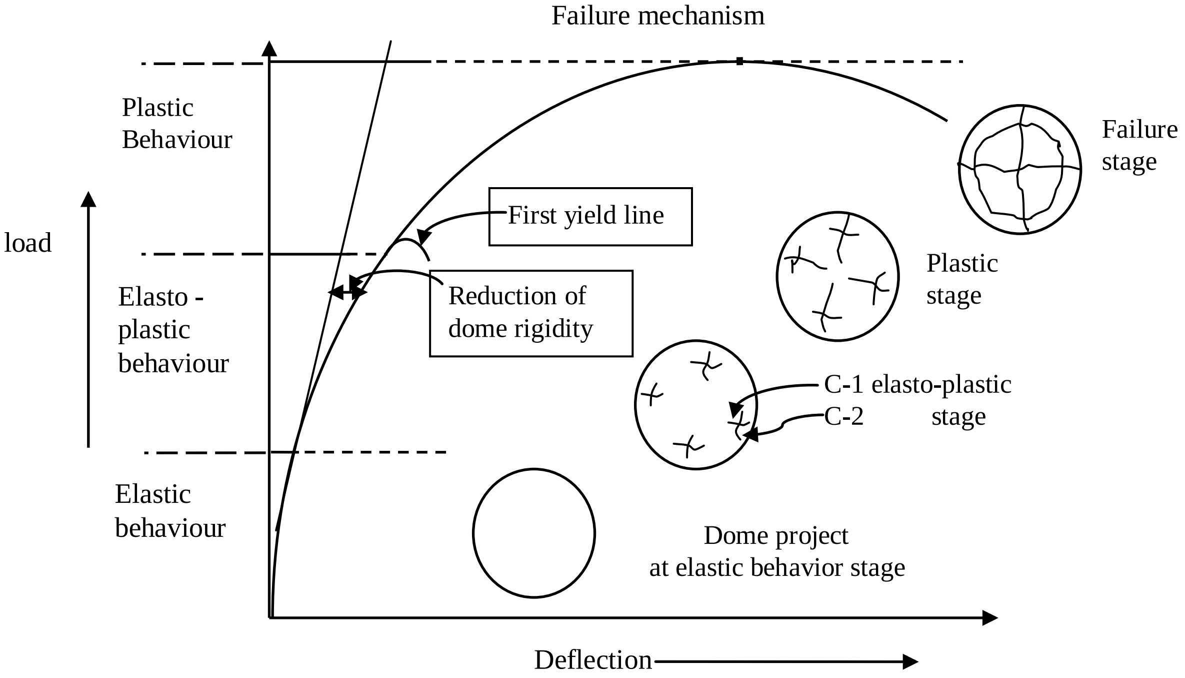
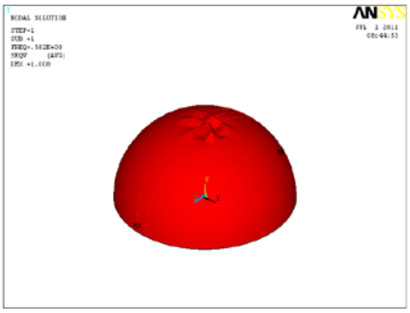
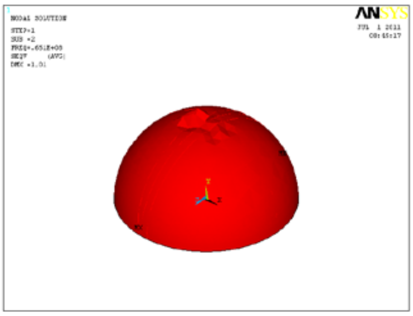
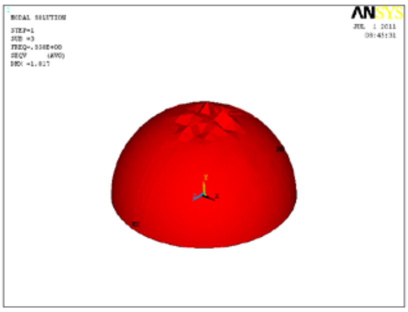
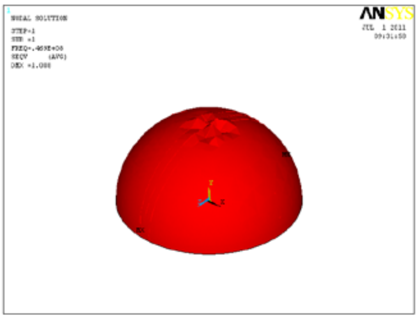
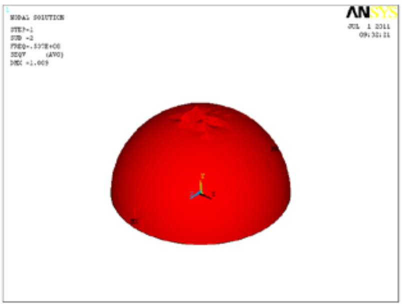
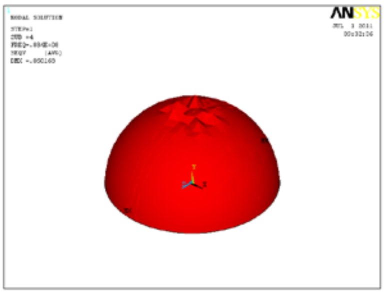


Fig.9 : Dome behavior under loading without buckling

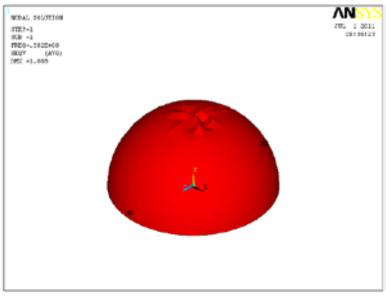
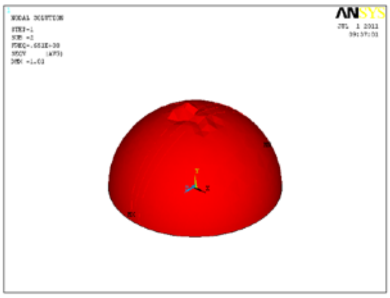
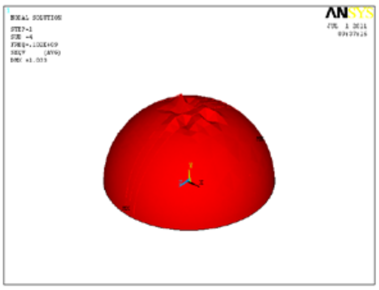
First mode Second mode Third mode

Fig.10 : Three mode shapes for the dome without ribs

First mode Second mode Third mode

Fig.11: Three mode shapes for the dome with one rib

First mode Second mode Third mode

Fig.12 : Three mode shapes for the dome with two ribs