LINEAR AND NONLINEAR BUCKLING ANALYSIS VIA ABAQUS

by

Kadmous Moufid El-Adas

Project submitted to the Faculty of the

Virginia Polytechnic Institute and State University

in partial fulfillment of the requirements for the degree of

Master of Engineering

in

Civil Engineering

APPROVED:

Siegfried Holzer

Dr. S. M. Holzer (Chairman)

Raymond H Plaut

Dr. R. H. Plaut

Don A Garst

Prof. D. A. Garst

AUGUST 1990

Blacksburg, Virginia

LINEAR AND NONLINEAR BUCKLING ANALYSIS VIA ABAQUS

by

Kadmous Moufid El-Adas

Dr. S. M. Holzer (Chairman)

Civil Engineering

(ABSTRACT)

Results of stability analyses with the commercial finite element program ABAQUS are presented. Buckling and nonlinear analysis of an Euler column and a Von Mises truss are presented. In addition, linear, buckling and nonlinear analyses are carried out on a Varax dome.

Acknowledgements

I would like to thank all those who have helped me in completing this project. Of the faculty members I would like to thank my advisor, Professor S. M. Holzer, for his help and guidance with the preparation of this project. I also wish to thank Professors D. A. Garst and R. H. Plaut for reviewing this material and being on my project committee.

I am also grateful to all the students who helped me. Among the graduate students I would like to thank Julio Davalos and John Huang for helping me in learning to use ABAQUS. I would also like to thank Jacem Tissaoui for aiding me in providing the necessary informaton of the Varax space frame geometry and element connectivity through use of the I-DEAS program.

Finally I would like to thank my parents for their continuous support throughout my education.

Table of Contents

List of Illustrations

List of Tables

Chapter 1 INTRODUCTION

The purpose of this project is to gain insight into the stability of structures through nonlinear finite element analyses. The commercial program ABAQUS is used to carry out the linear and nonlinear analyses of the various structures considered in this project. The structures consist of an Euler column, a Von - Mises truss and a Varax dome.

The Euler column and Von-Mises truss are structural problems whose finite element solutions can be checked by comparison to continuum solutions. For more complicated structures, like the Varax dome, the problem is best approached by making a linear run first and then by introducing nonlinearities into the problem.

A buckling analysis is carried out for the Euler column. The Von - Mises truss has a nonlinear and buckling analysis carried out. The Varax dome, of the Church of the Nazarene, is analyzed under two different conditions. The first condition considers the case in which the whole dome is composed of truss elements, and the second case considers the dome, except for the tension ring elements, to be composed of beam elements. A buckling analysis is carried out for both the space frame and the space truss models of the dome.

Finally an incremental, iterative and geometrically nonlinear analysis for both models of the dome is carried out. Results are studied and comparisons and conclusions made about the behaviour of a space frame compared to that of a space truss. The assumption that the ultimate load capacity of domes is governed by geometric nonlinearities and not by material nonlinearities will be checked.

The effect of decking is not considered in this study. As a result the surface pressures are converted into member loads through use of shell elements.

One load combination is considered in the analysis of the two types of dome models. This combination consists of a uniform dead load and uniform snow load (Varax Engineering Co., 1986).

The commercial program ABAQUS is used to perform linear and nonlinear analyses on the dome. ABAQUS has the capability to trace geometrically nonlinear equilibrium paths by two methods : the Newton / Raphson method in which fixed or variable step incrementation can be used and the modified Riks / Wempner method.

ABAQUS can also predict the critical load using a linearized eigenvalue prediction which is particularly useful for " stiff " structures which have an almost linear pre-buckling load-displacement curve. The analysis is also useful for estimating the length of the equilibrium path which must be traced to locate the first critical point in a nonlinear analysis.

Setting up an input file and how to submit it is also discussed. The kinds of options and commands available in ABAQUS are discussed in some detail.

Chapter 2 STABILITY OF EQUILIBRIUM AND NONLINEAR ANALYSIS

Definition of Stability

"A system is classified as stable if infinitesimal disturbances cause only infinitesimal departures from the given equilibrium condition" (Langhaar, 1962). From a structural point of view an equilibrium configuration of a structure is said to be stable if accidental disturbances (forces, shocks, imperfections, eccentricities, vibrations, etc.) do not cause the structure to depart excessively or disastrously from that configuration (Langhaar, 1962).

Concept of Stability

The concept of stability is often explained by considering the equilibrium of a ball in various locations as shown in Fig. 1.1. If the ball in part (a) undergoes an infinitesimal displacement from its original equilibrium position it will return to that position once the disturbing force is removed. A body behaving in this manner is said to be in *stable equilibrium* . On the other hand if the ball in part (b) is displaced by an infinitesimal amount from its equilibrium position it will not return to its original position but will continue to move farther away from that position. The equilibrium in this case is a very precarious one and is referred to as *unstable equilibrium* . For the ball to go from a state of stable equilibrium to an state of unstable equilibrium it has to pass a state of *critical equilibrium* and the point at which the ball reaches this critical condition is referred to as a *critical point* (Thompson and Hunt, 1973).

A critical point is either a *bifurcation point* or a *limit point* . A bifurcation point (or branching point) is a point on the load-displacement curve where the fundamental (initial) equilibrium path intersects one or more secondary post-buckling paths. A bifurcation point may be either stable or unstable. A limit point is defined as a relative maximum on the load-displacement curve (Thompson and Hunt, 1973).

Behaviour of Structures

Structures in general can be in any of the states mentioned above for any loading condition. Usually structures that are symmetric have bifurcation points. When imperfections are introduced into the system the structure may buckle at a lower load and a limit point may exist in place of a bifurcation point (Thompson and Hunt, 1973).

Systems having a finite number of degrees of freedom have their applied forces related to their displacements by means of a stiffness matrix. The entries in the matrix in linear analysis are independent of the displacements; however in nonlinear analysis they become a function of the displacements.

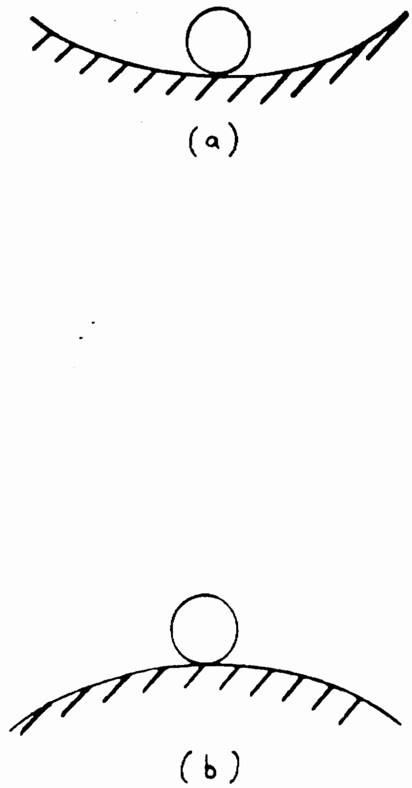


Figure 1. Fig. 1.1. Stability of equilibrium.

The stiffness matrix in nonlinear analysis is called the tangent stiffness matrix. An equilibrium state becomes unstable when the determinant of its tangent stiffness matrix becomes equal to zero (i.e. the matrix becomes singular) (Cook, 1989).

As loads are applied to an initially stable structure, the structure under increasing loading may eventually reach a *transition point* which marks the passing of the structure from a stable equilibrium state to an unstable equilibrium state.

The behaviour of a slender column is very similar to that of the ball explained above. The straight configuration of the column is stable for small loads but unstable for large loads. The load corresponding to the transition from stable to unstable equilibrium is referred to as the *critical load* . This is the load at which the straight configuration of the column becomes susceptible to any lateral disturbance. At the critical load level a line branches out. This line follows a different path than that of the previous load-displacement curve. This line, referred to as the secondary post-buckling path, is the alternative path which the structure will follow if a lateral disturbance occurs. The response of the column to any such disturbance is to take on a bent configuration. The bifurcation point in this situation is in itself stable and it separates an unstable region from two stable regions. The two stable regions are the portion of the curve representing the load level below the bifurcation point and the secondary post-buckling path. The unstable region is the primary equilibrium path beyond the bifurcation point.

The secondary (postbuckling) path for the column rises which indicates that the structure has postbuckling strength.

The Von-Mises truss has no bifurcation point because it is a one degree of freedom system. It does however have a limit point which if exceeded causes the structure to *snap-through*. The structure reaches a stage in the loading process where a slight increase in load results in the structure going through a sudden jump in displacement values. The structure "snaps-through" to a post-buckled state which resembles the original structure in an inverted form.

Nonlinear Analysis

The usual aim of nonlinear analysis is to estimate the maximum load that a structure can support before structural instability and collapse occur.

In nonlinear analysis of proportional loading cases the load distribution on the structure is known but the load magnitude that the structure can sustain is unknown.

Shallow arches and spherical caps under uniformly distributed loads usually undergo "snap-through" failure. These structural systems, which are nonlinear, initially deform slowly with increasing load. As the load approaches the maximum value the deformation rate increases until, reaching a state of critical equilibrium, the shallow arches and caps subsequently "snap-through" to a post-buckled state.

To perform effective nonlinear analysis a good physical and theoretical understanding is essential. The best approach to performing nonlinear analysis is to use reliable and generally applicable finite elements to build up relatively small and reliable models whose behaviour can be easily understood. A linear analysis is performed first and the results checked to see whether the results are reasonable. The model is later refined by introducing the nonlinearities as desired. An incremental approach, measured in load or time steps, is used in nonlinear analysis. Time is an artificial variable indicating the load level in nonlinear static analysis with materials having time-independent material properties. The three basic requirements of mechanics have to be satisfied at the end of each load or time step. These requirements are :

- Equilibrium
- Compatibility

- Stress strain law

(Bathe, 1986)

Buckling

Buckling is a type of behaviour to which slender compression members (e.g. columns) are susceptible.

As long as the load on a slender compression member is relatively small, increasing the load results only in a further axial shortening of the member. Once a critical load is reached, however, the member, if subjected to a lateral disturbance, will suddenly bow out sideways (i.e. axial load is transformed into bending deformations) . This bending gives rise to large deformations because for absorbing an amount of strain energy the flexural stiffness, being much smaller than the axial stiffness, has to permit deformations which are much larger than axial deflections.

The conditions under which buckling occurs depend on a variety of factors which include: the member dimensions, the way in which the structure is supported, and the properties of the structural material.

For the Euler column the buckling load is the limiting load under which, with no lateral disturbance, compression in an unbent configuration is possible. The buckling load is therefore the critical load which marks the limit of stability for the unbent configuration. The transition from the straight to the bent configuration occurs if the straight configuration is subjected to a lateral disturbance at or beyond the bifurcation point.

At a bifurcation point a reference configuration of the structure and an infinitesimally close (buckled) configuration are both possible equilibrium configurations for the same load. For a buckling displacement taking place from a reference configuration, no change in load takes place.

Once past the bifurcation point the deformations on the load-deflection curve begin to grow in a new pattern which is quite different from the pre-buckling pattern.

It is assumed in linearized buckling analysis that nonlinearities and geometric changes prior to buckling can be ignored. As a result it can be assumed that the linear strain stiffness matrix does not change appreciably prior to buckling and that the nonlinear strain stiffness matrix is just a multiple of its initial value (Bathe, 1982).

ABAQUS has the capability to estimate elastic buckling by eigenvalue extraction. For "stiff" structures this method of estimation is particularly useful since these structures have an almost linear pre-buckling response.

Axial force effects are accounted for by the matrix k^{σ} which is added to the conventional stiffness matrix k. The matrix [k^{σ}] is called the initial stress stiffness matrix. It is determined by element geometry, displacement field and state of stress. As a result it is independent of elastic properties (Cook, 1989).

The system stiffness matrix [K^{σ}] of the structure is built by adding overlapping terms of element matrices k^{σ} the same way as the conventional system stiffness matrix K is built by summing overlapping terms of element matrices k . The tangent stiffness matrix is $k + k^{\sigma}$ for an element and $K + K^{\sigma}$ for a structure. By accounting for both conventional and stress stiffness the effect of axial loads on bending stiffness is taken into account (Cook, 1989).

Chapter 3 EULER COLUMN ANALYSIS

Classical Solution of the Euler Column Problem

The Euler column shown in Fig. 3.1 is assumed to have the following properties:

1- A constant cross-sectional area.

2- The material obeys Hooke's law.

3- The material is homogeneous.

4- The deformations are small enough so that in the expression

$y'' / [1 + (y'^2)]^{3/2}$

the term y'^2 is negligible with respect to unity and so the curvature can be approximated by y''.

For the shape in Fig. 3.1 the internal resisting moment at a point a distance x from the origin is

$$M(x) = -EIy'' \qquad [3.1]$$

Equating it to the externally applied bending moment Py gives

$$EIy'' + Py = 0 \qquad [3.2]$$

Letting

$$k^2 = P/EI \qquad [3.3]$$

Eqn. 3.2 becomes

$$y'' + k^2y = 0$$

The general solution is

$$y = C_1e^{ikx} + C_2e^{-ikx}$$

Using

$$e^{-ikx} = \cos kx \pm isinkx$$

the answer can be written as

$$y = A \text{ sinkx} + B \text{ coskx}$$

Using the boundary conditions:

$$y = 0 \text{ @ } x = 0$$

$$y = 0 \text{ @ } x = L$$

we find

$$B = 0 \Rightarrow y = A \sin kx \qquad [3.4]$$

$$Asinkl = 0$$

For a nontrivial solution

$$\sin kl = 0$$

$\therefore kl = n\pi$ for n = 1, 2, 3, ...

Substituting into 3.3 and 3.4 leads to

$$P = n^2\pi^2 EI/l^2$$

and $y = A \sin n\pi x/l$

For n = 1

$$P = \pi^2 EI/l^2$$

and $y = A \sin \pi x/l$

This load, known as the Euler load, is the smallest load at which the column can bend sideways if subjected to a lateral disturbance.

At the Euler load there is a bifurcation point. This means the column can either remain straight or can take a deformed shape of unknown amplitude. This shows that at the Euler load the transition point from stable to unstable equilibrium has been reached.

A bifurcation point exists if the column is perfectly straight, homogeneous, and has a perfectly centered axial force only. In practice there are always imperfections and real columns do not have bifurcation points but "limit points". A computed buckling load is therefore only an approximation of how much load a real structure can carry.

Large-deformation theory predicts a very slight increase in load with increasing deflection during the early stages of bending and a more marked increase in load after considerable deflection has occurred.

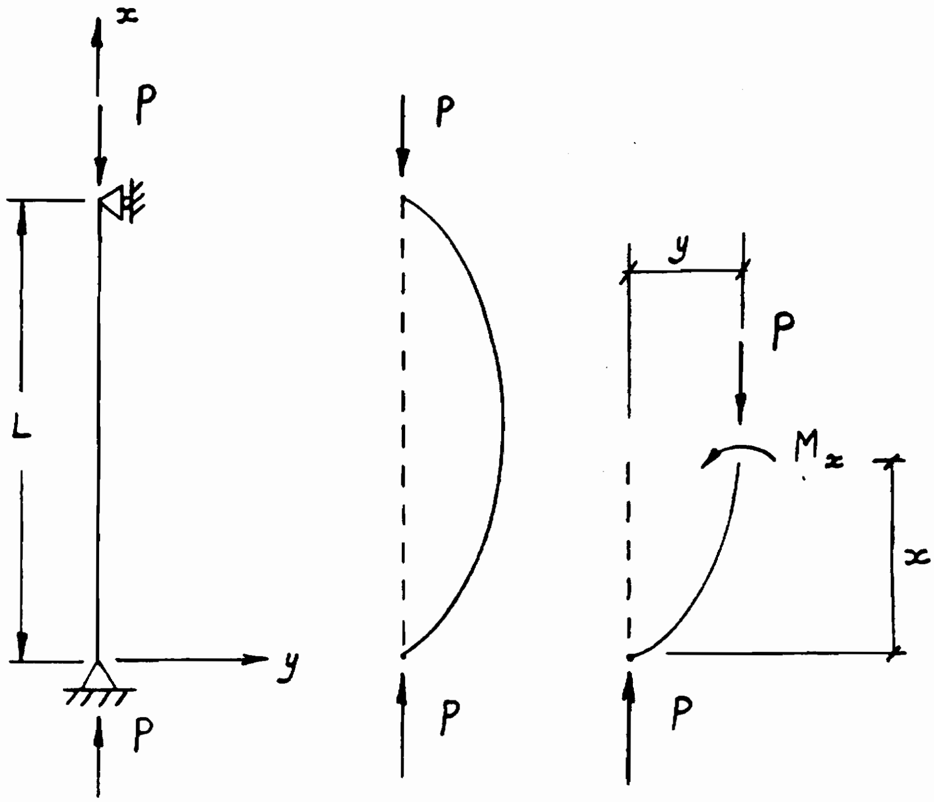


Figure 2. Fig. 3.1. Euler column.

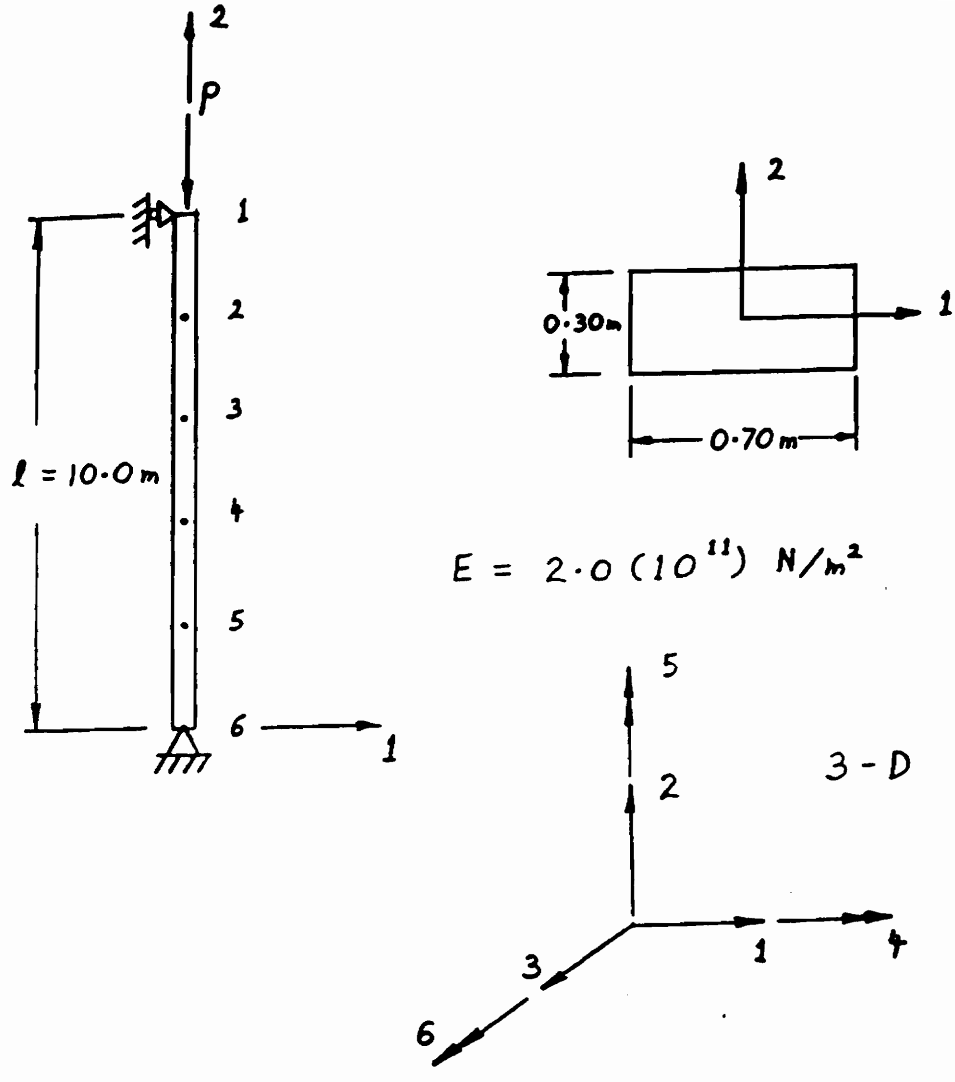


Figure 3. Fig. 3.2. Euler column 5 element mesh.

Finite Element Analysis of Euler Column

The buckling load analysis is carried out for the case of the "perfect" Euler column. The following meshes have been used : 2 element, 3 element, 4 element and 5 element.

The B33 element of ABAQUS has been used. This element models a beam in space, consists of three nodes and uses cubic interpolation. The results are shown in table 3.1. As expected, the finer the mesh used, the closer is the finite element answer to the continuum answer.

TABLE 3.1. Results of buckling load estimate analyses.

Mesh	Buckling load around strong axis (N)		Buckling load around weak axis (N)	
	(Theoretical)	(Estimated)	(Theoretical)	(Estimated)
2 Element	1.69264(10^8)	1.70332(10^8)	3.10892(10^7)	3.13165(10^7)
3 Element	1.69264(10^8)	1.69412(10^8)	3.10892(10^7)	3.11352(10^7)
4 Element	1.69264(10^8)	1.69247(10^8)	3.10892(10^7)	3.11031(10^7)
5 Element	1.69264(10^8)	1.69185(10^8)	3.10892(10^7)	3.10942(10^7)

Chapter 4 THE VON MISES TRUSS

The Von Mises truss problem is a simple example of a geometrically nonlinear problem involving a flexible structure. A flexible structure, unlike a "stiff" structure, is one that exhibits nonlinear behaviour throughout its deformation. Since the problem only involves one degree of freedom it is a very simple problem to solve in closed form. The type of equilibrium response is in some respects typical of the response in some important practical cases.

The truss consists of a uniform rod, pinned at one end and free to slide vertically at the other end. The material is assumed to be entirely linear elastic; as a result the only nonlinearity arises from rotation. The initial height h of the movable end of the rod above the horizontal position is considered to be small compared to the horizontal distance between the supports, d. As a result the strain in the rod is small as long as $0 \leq u \leq 2h$ (ABAQUS, 1989).

To find the limit point of the structure, a force is applied to displace the free end downwards. No bifurcation is possible since only one displacement variable is involved. The system response is therefore quite simple compared to what might happen in other structural systems with many degrees of freedom.

Classical Solution of the Truss Problem

The element being considered is a truss element so logarithmic strain is used and the force is taken as uniform and equal to the stress multiplied by the cross sectional area (ABAQUS, 1989).

Strain in the truss is assumed to be uniform so:

$$\varepsilon = \ln(l/L)$$

where L is the original length and l is the current length

From Fig. 4.1

$$l^2 = d^2 + (h-u)^2$$

and

$$L^2 = h^2 + d^2$$

so that strain is obtained as

$$\varepsilon = \frac{1}{2}\ln(1 + u\frac{(u-2h)}{L^2}) \qquad [4.1]$$

and the first variation obtained as

$$\delta\varepsilon = \frac{1}{l^2}(u-h)\delta u \qquad [4.2]$$

The material is assumed to respond in a linear elastic manner so

$$\sigma = \mathrm{E}\varepsilon$$

Assuming the initial cross sectional area to be A and the material to be incompressible, the virtual work equation is

$$\int \sigma \delta \varepsilon \mathrm{AdL} = P\delta u \qquad [4.3]$$

Since stress and strain are uniform

$$\int \sigma \delta \varepsilon \mathrm{AdL} = \mathrm{AL}\sigma\delta\varepsilon$$

Using Eqns. 4.1, 4.2 and 4.3 in the above equation yields

$$P = \frac{AEL}{l^2}(u - h)\ln(l/L)$$

which is the static equilibrium equation of the system shown in Fig. 4.3.

Analysis by ABAQUS

The analyses run are for computing the buckling load of the truss and tracing the equilibrium path.

Buckling Analysis

The buckling load is computed from the formula

$$\mathrm{P}_{total} = \mathrm{P}_0 + \lambda(\mathrm{P}_{ref} - \mathrm{P}_0)$$

where P_0 is the magnitude of the load component at the start of the step; P_{ref} is the magnitude of this component as defined in the input file; λ is the load proportionality factor; and P_{total} is the current magnitude of the load component (ABAQUS, 1989). The "loads" referred to above are forcing functions which can be either prescribed loads or prescribed displacements. Since the DIRECT parameter is used, constant increments of displacement are used. The increment size is obtained by dividing the specified terminating displacement by the maximum number of increments specified in the input file. If the parameter had been omitted, ABAQUS would have chosen the increment sizes after trying the user's initial increment for the first attempt at the first increment.

Tracing Nonlinear Path

The nonlinear analysis is carried out by using the DIRECT option in ABAQUS. The limit imposed on the deflection is 3 units of length in the downward direction, so the increments ABAQUS uses are constant fractions of this length as forcing functions and the applied load is obtained as a reaction.

Results

The eigenvalue prediction results are presented in Fig. 4.2. Table 4.1 lists some of the buckling load estimates and their variation from the actual load. The eigenvalue estimates of the points used in determining the graph drop from an estimation of 27% higher than the actual value to an estimation of 6.5% higher than the actual value.

For constant increments of live load the buckling estimate converges to the exact answer. The closed form solution for the truss is 0.388 units of force.

To estimate the critical load , P_{cr}, the formula

$$P_{cr} = P_D + \lambda_{cr} P_L$$

Table 4.1 Eigenvalue estimates of critical load.

P_d	P_L	α	β	c_{cr}	λ_{cr}	P_{cr}
0.25	0.02	1.0	0.0	12.15	12.15	0.493
0.27	0.02	1.0	1.0	10.38	11.37	0.478
0.29	0.02	1.0	2.0	8.58	10.58	0.462
0.31	0.02	1.0	3.0	6.83	9.826	0.446
0.33	0.02	1.0	4.0	5.01	9.010	0.462
0.35	0.02	1.0	5.0	3.16	8.157	0.413

${}^{b}\lambda_{cr}$: Predicted critical load parameter.

λ_{cr} : Actual critical load parameter.

$$\lambda_b = \alpha + \beta$$

$$\lambda_{cr} = \beta + \alpha c_{cr}$$

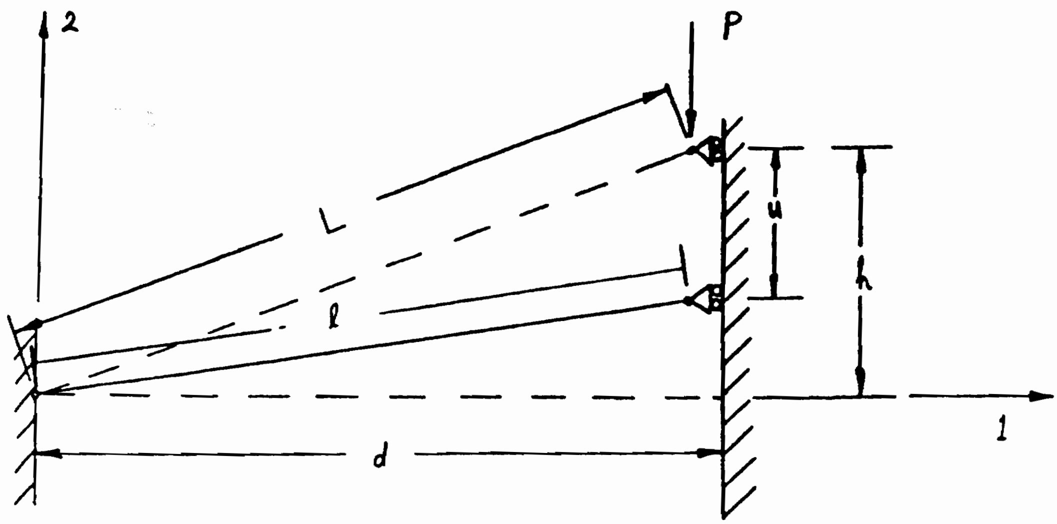
$$\mathrm{P} = \mathrm{P}_D + \lambda\, \mathrm{P}_L$$

$$\mathrm{P}_{cr} = \mathrm{P}_D + \lambda_{cr}\, \mathrm{P}_L$$

is used. The dead load, P_D, is taken as 0.25 units of force while the increments of live load, P_L, are taken as 0.02 units of force. The graph of the predicted buckling values is extended until it intersects the 45 degree line and the critical load proportionality factor, λ_{cr}, is read as 7.1. Using the above equation P_{cr} is found to be equal to 0.392 force units which overestimates the exact value by only 1.0 %.

The results of tracing the nonlinear path are shown in Fig. 4.3. The initial truss response is stable and almost linear. As the deflection increases the element stiffness decreases until a limit load P_{cr} is reached. The corresponding displacement for P_{cr} is about 42% of *h*. Beyond this value of displacement the system has negative stiffness and the response is unstable until the displacement reached is about 158 % of *h* at which point the equilibrium configuration again becomes stable. In the range $-P_{cr} < P < P_{cr}$ the system has 3 static equilibrium configurations, two of which are stable and one is unstable. Beyond this range the system has only one static equilibrium configuration which is stable. The conclusion is that for even a simple elastic system having only one degree of freedom, introducing geometric nonlinearities may lead to loss of uniqueness and stability of the solution.

For this simple problem it was enough to specify the only active degree of freedom of the system in order to obtain the equilibrium solution for even the unstable response phase. For more practical cases the "Riks" algorithm must be used instead (ABAQUS, 1989).



$E = 2000$
$A = 1$
$L = 10$
$h = 1$

Figure 4. Fig. 4.1. Von-Mises truss model.

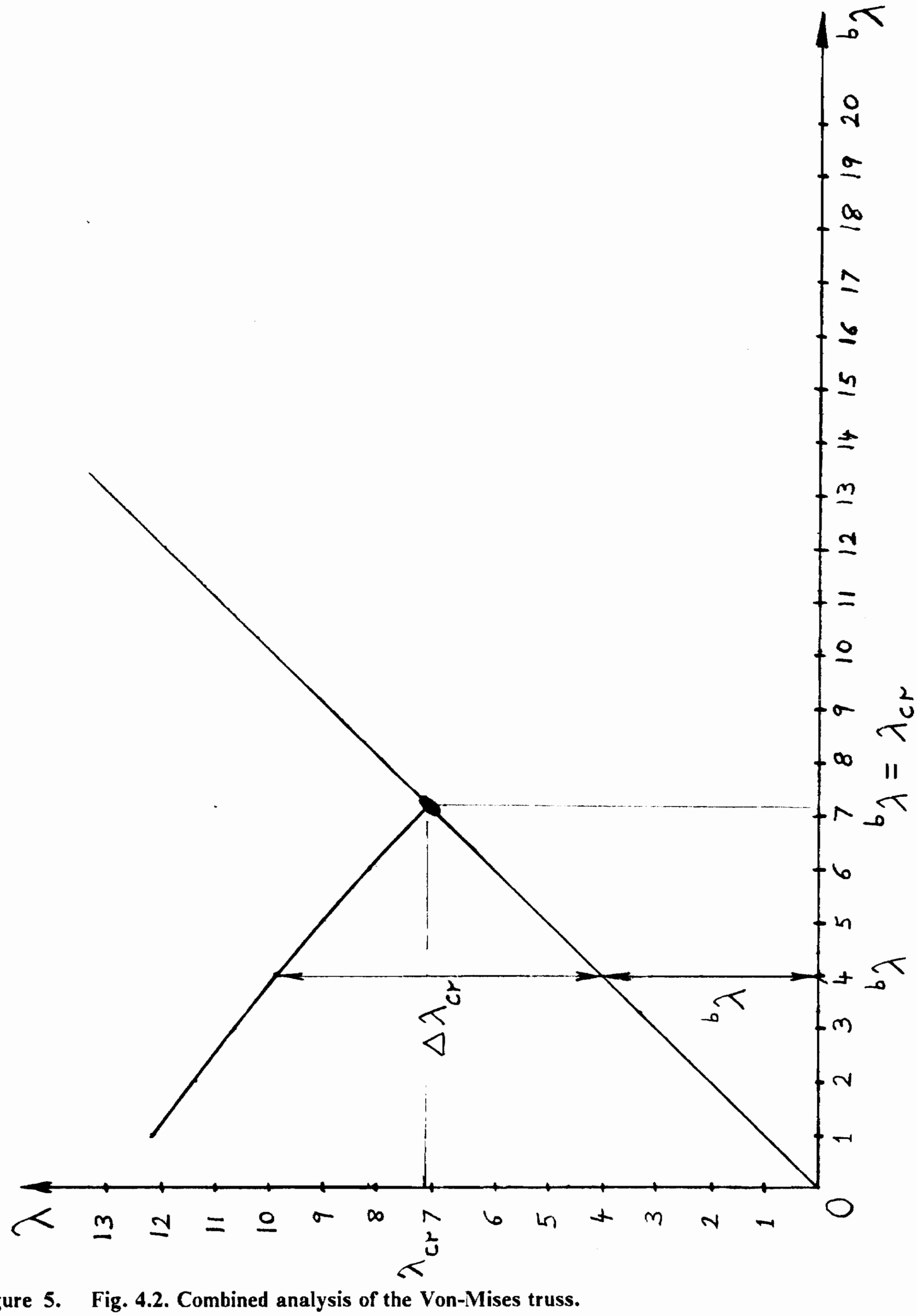


Figure 5. **Fig. 4.2. Combined analysis of the Von-Mises truss.**

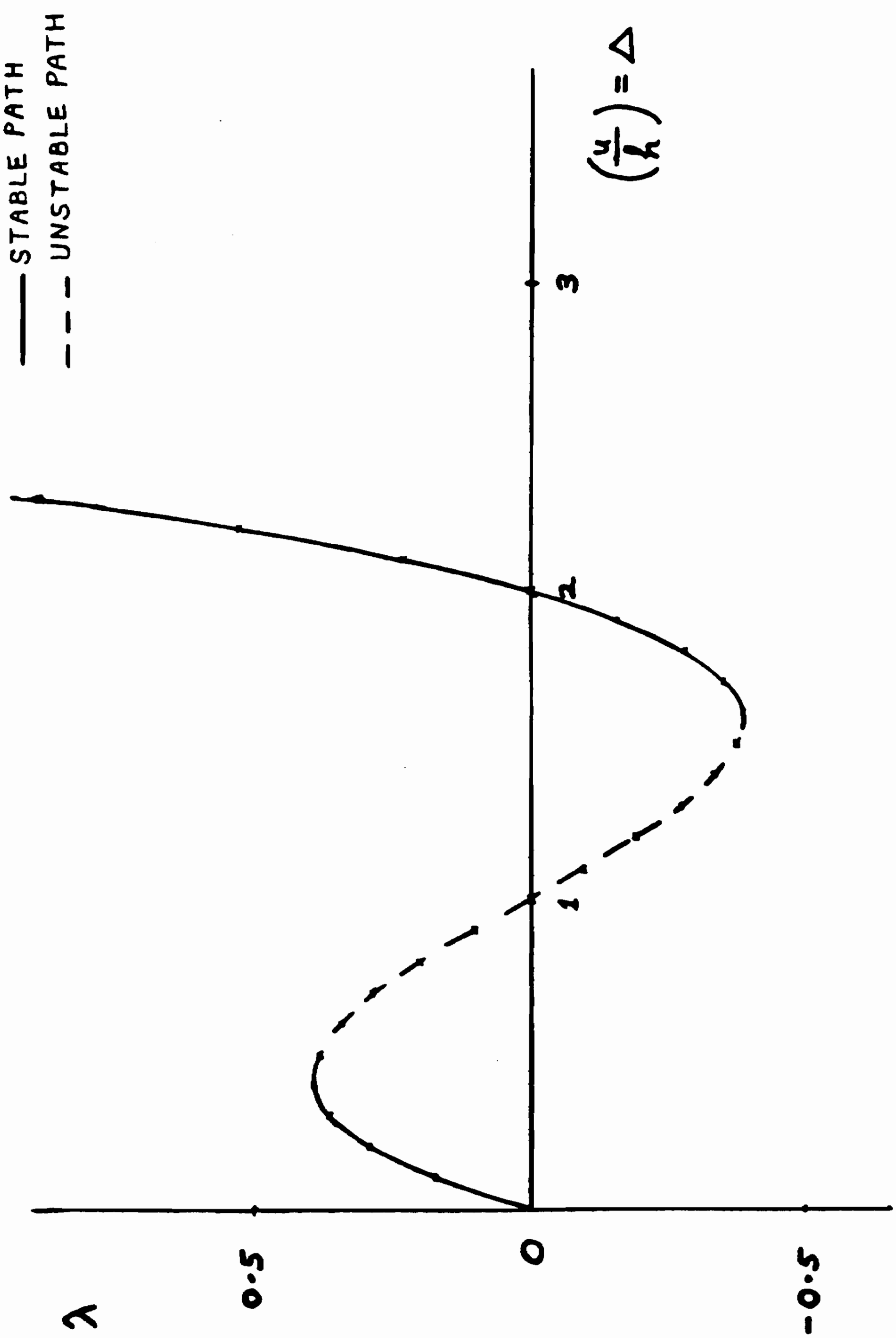


Figure 6. Fig. 4.3. Nonlinear equilibrium path tracing.

Chapter 5 VARAX DOME

Introduction

This part of the study contains the linear and nonlinear analyses of a VARAX dome which has been built in Corvallis, Oregon.

Glulam domes have evolved into well designed structures over the years. In spite of this their complete structural response has not been defined and the design is based on simplified methods of analysis. Elastic instability, which is governed by geometric nonlinearities, is the dominant type of failure for domes similar to the one being studied (Holzer and Loferski, 1987).

The dome is analyzed first as a space truss and then as a space frame. Each dome model is analysed for one loading condition. The analysis consists of three parts: a linear analysis to find the stresses and deflections under service (working) conditions, a linear buckling analysis to obtain an idea of the ultimate load capacity of the dome and a nonlinear analysis to trace the response until a critical load is reached.

The deflections and stresses for the truss and frame models are compared and some conclusions are drawn. The assumption that instability of the dome models is governed by geometric nonlinearities and not by material nonlinearities will be checked.

The general dimensions of the dome in profile for both cases are shown in Fig. 5.1.

The plan view with node numbering of the nodes for the space truss model is shown in Fig. 5.2, and Fig. 5.3 shows the profile view of the truss model.

The most important design loads for lattice domes are wind and snow loads. For all types of analysis the loading for both types of models is the same and consists of a 15 psf uniform dead load and a 25 psf uniform live load (Varax Engineering Co., 1986).

Geometry

The dome being analysed is made out of glulam longleaf pine with the exception of the tension ring members which are steel members. The dome has a diameter of 130′ and lies on a great sphere having a radius of 106′. The height of the dome above the tension ring is 22.27′ .

The tension ring extends along the dome perimeter. To eliminate eccentric ring moments the tension ring is made out of straight members. Each sector of the dome consists of a number of triangles (or panels). The space truss in elevation is shown in Fig. 5.4.

Since the radius of curvature of the dome is relatively large (106′), the chord and arc lengths of members are approximately equal. Thus each trangle is decomposed into flat shell elements. The film elements convert the uniform load into member and nodal forces on the beams and purlins.

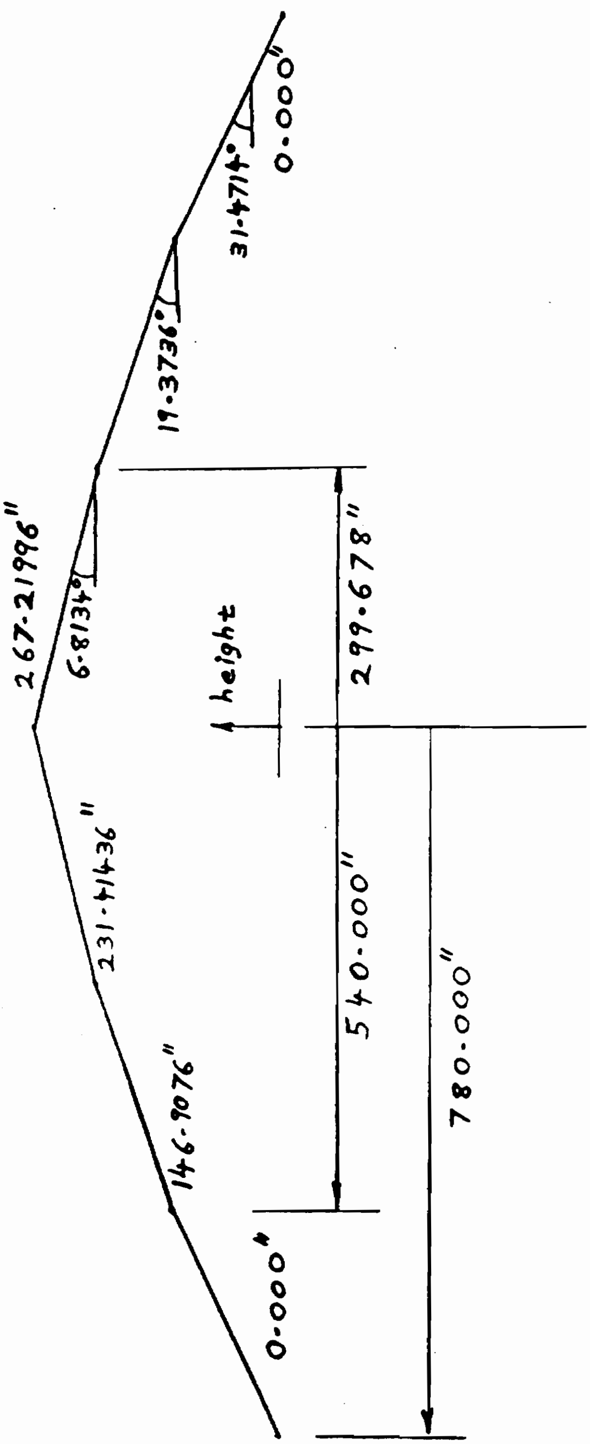


Figure 7. Fig. 5.1. General dimensions of dome in profile.

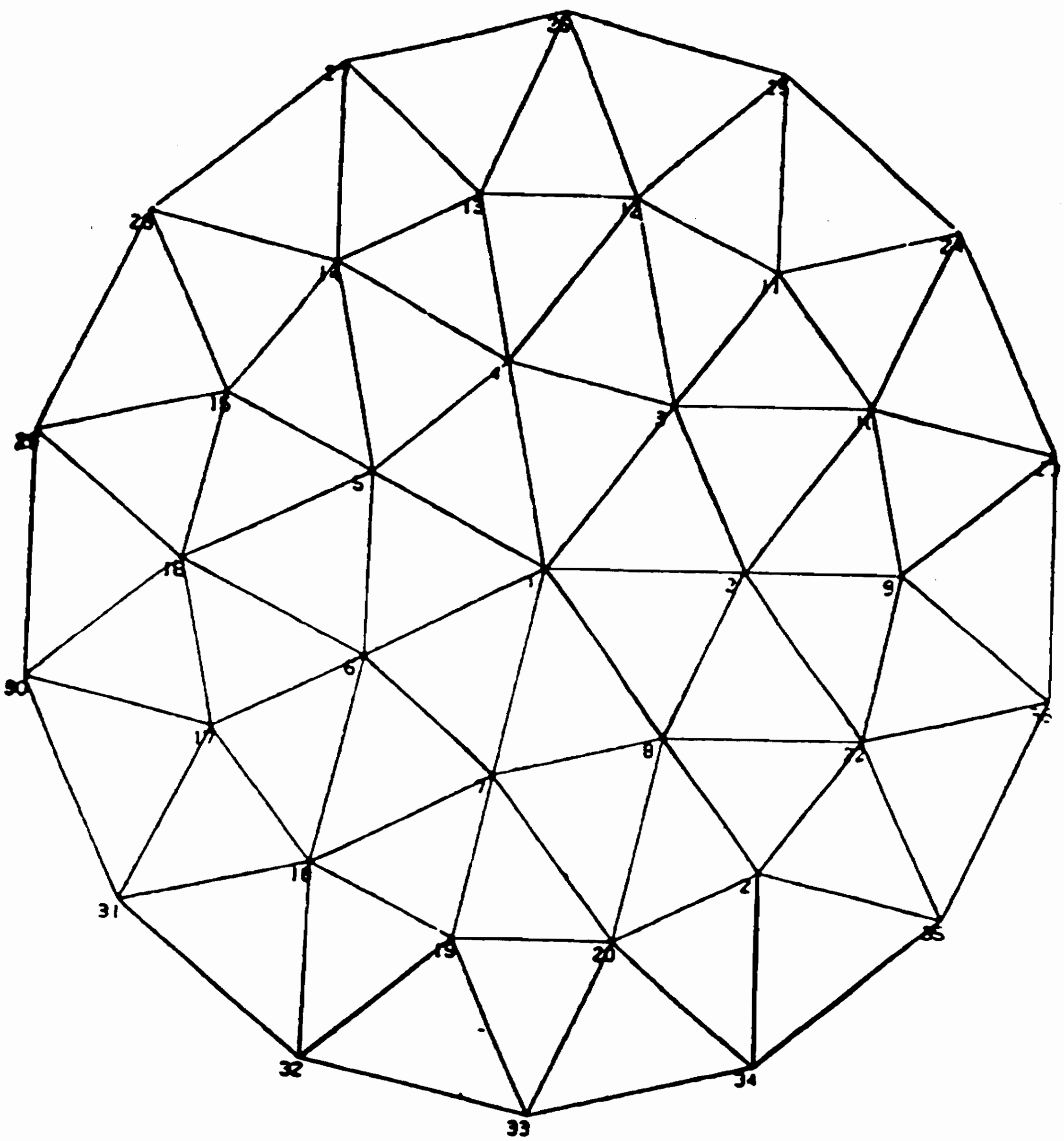


Figure 8. Fig. 5.2. Plan view of space truss with node numbering.

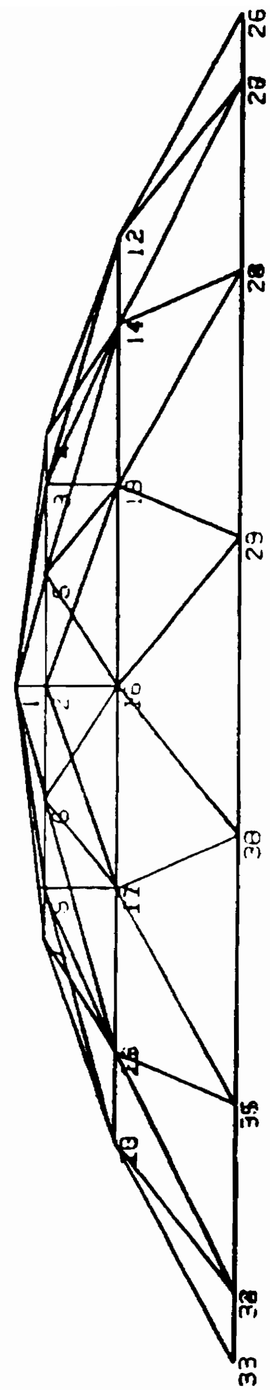


Figure 9. Fig. 5.3. Elevation view of space truss with node numbering.

The dome is made out of seven identical sectors. For the space truss each sector has a total of 14 truss members. Of these 13 are glulam truss members and one is a steel truss member. For the space frame each sector has a total of 71 flexural members and two truss members. Of the flexural members, 39 are main beams, 22 are purlins and 3 are edge beams. Each sector has a total of 152 shell elements. There are 6 different types of triangular panels in each sector, making up a total of 9 triangular panels per sector.

All the flexural members of the dome lie on great circles. Great circles are the intersections of a plane passing through the center of the sphere with the spherical surface.

The flexural members of the dome are for the most part inclined at some angle to the vertical. The only exceptions are the flexural members passing through the apex of the dome. These members in plan view radiate out from the dome center as spokes of a wheel and have their top edges parallel to the horizontal plane.

Originally only the nodal point coordinates making up the structure shown in Fig. 5.2 were available. An I-DEAS program, developed by Jacem Tissaoui, was used to generate the rest of the nodes by using the original 36 nodes. The nodes obtained as well as the original define the end and midpoint locations of all the flexural members in the dome which are B32 elements. The I-DEAS program also provides the member connectivity information which indicates the nodes which make up each element. This member connectivity information not only indicates which nodes make up each flexural member but also which nodes make up each of 280 quadrilateral and 784 triangular film elements which are needed to transfer uniform dead and snow load into concentrated loads at the midpoint and end nodes of each flexural member.

Members

The dome has 5 different types of members. The members making up the tension ring are steel members having rectangular cross sections of 5.0″ × 1.0 ″. The other four types are beam members and all have rectangular cross sections. The main beams have a cross section of 5.125″ × 12.0″. There are two types of purlins used. The first type has a cross section of 3.125″ × 7.5″ and the second type has a cross section of 5.125″ × 7.5″. The last type of member is an edge beam having a cross sectional area of 3.125″ × 22″. All beams are inclined such that the centroids of the cross sections of each beam all lie in one plane which passes through the center of the great sphere on which the dome lies.

Modelling

Truss elements model the tension ring. ABAQUS B32 elements are used to model the curved wood members (main beams, the two types of purlins used, and the edge beams) of the dome.

To model curved beams and purlins, quadratic interpolation is necessary. The B32 element has the ability to do this, since quadratic interpolation is associated with its three nodes, and it has the ability to account for shear deformations. It may be necessary to include shear effects in the element formulation because shear deformation might be significant in wood material.

Use of the ABAQUS B32 element involves the following assumptions:

1- Cross sections remain plane before and after bending but are not necessarily normal to the centroidal axis.

2- The B32 element does not incorporate warping (i.e., there is no deformation normal to the plane of the cross section).

3- Quadratic interpolation discretization functions are used.

4- Material nonlinearity is included in the formulation.

Use of the ABAQUS C1D2 element involves the following assumptions:

1- Only axial deformations take place.

2- The interpolation functions are linear.

Beam Orientation in Space

The orientation of members in space is defined in ABAQUS by use of a vector called the local 1 axis of the element. ABAQUS requires that the 1 axis of a flexural cross section has to be specified in order for the inclination of a beam member to be taken into account.

Direction cosines of the local 1 axis for each of the flexural members in the dome are obtained by use of the vector cross product operation. The vectors operated on are directed from the center of the great sphere to the nodes at the ends of each flexural member. The vector connecting the lower node number is post multiplied by the vector having the upper node number at its tip and a vector perpendicular to both is obtained. The division of this vector by its magnitude results in a unit vector which is the local 1 axis for the cross section since it is normal to the plane in which the beam lies. This vector is the local 1 axis specified in the ABAQUS input file for each beam cross section. An illustration of the local 1 axis is provided in Fig. 5.4.

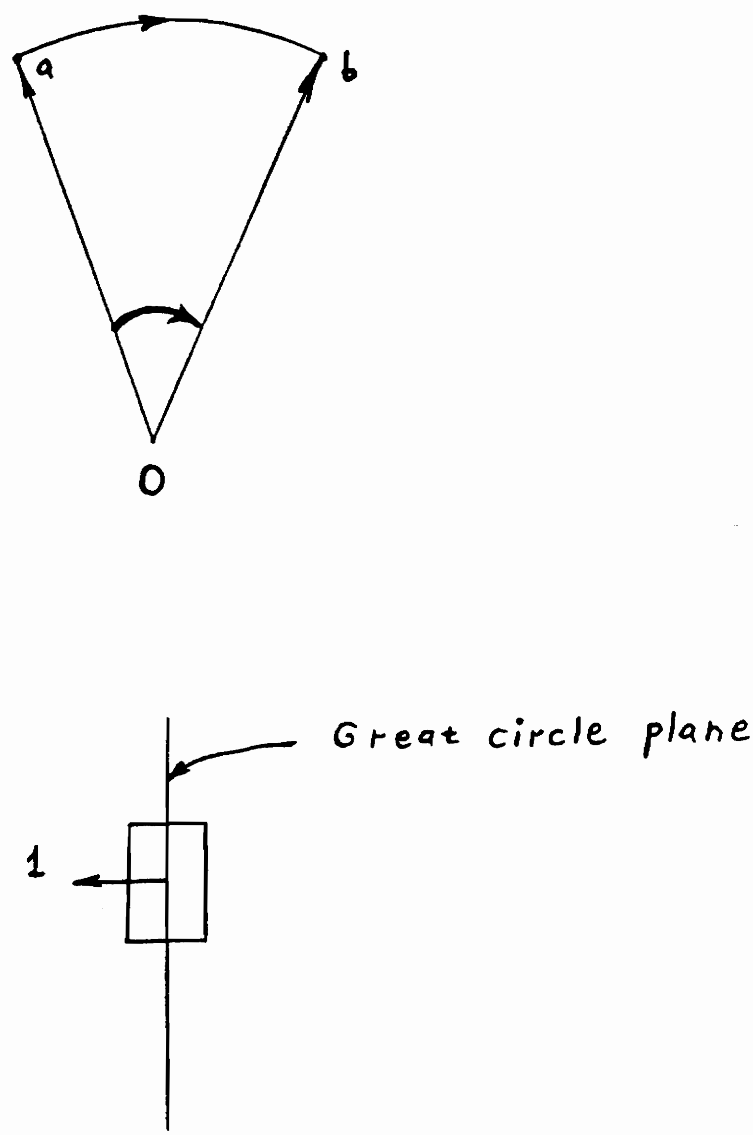


Figure 10. Fig. 5.4. Local 1 axis of curved beam.

Transformation of Load

By using STI3 and S4R shell elements, uniform load is changed into concentrated loads. The unit load (which has to be specified in units of force per unit length[3]) is multiplied by the shell thickness to produce a uniform load on the shell elements. Analysis is carried out to find the concentrated forces transferred at the corners of the element. These concentrated loads are exerted on the beams, purlins and edge beams of the dome.

Analyzing the Dome

For each type of dome three different analyses are carried out: a linear analysis to check behaviour under working stress conditions, a combined linear/nonlinear buckling load prediction analysis, and a nonlinear analysis.

In running the linear and nonlinear analyses of the space frame, two stages are necessary for each analysis. The first stage consists of finding the concentrated loads at the nodes which result from the distributed loading. All the beams and purlins are taken out and all the nodes are fixed. A linear analysis is performed to obtain the reactions at the nodes. The second stage involves keeping the beams and purlins while eliminating the shell elements. This greatly reduces the cost of a linear and a nonlinear analysis. The reactions obtained in the first stage are applied as concentrated loads at the nodes.

Analysis Results

As expected, the response of both models is cyclically symmetric both in displacement and stress responses for the linear runs. For the nonlinear cases the frame model is cyclically symmetric in the vertical displacement and stress values but the truss model has no symmetry in displacement values after the bifurcation point.

For both linear and nonlinear analyses the vertical deflections at the apex of the frame model are smaller than those of the truss model. This is also true for the maximum vertical deflections which occur at nodes close to the apex for both models.

For the linear runs the maximum stresses in the main beam members of the frame model are over twice as high as the stresses in the members of the truss model. This can be attributed to bending which is absent in truss elements. This would also account for the tension ring elements in the frame model having a lower stress than those of the truss model, because more strain energy has to go into the glulam members of the frame.

For the nonlinear analyses cases the response of the truss model is similar to that of the frame model, the reason being that both the truss and frame models are loaded at their nodes. Both models have an almost linear prebuckling path and bifurcation point. The frame model turns out to be softer than the truss model. The critical snow load proportionality factor for the truss model is over twice as high as that of the frame model (9.88 versus 4.31). The critical pressure at which the truss model becomes unstable is 262 psf compared to a critical pressure of 123 psf in the frame model. The type of instability is a bifurcation instability for both models. The truss model has a higher failure load than the frame model because member buckling is eliminated when truss elements are used instead of beam elements.

Linear Analysis Results

The displacement results for the linear analysis of the truss model are shown in Table 5.1.

The elements of the steel tension ring are the only elements of the dome which experience tensile stresses.

The wooden members are all in compression, while the tension ring has tensile stress in all members. The response is symmetric around the ring of nodes closest to the apex. Along the ring of nodes next to the perimeter (nodes 9 to 22) the response is cyclically symmetric which is what it should be since the dome has seven cyclically symmetric sectors and is under a uniform load. The nodes along the perimeter have no vertical deflection and all move the same radial distance (.584") outward.

The maximum stress values obtained for the glulam members (567.4 psi) are well within the linear range of stresses for timber.

The stress in the tension ring (21.7 ksi), on the other hand, is so close to the allowable stress in steel design (22.0 ksi) that the safety factor is almost equal to one.

The space frame model undergoes the greatest vertical deflection at the nodes near the apex, while the radial deflection in the horizontal plane is 0.184" at the supports.

The steel tension ring members experience the highest tensile stress found in the entire dome. The stress value, 15.9 ksi, is significantly lower than the allowable value in steel design.

Maximum values of stresses and displacements obtained for the frame model are shown in table 5.2.

A comparison between the result obtained for the space frame of this project and that of the frame run by Jassem Tissaoui, who used the B33 element of ABAQUS, has also been carried out. Both types of beam modelling show very close agreement in stress

TABLE 5.1. Results of linear truss model analysis.

Displacements			Stresses	
Apex (Vertical)	Maximum (Vertical)	Radial	Maximum Tensile	Maximum Compressive
1.387″	1.467″†	0.584″	-	567.4 psi
			21.7 ksi (Tension ring)	

†Nodes 2 - 9

TABLE 5.2. Results of linear frame model analysis.

Displacements			Stresses	
Apex (Vertical)	Maximum (Vertical)	Radial	Maximum Tensile	Maximum Compressive
1.285″	1.297″	0.184″	663.6† psi	1233.0‡ psi
			15.9 ksi (Tension ring)	

†Element #25 and cyclic equivalents: Main beams running parallel to permiter of dome.

‡Element #35 and cyclic equivalents.

values obtained for both linear and nonlinear analysis. The displacement values were close to each other.

Nonlinear Analysis

Attempts to get meaningful values of response using the linearized eigenvalue method of analysis for the space truss model were unsuccessful. Only large negative eigenvalues were obtained. Obtaining reasonable answers for structures whose joints are pinned is usually difficult when the linearized eigenvalue method is used in analysis.

Tracing the nonlinear equilibrium path of the space truss model was successful and the path of the apex node and that of a node in the inner ring are shown in Fig. 5.5.

Results of the nonlinear run for the frame model are shown in table 5.5. Tracing the equilibrium path is shown in figure 5.6.

The compressive stresses in the main beams are maximum at locations close to the perimeter of the dome and begin decreasing as one approaches the apex. The stresses obtained in the main beams of the frame model at failure are within the proportional limit for the type of wood being used since the proportional limit, 5896 psi, exceeds the maximum stress in either of the dome models which is 4743.0 psi compression.

Table 5.5 lists the allowable and ultimate stresses for longleaf pine.

Conclusion

The results show that the dominant mode of failure for both dome models is elastic instability. The assumption that geometric nonlinearities, and not material nonlinearities, govern instability of the dome models is justified.

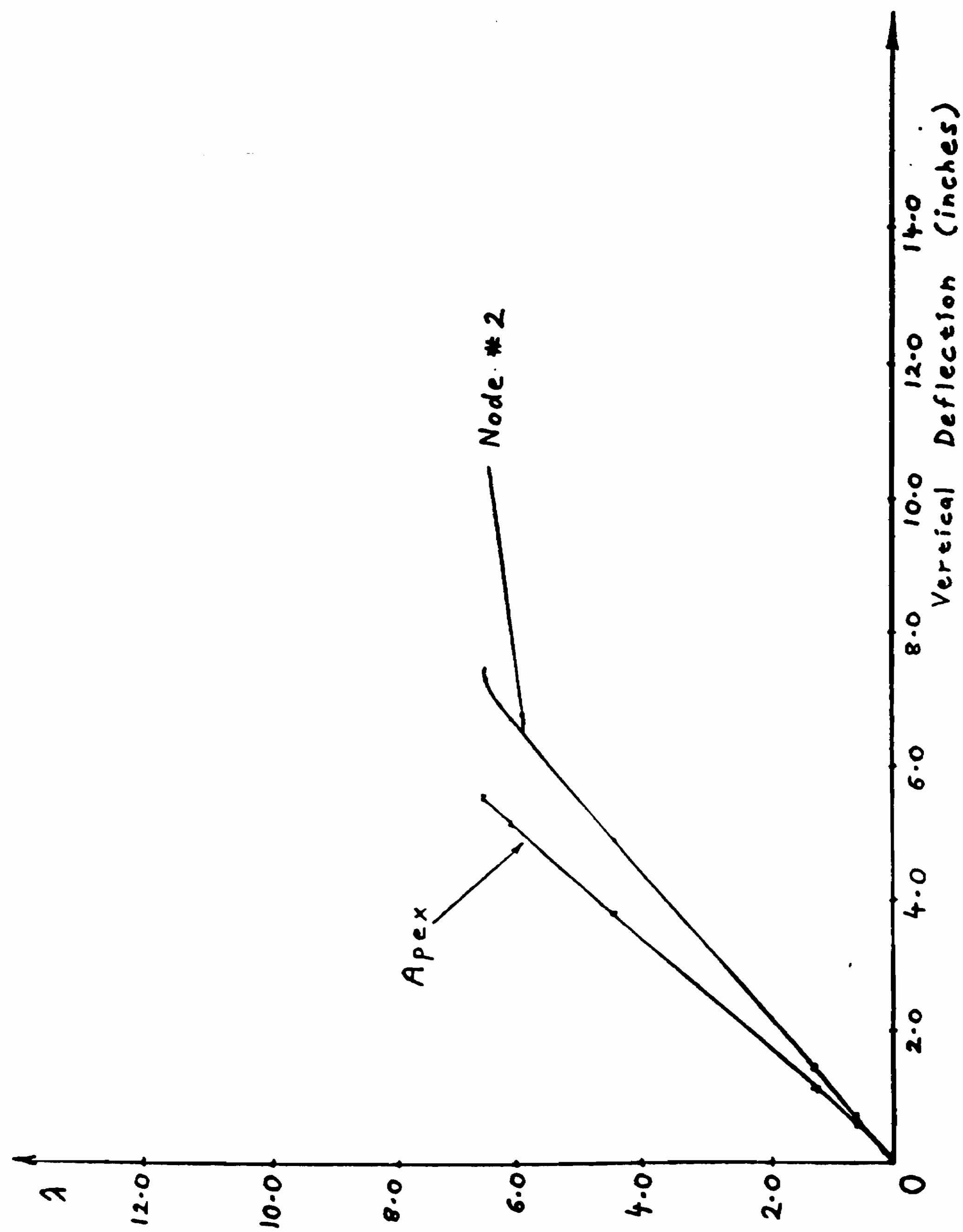


Figure 11. Fig. 5.5. Nonlinear equilibrium path of truss model.

TABLE 5.2. Results of nonlinear truss model analysis.

Displacements		
Apex (Vertical)	Maximum (Vertical)	Radial
5.43″	14.45″†	4.14″‡

†Node #14

‡Node #23

TABLE 5.4. Results of nonlinear frame model analysis.

Displacements			Stresses	
Apex (Vertical)	Maximum (Vertical)	Radial	Maximum Tensile	Maximum Compressive
3.861″	4.297″	1.311″	2610.0† psi	4748.0‡ psi
			48.7 ksi (Tension ring)	

†Element #25 and cyclic equivalents: Main beams running parallel to permiter of dome.

‡Element #35 and cyclic equivalents.

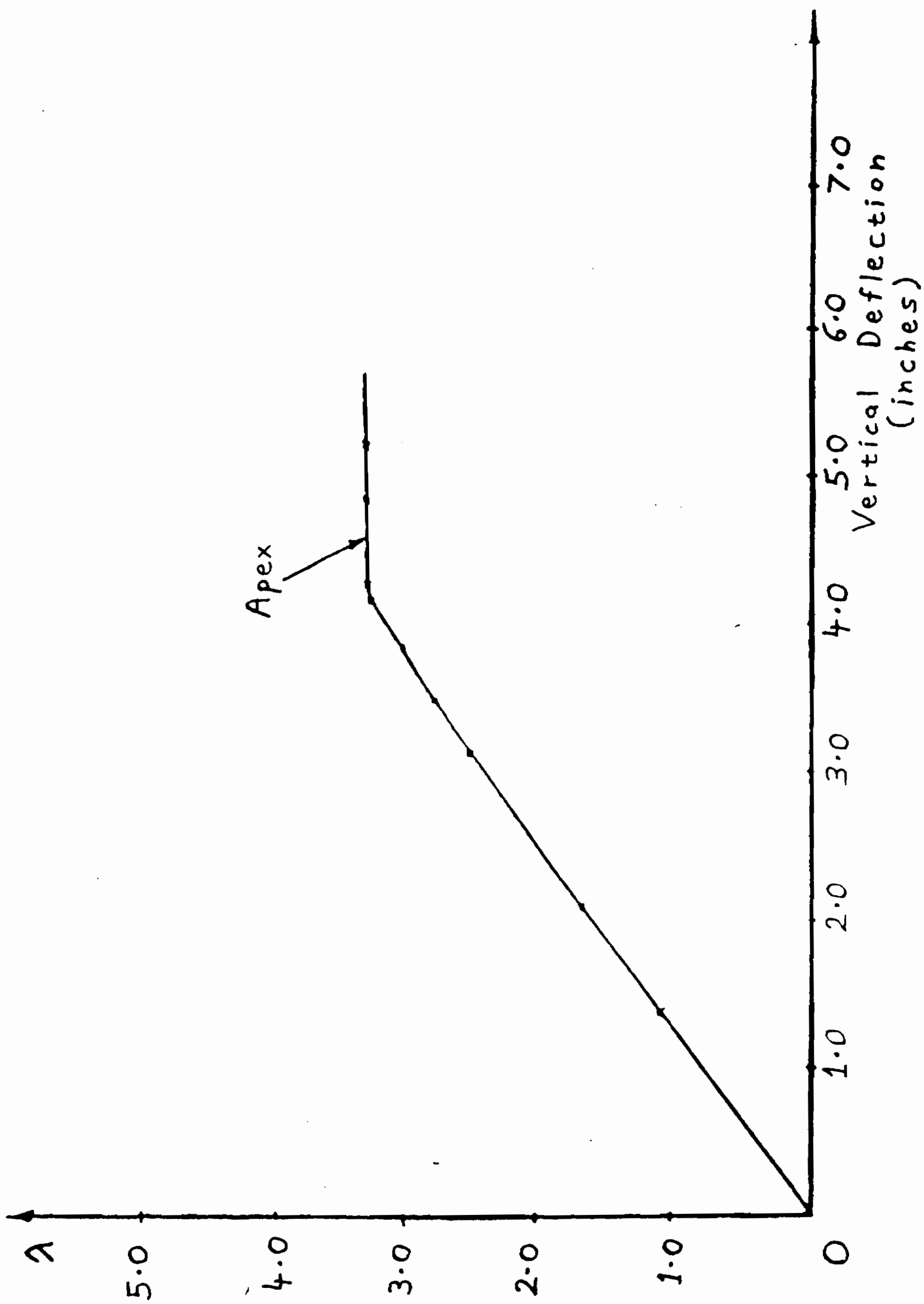


Figure 12. Fig. 5.6. Nonlinear equilibrium path of frame model.

TABLE 5.5. Parallel to grain stresses for longleaf pine (Wood Handbook, 1987).

Compression	Proportional limit:	5,896 psi
	Ultimate stress :	8,470 psi
Tension	Ultimate stress :	14,500 psi

The linear theory assumption of small displacements is justified because the nonlinear runs show linear load-deflection curves all the way up to the load level where instability occurs.

Bibliography

1. ABAQUS, Hibbit, Karlsson & Sorenson, Inc., 100 Medway Street, Providence, RI.

2. American Institute of Timber Construction, *Timber Construction Manual*, 3rd ed., John Wiley & Sons, Inc.,New York, 1985.

3. Bathe, K. J., *Finite Element Procedures in Engineering Analysis*, Prentice-Hall, Englewood Cliffs, NJ, 1982.

4. Bathe, K. J., "Finite Element Procedures for Solids and Structures - Nonlinear Analysis," Video Course Study Guide, *MIT Center for Advanced Engineering Study*, Reorder No. 73-2200, 1986.

5. Chajes, A., *Principles of Structural Stability Theory*, Prentice Hall Inc., Englewood Cliffs, N.J., 1974.

6. Cook, R. D., *Concepts and Applications of Finite Element Analysis*, 3rd ed., John Wiley and Sons, New York, 1989.

7. Davalos, J. F., "Geometrically Nonlinear Finite Element Analysis of a Glulam Timber Dome," Ph. D. Dissertation, Virginia Polytechnic Institute and State University, July 1989.

8. Holzer, S. M. and Huang, C. Y., "Stability Analysis by the Finite Element Method", Report No. CE/VPI-ST89/05, Virginia Polytechnic Institute and State University, Blacksburg, VA.

9. Holzer, S. M. and Loferski J. R., "Background for Research on Glulam Lattice Domes," *Proceedings of the Sixth Annual Congress*, ASCE Structural Division, Orlando, FL, August 1987, pp. 305-318.

10. Huang, C. Y., "Geometrically Nonlinear Finite Element Analysis of a Lattice Dome", M. S. Thesis, Virginia Polytechnic Institute and State University, November 1989.

11. Langhaar, H. L., *Energy Methods in Applied Mechanics*, Wiley, New York, 1962.

12. Thompson, J. M. T. and Hunt, G. W., *A General Theory of Elastic Stability* , John Wiley and Sons Ltd., London, 1973.

13. Timoshenko, S. P. and Gere, J. M., *Theory of Elastic Stability*, 2nd Edition, McGraw Hill, New York, 1961.

14. United States Department of Agriculture, 1987, *Wood Handbook: Wood as an Engineering Material*, U. S. Forest Products Laboratory, Agriculture Handbook No. 72, U. S. Government Printing Office, Washington. D.C.

APPENDIX A

Appendix A. WRITING AND SUBMITTING PROBLEMS IN ABAQUS

Overview

To run a problem on ABAQUS, one has to submit an input file which gives the geometry, boundary and loading conditions of the problem being studied.

The input file should consist of command names which define the geometry (i. e. node locations), member connectivity, cross section and material properties of all the elements. In addition the boundary conditions are specified. The loading is then specified. For nonlinear problems the starting increment, force tolerances, time period, minimum time step and number of increments have to be decided and placed in the input file. Options to determine the point at which analysis stops include setting maximum values of displacement and/or load magnification which if reached lead to the analysis being terminated.

Setting up an input file

Storage

The command *HEADING is placed on the top of the input file, followed by one or more lines on which any comments can be written.

Specifying computer memory can be done either through TDISKS or virtual memory. To specify virtual memory the first line of the input data deck is recommended. For the nonlinear analysis of the space truss

*HEADING, CORE = 900000

was used.

Geometry, Connectivity and Material Properties

The command *NODE is placed next. This precedes the data giving the node numbers and their locations. The locations can be given in Cartesian, polar or spherical coordinates.

An example of a node which describes the kind of coordinate system is:

* NODE, NSET = MAIN, SYSTEM = SPHERICAL

1, 1.2720E+03, 3.4714E+02, 5.2178E+01

If the coordinate system had not been specified, the computer would have interpreted the data as default (Cartesian) coordinates. For spherical coordinates it would have been sufficient to give the first letter S. For rectangular or polar coordinates it would have been sufficient to give the letter R or P, respectively. The data lines which follow the node heading give the node number and coordinates for each node. The coordinate order is X, Y and Z for Cartesian (rectangular), R, θ, ϕ for spherical and r, θ, z for polar in that order. NSET = name defines a group of nodes as a single set having the assigned name.

The command *TRANSFORM is used to define new coordinate axes within a system or to allow only radial deflections of some nodes to occur. This is done by defining the center and radius of a cylinder as follows:

```
* TRANSFORM, NSET = TR 1, TYPE = C
0.000E+00, 0.000E+00, 0.000E+00, 0.000E+00, 0.000E+00, 1.000E+00
```

This indicates that a previous set of nodes having the name TR 1 is allowed to move radially.

The designation *ELEMENT defines elements by giving their nodes. Another option is *ELGEN, which defines elements using incremental generation.

An example of member connectivity and type is provided by the following command:

```
*ELEMENT, TYPE = B32, ELSET = E 25
26    38  39  40
```

This tells the computer that element number 26 is made out of three nodes which are numbered 38, 39 and 40 with node 39 being the central node. TYPE = B32 indicates that the member is a beam which is composed of three nodes and uses linear interpolation. It is necessary when beams have different orientations to give each beam its own ELSET. E 25 is the name of the element set to which the direction cosines of beam element number 26 have to be assigned. For the beam mentioned above,

```
*BEAM SECTION, SECTION=RECT, MATERIAL=WOOD, ELSET= E 25
5.125, 12.0
-0.4436  0.8756  -0.1913
```

is a necessary item of information. This lets the computer know that the beam has a rectangular section which is 5.125 by 12.0 and whose direction cosines of the 1-axis with the global X, Y and Z (or 1, 2, and 3) axes are -0.4436, 0.8756, -0.1913, respectively. All elements in ELEST E 25 will then have the same 1-axis and the same dimensions. The moment of inertia calculated by the computer will then be assigned to this element set.

When bending is not taking place around any of the principal axes of a flexural member, then the only way for the computer to take this into account is to assign a separate ELSET to each element and input the direction cosines of the 1-axis of each beam into its corresponding element set.

If any other way is used to define the direction cosines, the computer will ignore the values input and the default value for the cosines of 0.000, 0.000, -1.000 will be used. For example, putting direction cosines together with the node coordinates is ineffective. The computer ignores these during the analysis as can be seen by inspecting the PREOUT file. Inspection will show that the orientation of the beam sections will be taken by the computer as the default values as far as inertia properties are concerned. These default values mean that the computer will solve the problem by considering that bending is taking place around one of the principal axes. As a result the computed answers will turn out to be incorrect.

The only way to get the computer to accept the values for beam orientations other than the default value is to define an ELSET for each beam which has the geometric dimensions and orientation of the cross section of that beam.

A unique element set and section card is therefore provided for each member. The output from the Fortran program written by me is entered into the input data deck.

STRI3 and S4R elements need to have the node numbers of the nodes making up each element put in the member connectivity data input. If all elements have the same material properties and thickness, then all the elements can be listed in one ELSET. The important thing is to ensure that each node at the corner of a shell element is found to coincide either with the end or middle node of a flexural element.

For truss and shell elements the geometric and connectivity specifications are as follows:

```
*SOLID SECTION, ELSET=TRUSS, MATERIAL=STEEL
 5.0
*ELEMENT, TYPE=C1D2, ELSET=TRUSS
 262   2  84
```

for a truss and

```
*ELEMENT, TYPE=STRI35, ELSET=FILM
 245   72  71  117
*SHELL SECTION, ELSET=FILM, MATERIAL=STEEL
 1.000E-03  3
```

for a shell element. These specifications indicate that the truss element #262 has nodes 2 and 84 as its end nodes, that its cross-sectional area is 5.0 square units, and that the shell element #245 has the 3 specified nodes at its corners and a thickness of 1/1000 of a unit length.

To introduce the material properties, the specifications

```
*MATERIAL, NAME = STEEL
*ELASTIC
2.900E+07, 0.3
*DENSITY
0.284
```

for truss and shell elements and

```
*MATERIAL, NAME = WOOD
```

```
*ELASTIC
1.80E+06, 4.625
*DENSITY
0.022
```

for beams and purlins are used.

The numbers following the *ELASTIC designation are the values of the modulus of elasticity and the Poisson's ratio, respectively. The density of the material also needs to be specified for dynamic loading situations such as wind loading.

Boundary Conditions

The boundary conditions are denoted by the command *BOUNDARY. Following this, the nodes to which the boundary command applies are listed. Each line begins with the node number being restricted followed by the global direction(s) in which movement is prevented. Examples are :

```
*BOUNDARY
  6, 1, 2
  8, 1
  8, 3
*BOUNDARY
 604, 2, ,  0.00000E+00
 604, 2, ,  0.00000E+00
```

Analysis

Linear Analysis

Linear analysis is prompted either by default or by specifying the LINEAR parameter on the same line after the *STEP option. One way of specifying the analysis procedure is as follows:

```
* STEP, LINEAR=NEW
* STATIC
*DLOAD, OP=NEW
FILM, BZ = -278.00
```

The *STEP option indicates a loading stage and the analysis procedure is specified by the *STATIC option. The parameter LINEAR indicates that linear analysis only should be used in the step. This parameter has two possible values: OLD and NEW. The value LINEAR = NEW indicates that the stiffness matrix which is formed at the first increment can be used throughout the whole analysis. This value should be used for the first step. LINEAR = OLD indicates that the stiffness matrix of a previous step can be used in the current step.

The option *DLOAD provides input of distributed loads. The parameter OP can be set to either NEW or MOD. For OP = NEW all distributed loads defined in the step will be removed. For OP = MOD the previous loads will only be modified by the new input.

The uniform loading on the dome is defined by

```
FILM, BZ = -278.00
```

indicating that the STI3 elements, having FILM as the name of their element set, are subjected to a vertical pressure of 278.00 units in the downward (Z) direction.

Buckling Analysis

A 2 - stage process is used in buckling analysis.

In the first stage after the "dead" loading is applied, the stiffness matrix is stored by using:

```
* STEP
* BUCKLE, DEAD
* END STEP
```

After this the " live " loading is applied as a small fraction of the expected critical load. Usually "live loading" is applied in a single step, although it can be done over several steps.

In the second stage the difference in stiffness is calculated and the eigenvalue problem is solved by using the commands:

```
* STEP
* BUCKLE, LIVE
  1
* END STEP
```

The number 1 indicates that the first buckling load estimate is desired.

Nonlinear Analysis

It may not be possible or even desirable to obtain a solution immediately for a nonlinear problem by imposing the entire load at the beginning of nonlinear analysis. The load often has to be applied gradually in order to obtain a correct solution. In order to apply the load gradually, the "step" is divided into increments.

For problems involving geometrically nonlinear analysis the parameter NLGEOM is specified on the same line after the *STEP option. The NLGEOM parameter signals that a large displacement analysis is to be used. This command is put in the *STEP option and a force tolerance, PTOL, must be provided. A moment tolerance, MTOL, also may be specified for structural elements having rotational degrees of freedom.

These tolerances define the accuracy with which equilibrium must be satisfied at every node. A compromise has to be made between setting very large and very loose error tolerances. Very tight tolerances lead to more iterations and more time and expense spent on problem solving. Loose tolerances may lead to bad results in which the structure is not in equilibrium. A general guideline for choosing tolerances is that PTOL is taken as 0.1% - 1% of typical element forces and MTOL is taken as PTOL multiplied by a typical element length (ABAQUS, 1989).

An example incorporating all of the above is:

```
* STEP, NLGEOM, INC=40
* STATIC, PTOL= 1.0, MTOL=240., RIKS
0.20, 1., 0.01, 0.2, 76., 3, 3, -2.50
*DLOAD, OP=NEW
FILM, BZ = -278.00
```

The above example specifies a maximum of 40 load increments as well as the type of incrementation being used.

For controlling the solution process, two options are available in ABAQUS. The first is the automatic incrementation option. This is based on tolerances set by the user and is recommended for most cases. The second option is to have the user specify the way the load is incremented. This is recommended when automatic control option encounters difficulty in converging.

In nonlinear static analysis the method of automatic incrementation is usually preferred since Newton's method (which has a finite radius of convergence) can overshoot.

In the above example the RIKS option for automatic incrementation was chosen. If direct user control of incrementation during the step were desired, the DIRECT parameter would be specified instead.

The first four numbers on the data deck are the initial time increment, the time period of the time step, the minimum time increment, and the maximum time increment specification in that order. Specifying a value for the minimum time increment prevents the use of excessive computer time. Specifying a maximum time increment ensures that enough solution points are available so that the nonlinear response path can be plotted. The last four numbers are the load magnification factor, the node number having a limit to its displacement, the direction in which the displacement is limited and the displacement value which if exceeded leads to the termination of the analysis. The load magnification values and the displacement values are optional specifications. If neither are specified the only criterion for termination is the number of increments. If both specifications are present then the analysis ends as soon as one of the three conditions is met.

Output Control

In running ABAQUS the user, guided by menus, can choose which class to allocate the analysis to, what time to begin the analysis and/or which time the analysis should be terminated.

An example of requested output is given by

*EL PRINT, SUMMARY = NO

```
*NODE PRINT, NSET = MAIN, GLOBAL = YES
  U, RF
  CF
```

where U denotes all displacement and rotation components and RF and CF denote all reactive and concentrated forces, respectively.

For output control the commands *NSET and ELSET are used to find displacements, stresses, strains etc. of various nodes and elements.

Submitting a job in ABAQUS

In order to submit a problem "ABAQUS" has to be typed before the filename or in the command area of the filelist. A menu appears showing which version of the ABAQUS program is being used. The filename of the problem is entered and the RETURN key is pressed. A prompt appears asking for the name of a RESTART file. If the solution of the problem involves any sort of linear analysis following a nonlinear analysis this option should be used. In this case the filename of a previous file analysed by ABAQUS is entered. After pressing the RETURN key several times a menu appears for choosing the name of the output file. The next prompt to appear is 'CPU TIME FOR THIS JOB IN SECONDS'. If the number of seconds required exceeds 7200 seconds one enters the number of seconds and hits RETURN. Following this a prompt appears asking how much T disk space is needed to run the problem. The default will be 20 cylinders. For small problems, having fewer than 100 degrees of freedom, 3 - 20 cylinders provide enough space allocation for running the problem. For larger problems having thousands of degrees of freedom, cylinder space required is usually in the 100 - 200 range. Sometimes more cylinder space is needed in order to run large complicated nonlinear problems. If the number of cylinders exceeds 20, the number is entered and the

RETURN key is pressed. A menu appears providing several options. These include choosing the time when the analysis begins, when it ends, the job class, and how many seconds are needed for the analysis.

The class of the job being submitted can be changed if more time is needed for running the problem. The default is class B which allocates a time of 600 seconds. To run the nonlinear problem for the space and frame models, class Y problems can be used which has a time allocation of 2400 seconds.

For very large problems one can submit jobs having a C, D or Z class designation through the BATCH machine. These classes allow for even greater memory allocation. The nonlinear analysis for the space frame was assigned to class C. CPU time is 7200 seconds on the ABAQUS time scale.

The final inquiry by ABAQUS will be whether another job (problem) is to be submitted. If no other analysis is required the default command is given by hitting the return key.

A message will then appear that the job with the filename has been submitted. While the program is running, the batch status can be checked by pressing the PF4 key. Once the analysis has been completed, four files are obtained as an output. If the D option has been specified they will all have the same filename as the input file.

If the input file has no errors and the time allocated for the analysis turns out to be sufficient, then four files will be obtained as an output. The file having filetype MAINOUT will contain the analysis results; a second file of filetype PREOUT will contain a reproduction of the input data file. In case the input data file contains any syntax or other errors, the PREOUT file will contain messages indicating where and what type of errors occur. A third file will be of filetype RES. This will be of use if the RESTART option needs to be used. The fourth file (if requested) will be of type SUP.

Vita

Kadmous El-Adas

Date of Birth: 6th Nov. 1962

Education

M.E., 1990

Civil Engineering

Virginia Polytechnic Institute and State University, USA.

Project: Linear and nonlinear buckling analysis via ABAQUS.

B.E., 1987

Civil Engineering

American University of Beirut, Lebanon.

Major: Civil Engineering