ANALYSIS OF MULTIPLE SHELL STRUCTURES SUBJECTED TO LATERAL LOADS

횡력을 받는 합성 쉘 구조의 해석§

이 평 수*
Lee, Pyong Soo

요 약

2차원인 유한요소들을 각 절점에서 6개의 자유도를 갖는 3차원인 입체로 결합함으로서, 횡하중을 받고있는 합성 쉘 구조를 해석할 수 있는 프로그램[MSSLL]을 개발하였다. 전체 구조물이 여러개의 반복되는 Substructure들로 이루어졌을 때에는, 인력의 소모를 극소화 하고 계산시간을 절약할 수 있도록 해석과정에 Substructuring 기법을 본 프로그램에 도입하였다. 프로그램의 신뢰도를 확인하기 위하여 본 프로그램에 의한 해석결과와 다른 방법에 의한 결과를 비교분석 하였으며, 지진력을 받고있는 8개의 개별 원추들로 구성된 쉘 구조의 거동에 대한 높이－경간비의 영향을 규명하기 위하여 변수연구를 수행하였다.

Abstract

A computer program, MSSLL, was developed for the analysis of the "Multiple Shell Subjected to Lateral Loads" by utilizing 2-dimensional finite elements in a 3-dimensional global assemblage with 6 DOF at each nodal point. In this program, substructuring procedure with frontal solver was introduced in the solution procedure to save both human and computer resources when the whole structure consists of repeated identical substructures. Some of the results obtained by MSSLL were compared with the existing solutions by other methods. The effect of rise to span-length ratio was investigated for the behavior of the multiple conical shell with 8 substructures subjected to seismic loads by performing a parametric study.

1. INTRODUCTION

The multiple shell structure which may consist of several repeated shell sectors or different type of shell sectors is widely used as roofs for industrial buildings, gymnasiums, grandstands, exhibition

* 정회원, 육군사관학교 교수, 공학박사
§ 본 연구는 한국과학재단의 연구비 지원에 의한 것임

□ 이 논문에 대한 토론을 1989년 9월30일까지 본학회에 보내주시면 1990년 3월호에 그 결과를 게재하겠습니다.

halls, museums, ect., and as main structures in the military industries such as airplanes, shipbuildings or other weapon systems, since it is esthetic in appearance, effective in structural behavior due to its geometry, and it covers large areas without intermediate supports.

The classical thin shell theory yields differential equations of equilibrium or continuity whose complexity depends greatly on the shell geometry and whose solution is a function of the geometric position of the boundary and the type of forces or displacement quantities which must be satisfied there. Hence, it is almost impossible to analyze the multiple shell structures by classical methods in case of complicated geometry and boundary conditions together with variable thickness or material properties, discontinuities on the shell surface, and general loading conditions. Thus a finite element method is employed to analyze those structures in this study.

Many investigators analyzed the multiple cylindrical and hyperbolic paraboloid shells by finite element method [1, 2, 3 etc.]. Few topics [4, 5, 6, 7] can be found for the analysis of the multiple conical shells up to present. Furthermore, they were carried out either for the specific shells under simple loading condition or by applying the membrane theory only.

The objective of this study is to develop a computer program which can analyze the multiple shell structures under general loading conditions, to provide the input guide which can simplify the large input data of physical problems, and to perform the parametric study of multiple conical shell subjected to lateral loads since this shell type has not been dealt with profoundly in the literature.

2. FINITE ELEMENT IDEALIZATION

2.1 General Procedure in Finite Element Method

The finite element method is a general technique for constructing approximate solutions and its basic concept is the idealization of the continuum as an assemblage of discrete structural elements. The continuum is first divided into a finite number of elements which are interconnected at a discrete number of nodal points situated on their boundaries. Then the stiffness properties of each element are evaluated and the global stiffness of the complete structure is obtained by superposition of the individual element stiffness corresponding to the degree of freedom at nodal points. This gives a system of linear equations relating the nodal point loads and displacements whose solution yields the unknown nodal point displacements. The stress resultants of each element are obtained through the stress recovery procedure.

2.2 Types of Finite Element

Flat plate elements were introduced for the idealization of thin shells in the early 1960s, doubly curved elements were introuduced a few years later as an attempt to find more appropriate elements for shell geometry, and three-dimensional isoparametric solid elements followed. These different types of elements have their own advantages and disadvantages. The higher order elements may lead to more accurate results but greater complexity and more computer time in application, while the simple flat plate elements with coarse mesh may lead to undesired final results. The flat plate elements were chosen in the present study. This simplifies the evaluation of the element stiffness properties, but results in some geometric discretization error since the behavior of the discretized shell can only approach that of the actual shell with decreasing mesh sizes.

There are three membrane elements and one plate bending element available in the program used in this study. The three membrane elements are constant strain triangular (CST) [8], constrained linear strain triangular (CLST) [9], and refined quadrilateral membrane (QM5) [10] elements. The stiffnesses of these elements are derived based on the assumption of plane stress.

The CST – triangle is recommended only when it is required for the geometric idealization of the shell structure, since this element is inferior to other membrane elements in the convergence of the solution. In general, the QM5 quadriliateral element is preferable to the CLST quadrilateral except when the geometry of the element is skewed [11]. The HCT [12] element used as the plate bending element has 3 DOF at each nodal point. This element is fully compatible for plate bending problems since the normal slopes along the element edges are constrained to be linear.

The complete flat triangular element for both membrane and bending stiffnesses is achieved by adding CST element to HCT element as shown in Fig. 1(a) and so this element has 5 DOF with a zero rotational stiffness about the axis normal to the element at each nodal point. Two versions of the quadrilateral element are available in the program as indicated in Fig. 1(b) and (c). The quadrilateral element in Fig. 1(b) is obtained by combining the assemblage of 4 CLST elements with that of 4 HCT elements for the nonplanar elemtnt, while the element in Fig. 1(c) consists

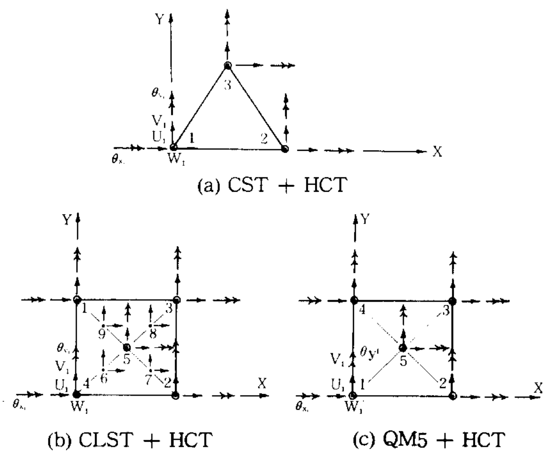


Fig. 1 Combination of membrane element with bending element

the assemblage of QM5 element with 4 HCT elements for planar element. Both of the above quadrilateral elements have also 5 DOF at four corners and the interior DOF of the coentral and mid – side nodes are eliminated by static condensation.

2.3 Out of Plane Rotational Stiffness

The elements previously described have only 5 DOF at each node on the element level and these 5 DOF are also maintained in describing the stiffness of the element assemblage in the present program. In order to include, in a systematic way, the 6 DOF for the stiffness of the element assemblage, Zienkiewicz [13] and Johnson [14] suggested the ficititious rotational coefficients for the triangular element and the quadrilateral element, respectively.

These rotational stiffness coefficients were constructed such that equilibrium is preserved at the element level, and Zienkiewicz pointed out that the fictitious stiffness will affect the response of structure. In general, this fictitious stiffness will stiffen the structure if all the elements are not co – planar.

3. SUBSTRUCTURING TECHNIQUE

3.1 Substructuring Concept

The static analysis of a structure requires to solve a set of linear equilibrium equations given by

$$K_r = R \qquad (1)$$

where K is the stiffness matrix of the whole structure, r is the unknown displacement vector, and R is the corresponding load vector.

In the finite element procedure of the analysis of multiple shell structures, great efforts are frequently required for input preparati on as well as computer time and memory storage. These may be overcome by employing the substructuring technique [15] in the solution procedure.

The basic concept of substructuring consists of dividing the whole structure into several substructures and of integrating them after proper modification in each substructure. In this procedure, the order of the finite element model is reduced by a systematic reduction of the repeated substructures whose properties are identical in sequence. Thus it saves the reduction and modeling efforts and the reduced model can be efficiently solved by the standard solution procedure.

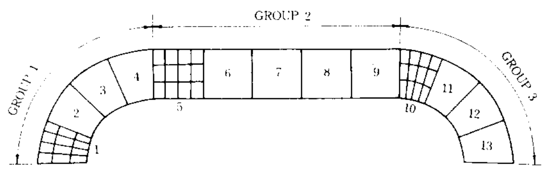
The effort required to solve Eq.(1) depends on the half bandwidth which is determined by the node numbering scheme. The node numbering scheme of minimizing the half bandwidth is very time consuming in the modeling of the complex and large structure. However, in the substructuring procedure, the individual substructures can have their own optimal numbering scheme to reduce the half bandwidth, and also they require smaller number of operation in the elimination of the interior DOF. The interior DOF of each substructure can be eliminated since the stiffness matrix and the applied load vector are uncoupled with respect to the unknown displacements. Consequently, the solution of the original system can be obtained from the solution of the reduced equations defined in terms of the exterior DOF at the juncture of individual substructures.

3.2 Frontal Method

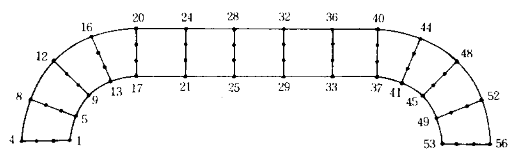
In the substructuring procedure to reduce the order of the finite element model by eliminating the interior DOF, the frontal technique [16, 17] is very effective since the front itself constitute a substructure stiffness for the elements which have been processed. The frontal process alternates between accumulation and elimination of element coefficients until the desired form of reduced equations is obtained from the original equations.

In this process, the front width is determined by the connectivity of the current element to the other elements assembled in the front. Thus the element numbering should run in the direction with the smallest number of elements in order to reduce the front width. However, the node numbering scheme is immaterial, which in contrast is very important in the banded solution procedure. This characteristic of the frontal procedure may help the remodeling with little modifications when the local refinement of meshes is necessary in the original coarse mesh layout.

Only the first substructure of Fig. 2(a) needs to be generated in each group and the remaining substructures in each group need not be generated since they are repeated identical substructures. However, if the load vectors in the individual substructures are different, they should be generated corresponding to the process of the stiffness. The final structure after the elimination of the interior DOF is shown in Fig. 2(b). The solution of equilibrium equations can be obtained by employing the standard solution techniques for this reduced structure. However, the solution also can be obtained by applying the same frontal procedure for each substructure to this final structure as if the substructures in this structure were large elements having a very large number of DOF.



(a) Mesh layouls of first substructure in each group



(b) Master nodes after elimination of interior DOF

Fig. 2 Multiple shell structure with 13 substructures

4. DOCUMENTATION OF PROGRAM MSSLL

4.1 Program Information

A computer program MSSLL (Multiple Shell Subjected to Lateral Loads) has been developed for this research. This program utilizes 2–dimensional finite elements in a 3–dimensional global assemblage with six DOF at each nodal point and provides a general capability for the static analysis of prismatic, cylindrical conical, hyperbolic paraboloid, and other shells of arbitrary geometry with arbitrary loads and boundary conditions.

The subject computer program has automatic mesh generation options and recognizes repeated substructures in both the stiffness and the stress recovery in order to simplify and reduce the efforts for the input data preparation and to save computer time and memory requirements. This program consists of three subprograms called SHELLG, SHELLP, and SHELLR which have to be executed in sequence.

The subprogram SHELLG is mainly for the generation of nodal point's coordinates, meshes, and loads and for the calculation of stiffnesses. Each element stiffness is calculated and tapes are created in this subprogram for the subsequent execution of SHELLP and SHELLR. The second subprogram SHELLP, a single level substructuring package, is the principal program which has 50 subroutines. This program needs input data for substructure information and boundary conditions and tapes from SHELLG for execution, and calculates the nodal point displacements and saves them on tapes for SHELLR. The third subprogram SHELLR is the stress recovery program which also requires some input data and tatpes generated by former two subprograms. It gives stresses, stress resultants of each element, or stress resultants at each nodal point by user's options.

4.2 Coordinate Systems

Two types of base coordinate systems are used in this analysis, They are the global coordinate system (X, Y, Z) which is a fixed set of Cartesian coordinates and the surface coordinate system (ξ_1, ξ_2, ξ_3)which is necessary to specify all translations and rotations in the desired directions.

There are two local coordinate systems ; element coordinates ($\bar{x}$, $\bar{y}$, $\bar{z}$) for triangular elements and η–coordinates (η_1, η_2, η_3) for quadrilateral elements. They are shown in Fig. 3 and constructed automatically by the program. The stresses and stress resultants of each element are oriented to these local coordinate systems in the computer output.

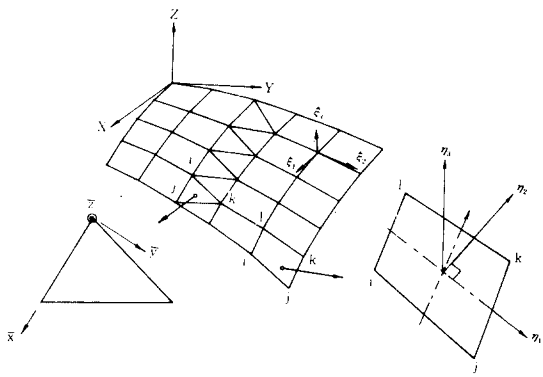


Fig. 3 Discretized shell with coordinate systems

The coordinate systems for different shell geometries are appropriately established in order to generate the input data of nodal point coordinates, and the components of surface coordinates in the direction of global coordinate are also prepared to apply the continuity and boundary conditions. However, for simplicity, only the coordinate system of the conical shell is presented here as an illustration.

The global coordinate system is shown in Fig.4 with the origin at the vertex of the conical shell. The local coordinate system is oriented in such a way that, with the same origin as the global coordinate system, the y-axis is the axis of the cone itself and the x-axis lies on the half section of the full cone whose plane may be inclined against the horizontal plane at an angle β. The horizontal projection of the y-axis (y_H) may have an angle α_y from the global Y-axis.

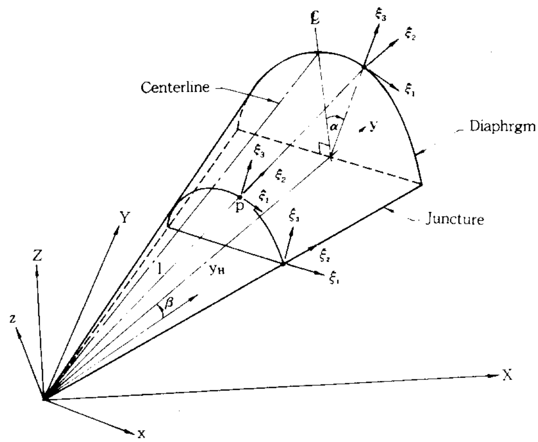


Fig. 4 Coordinate system for conical shell

Three independent surface coordinate systems are selected for the conical shell as shown in Fig. 4 in order to specify the boundary conditions at different locations. The ξ coordinates are constructed on the usual surface, the $\bar{\xi}$ coordinates are used at the juncture of the shell sectors, and the $\bar{\bar{\xi}}$ coordinates are defined at the outer edge of the shell sector where the diaphragm is provided. The positive directions of these coordinates are also given in the same figure. The global components of the surface coordinates can be obtained by considering the geometry of the conica l shell [7].

4.3 Load Vector

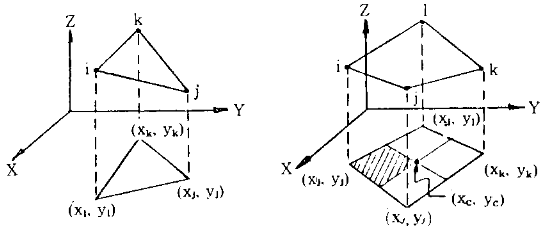
The dead weight of each element is automatically treated ty the consistent load procedure for nodal point loads in the program. These nodal point loads are first established in the element coordinates and then they must be transformed into the base coordinates for load assemblage.

When the shell is subjected to uniformly distributed live loads in arbitrary directions, the global coordinate components of the loads and the projected area of the element in the directions of the global coordinates are first calculated, then the nodal point loads are determined by considering the tributary areas of this projected element with the corresponding load components. In the triangular elements, the nodal point load at each node is a third of the total element loads. As an example, the Z-directional load at a node of the triangle shown in Fig. 5(a) is, for the uniformly distributed live load intensity of p_Z on the horizontal projection;

$$P_Z = p_Z\ [(x_j - x_i)\ (y_k - y_i) - (x_i - x_k)\ (y_i - y_j)]\ /6 \qquad (2)$$

The nodal point load of the quadrilateral element, for example, P_Z at node i can be obtained as follows: The tributary area for node i is the shaded area in Fig. 5(b) which is a part of the horizontally projected area and is determined by adding the half area of the triangle $i-j-c$ to that of the triangle $i-c-\ell$, thus

$$A_i = [\ [(x_c - x_i)\ (y_\ell - y_j) + (y_c - y_i)\ (x_j - x_\ell)\ ]/4 \qquad (3)$$



(a) Triangular element (b) Quadrilateral element

Fig. 5 Projected area on $x-y$ plane

The tributary areas for the remaining nodes j,

k, and ℓ are similary obtained, then the Z-directional loads at these four nodes are obtained by multiplying the load intensity of p_Z to the corresponding tributary areas such as

$$P_i = p_Z\,[\,[(x_c-x_i)\,(y_\ell-y_j) + (y_c-y_i)\,(x_j-x_\ell)]/4$$
$$P_j = p_Z\,[\,[(x_c-x_j)\,(y_i-y_k) + (y_c-y_j)\,(x_k-x_i)\,]/4$$
$$P_k = p_Z\,[\,[(x_c-x_k)\,(y_j-y_\ell)+(y_c-y_k)\,(x_\ell-x_j)\,]/4$$
$$P_\ell = p_Z\,[\,[(x_c-x_\ell)\,(y_k-y_i)+(y_c-y_\ell)\,(x_i-x_k)\,]/4 \quad (4)$$

Where x_c and y_c are the average coordinates of four nodes.

The nodal point loads for other loading conditions may be obtained by calculating their components according to the DOF at each node. For example, the components of loads applied to the conical shell can be obtained by the formular given in Ref. [18]. These load components together with those obtained from uniform live loads are simply added to the corresponding dead load components in the global coordinate system.

5. DEMONSTRATION ANALYSES

5.1 Cylindrical Shell subjected to Wind Load

The geometry and material properties for a cylindrical shell which consists of two barrels are shown in Fig. 6. This shell is supported by diaphragms at both ends, connected monolithically along the central edge, and free along the exterior edges. This shell is loaded by its own dead weight and wind pressure from left to right as shown in the same figure.

Only a half of the structure which is divided into two substructures needs to be considered in the analysis because of its symmetry about min-span Typical uniform meshes (8x4) of quadrilaterals are shown on the second substruture of Fig.6.

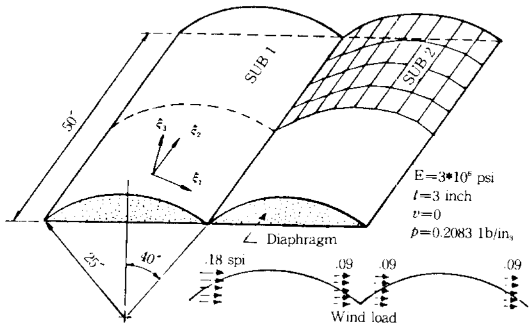
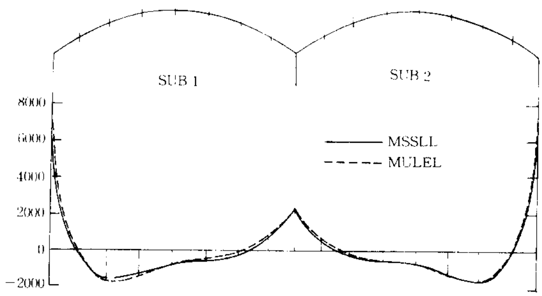
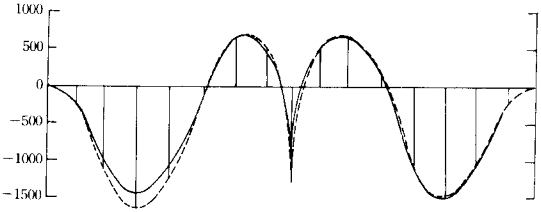


Fig. 6 Geometry and material properties of cylindrical shell with loading condition

The displacements and the stress resultants of this shell are obtained from the present program MSSLL and compared with those obtained by the execution of program MULEL [19] which is considered as "exact" solutions since it utilizes the Donñel-Jenkin's shell equations. The dominant



(a) Longitudianl forces at midspan

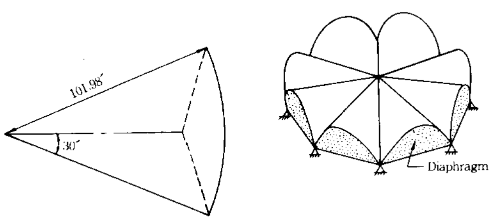


(b) Transverse moments at midspan

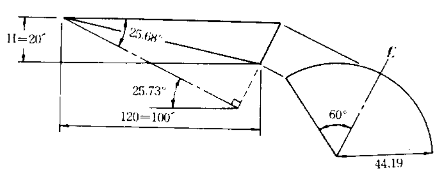
Fig. 7 Stress resultants for two barrel cylindrical shell stress resultants of the multiple cylindrical shell are longitudinal forces and transverse moments at midspan and they are plotted in Fig. 7(a) and (b), respectively, in which they show a good agreement on most regions for the solutions obtained from MSSLL and MULEL. Large deviations occur at free edges for the longitudinal forces and at crown and at joint for the transverse moments, respectively. These deviations may be due to the truncation error in the series solution of MULEL or the discretized error in the finite element model, and can be reduced by a local refinement of meshes in those regions in the modeling.

5.2 Conical Shell Subjected To Seismic Load

A typical roof structure for gymnasiums is selected from reference [7] to compare the results for the dead load case and to investigate the behavior of multiple conical shells under the lateral loads. This shell having a circular shape in plan consists of eight identical substructures as shown in Fig. 8(a) and the geometry of a conical shell is shown in Fig. 8(b). This structure is supported along the circumference by diaphragms perpendicular to the individual cone axes and by hinges at the joints



(a) Overal view



(b) Geometry of one substructure

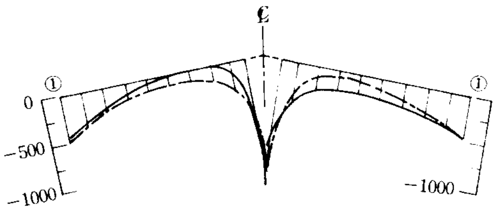
Fig. 8 Multiple conical shell having circular shape in plan

of the diaphragms. The material of this shell is assumed as reinforced concrete (E_c=4,287,000 psi, ν=o.167) and the thickness is 2 inches except for the center portion. It is naturally or intentionally thickened in the practical construction around the top of the roof where several apexes of individual conical shell meet and a special consideration is needed in the finite element modeling. Each substructure is divided into 52 quadrilateral and 12 triangular elements in the modeling.

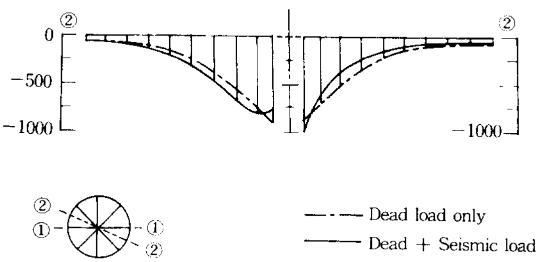
This multiple conical shell is subjected to its own dead weight and uniform seismic-type loading of p_s=0.25 pai on vertical projection of the shell surface perpendicular to load direction, where the vertical seismic acceleration is neglected and the horizontal seismic acceleration is assumed as a= 0.4 g where g is the gravitational acceleration.

Since the structure is symmetrical with respect to the center line of the load direction, only a half structure which has four substructures needs to be analyzed. The boundary conditions for this case are such that $\delta_{\bar{\xi}_1}=\theta_{\bar{\xi}_2}=\theta_{\bar{\xi}_3}=0$ at nodes on symmetric axis of whole structure, $\delta_{\xi_1}=\delta_{\xi_3}=\theta_{\xi_2}=0$ at nodes on diaphragms, $\delta_{\bar{\xi}_1}=\delta_{\bar{\xi}_2}=\delta_{\bar{\xi}_3}=0$ at the joint nodes of the diaphragms, where δ and θ represent the translations and rotations respectively and the subscripts are surface coordinates defined as in Fig. 4.

The displacements and the stress resultants obtained from the present program are exactly the same as those of Ref. [7] for the dead load case. In case of seismic load, the dominant stresses are also the longitudinal normal stresses and other stresses are negligible compare to these stresses. Thus these stresses are plotted in Fig. 9 and large stresses are visualized around the apex of the shell. These large stresses are due to the finite element modeling with very large stiffness near the apex. However, they are still far below than the allowable compressive strength of reinforced concrete structure in the practical design. The stress resultants of the present shell subjected to both the dead and seismic loads do not much differ from those for the case of dead load only on most region. Mesh refinements may be needed for more rigorous analyses around the apex of the structure.



(a) Along bottom juncture line ①－①

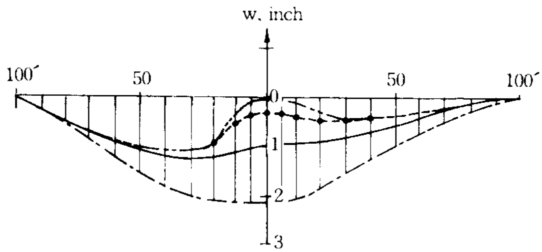


(b) Along top crown line ②－②

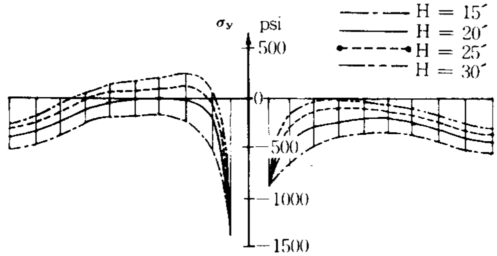
Fig. 9 Longitudinal normal stresses (psi)

To investigate the behavior of the multiple conical shell subjected to lateral loads, shell thickness, ratio of rise to span－length, central angle of individual cone, number of substructures, edge beam at juncture, etc., may be chosen as parameters to study their effects on the shell having the constant projected area in plan. Among them, the effect of rise to span－length ratio (H/Ro) is to discuss in detail for the shell shown in Fig. 8 as an illustration of the parametric study. The loading condition is the same dead and seismic loads discussed in the above. Keeping the radius of circle on horizontal plane constant, four different heights of the shell (H=15, 20, 25, 30 ft) are chosen as parameters to compare their results.

The displacements normal to the shell surface are shown in Fig. 10(a). They are plotted along the juncture which is the symmetrical line of the structure about the present loading condition. The increase of height results in a stiffer structure but it also increases the span length and the total amounts of materials. The location of maximum deflection in each case tends to shift toward the apex of the shell with decreasing the height and so the stiffner may be needed around the apex when the shell becomes flatter.



(a) Normal displacement along juncture of symmetrical line



(b) Normal stress along bottom juncture of symmetrical line

Fig. 10 Effect of height of multiple conical shell under seismic load

The dominant stresses on this multiple conical shell are the longitudinal normal stresses, and they are plotted in Fig. 10(b) along the same symmetrical line as for the above displacements. These stresses are almost proportional to the ratio of rise to span－length and tend to tensile side with the increase of height. However, the maximum stresses are always observed near the apex and nearly constant regardless of the rise to span－length ratios. Tensile stresses are not desirable in the design of a concrete shell structure and care should be taken in the practical construction of such shell when the heigth is relatively large compare to the span length since it may produce unnecessary large tensile stresses. In the present case, the ratio of 0.2 of the rise to span−length gives the best results. The effects of the other shell parameters to the multiple conical shell subjected to the seismic load are similar trends as those results discussed in Ref. [7], in which the shell was subjected to dead load only. Thus, it can be concluded that the major effect to determine the behavior of the multiple conical shell is the geometry of the shell itself.

6. SUMMARY AND CONCLUSIONS

The program MSSLL provides a versatile and economical method for predicting structural response of complex multiple shells subjected to the gravitational and lateral loadings. Uncoupled in−plane and plate bending stiffnesses were incorporated in each stiffness providing the element with 5 DOF at each nodal point and a fictitious rotational stiffness was introduced to treat systematically the element stiffness with 6 DOF. The novel feature of this program is that of substructuring. Using this approach, substructures with identical properties that appear in sequence can be treated very effectively since only the first substructure in the sequence has to be dealt with. The subject computer program recognizes repeated substructures in both stiffness calculations and stress recovery as well as input preparations.

The solutions obtained by MSSLL were generally in good agreement with the known solutions for the multiple shells treated here. However, some differences were observed at certain regions for specific structures, e.g., vertical displacements at free edges and transverse moments at the crown of midspan for the case of multiple cylindrical shell, and careful modeling should be taken into account for those regions in the finite element procedure.

A conical shell with eight substructures which was subjected to its own dead weight and uniform seismic loading was treated here in detail with some parametric studies by changing the shell geometries, from which it may be concluded that the behavior of the present shell is, in overall viewpoint, similar to that of the shell under dead load case. Although the loading condition is a very important factor to determine the behavior of structure, the geometry governs the behavior of the multiple conical shell. The parametric study was shown as a good tool to suggest economical and safe structure and to give some references for practical design of the multiple conical shell.

REFERENCES

1. Rao, P.S. and Prasadarao, A. "Influence of Corner Settlements on Stresses in Multispan Multibarrel Cylindrical Shells", World Conf. on Shell and Spatial Str., V.2, pp. 4.49−4.61, Madrid, Spain, 1979.
2. Tezcan, S.S., Agrawal, K.M. and Kostro, G., "Finite Element Analysis of Hyperbolic Paraboloid Shells", J. of Str. Div., Proc. of ASCE, Vol. 97, NO. STl, pp. 407−423, Jan. 1971.
3. Schaper, G. and Scordelis, A.C., "Comparative Linear and Nonlinear Analysis of Reinforced Concrete HP Groined Vaults", Bulletin of IASS, n.79, Vol. X X Ⅲ −2, Aug. 1982.
4. Setlur, A.V., "Analysis of Fluted Shell Roof Structures Circular in Plan", Ph.D. Dissert., Purdue Univ., 1965.
5. Torroja, J.A., "Use of Conical Vaults in Four Water Tanks with Straight Line Prestressing", Proceedings, World Cof. on Shell Structures, San Francisco, 1962.
6. Vyas, R.K. and Do, S.H., "Structural Economy of Certain Scalloped Shells", Proc. of National Str. Eng. Cof., Methods of Str. Analysis, Vol. 1, Published by ASCE, pp.205−221, New York, 1976.
7. Lee, P.S., "Finite Element Analysis of Multiple Shell Roof Structures Using Substructuring Techniques", Ph.D. Dissert. Univ. of Texas, Austin, Aug. 1985.
8. Turner, M.J., Clough, R.W., Martin, H.C. and Topp, L.J., "Stiffness and Deflection Analysis of Complex Structures", J. of Aero. Sci., Vol.23, No. 9, pp.80-

5－823, 1956.

9. Johnson, C.P., "The Analysis of Thin Shell by a Finite Element Procedure", Ph.D. Dissert, Univ. of California, Berkeley, 1967.
10. Doherty, W.P., Wilson, E.L. and Taylor, R.L.,"-Stress Analysis of Axisymmetric Solids Utilizing Higher Order Quadrilateral Finite Elements", SESM Report No. 69－3, Univ. of California, Berkeley, 1966.
11. Will, K.M., Johnson, C.P. and Matlock, H., "Analytical and Experimental Investigation of the Thermal Response of Highway Bridge", Research Report No.23－2, Univ. of Texas, Austin, Feb. 1977.
12. Clough, R.W. and Tocher, J.L., "Finite Element Stiffness Matrices for the Analysis of Plate Bending", Proc., Con. on Matrix Methods in Str. Mech., Air Force Institute of Tech., Ohio, 1965.
13. Zienkiwicz, O.C., "The Finite Element Method", 3rd edition, McGraw－Hill, 1977.
14. Johnson, C.P., Thepchatri, T. and Will, K.M., "Static and Buckling Analysis of Highway Bridges by Finite Element Procedures", Research Report 155－lF, UT at Austin, Aug. 1973.
15. Przemieniecki, J.S., "Matrix Structural Analysis of Substructures", Journal of AIAA, Vol. 1, pp. 138－147, Jan. 1963.
16. Irons, B.M., "A Frontal Solution Program for Finite Element Analysis", Inter. J. of Numerical Methods in Eng., Vol. 2, pp.5－32, 1970.
17. Alizadeh, A. and Will, G.T., "A Substructured Frontal Solver and Its Application to Localized Material Nonlinearity", Computers and Structures, Vol. 10, pp.225－231, 1979.
18. Lee, P.S., "A Study of Loads on Conical Shell Structures", Jour. of the Architectural Institute of Korea, Feb. 1986.
19. "IBM 7090 Computer Program for Analysis of Multiple Cylindrical Shells and Folded Plates with Arbitrary Edge Beams [MULEL]", Dept. of CE., Univ. of California, Berkeley.

(접수일자 1989. 3. 31)