PAPER NO.571

COMPUTER ANALYSIS OF COMPOSITE SPACE STRUCTURES

BY

M. Ashraf, Zahid A. Siddiqi
and
Afzal Javaid

COMPUTER ANALYSIS OF COMPOSITE SPACE STRUCTURES

M. Ashraf,[1] Zahid A. Siddiqi[2], and Afzal Javaid[2]

ABSTRACT

For the analysis of composite space structures having six degrees of freedom per node, a computer programme is developed using FORTRAN-77. The analysis is based on finite element method employing both skeletal and thin shell elements. It also incorporates the eccentricity of members, both in horizontal and vertical planes, connecting at a node. The programme was initially developed for its use on main frame computers and subsequently has been modified to be employed on personal computers using Microsoft compiler.

For the validation of assumptions employed in the analysis, the results obtained from the programme have been compared with observations made on an experimental model of a composite triple layer bridge-grid. The two types of results are found to have good agreement with each other.

INTRODUCTION

Finite element method is used for the solution of various types of structures. Particularly, it is useful for the analysis of composite space structures, where the skeletal component is represented by line elements while the continuum portion may be simulated by flat thin shell rectangular elements. The major steps involved in the analysis of both the skeletal and continuum types of structures are the same with the exception of formulation of element stiffness matrices which require different treatment. These structures are sufficiently rigid, and therefore, the assumption of linear elastic behaviour is quite reasonable and the same is considered in this study. The force displacement relationships are set up and written in matrix notation as

$$K\,d = P$$

where, K = primary stiffness matrix of a structure
d = unknown displacement vector of a structure, and
P = appended load vector of a structure.

The primary stiffness matrix of a structure is formulated as an assemblage of member stiffness matrices. The member stiffness matrices are formed in the local or global Cartesian co-ordinates. These matrices, if developed in the local co-ordinate system, have to be transformed to the global ones before they are stored in the appropriate locations of the stiffness matrix. Modifying the system $Kd = P$ to take into account support conditions results in a system which provides a complete expression of the conditions of compatibility

[1] Professor, Deptt. Of Civil Engineering, Univ. of Engg. & Tech. Lahore, Pakistan.
[2] Asst. Professor, Deptt. Of Civil Engineering, Univ. of Engg. & Tech. Lahore, Pakistan.

and equilibrium throughout the structure. This modified system, if internally and externally stable, can be solved to determine displacements, which are used to obtain member forces and support reactions. The validity of solution is checked by comparing the applied loading and calculated support reactions by satisfying equilibrium conditions. A flat shell, idealised as an assemblage of a number of discrete elements connected at the nodes, generally have both the in-plane and out-of-plane effects (Zienkiewicz et al 1968, Zienkiewicz and Cheung 1964). To deal with the eccentricities between the members connected at a joint, in vertical and horizontal planes, a method suggested by Zhao et al (1984) is used.

The skeletal element used in the present study is that of a two ended member in space having six degrees of freedom. The plate elements in space should also have six degrees of freedom for a compatible solution of the overall stiffness matrix. However, the present shell element formulation includes two translational degrees of freedom (dx & dy) per node for the in-plane stresses and three degrees of nodal freedom (dz, θx & θy) for the bending action. Hence combining the two actions results in an element with five degrees of freedom per node. It follows that inplane rotation "θz" has been neglected. The solution of the problem of non-existence of θz can be dealt with by associating a fictitious couple "Mz", which will have the two consequences as given below:

First that at nodes where both the skeletal and plate elements meet, the continuum elements will not contribute in resisting moment Mz, which is entirely taken by the line elements. And, second that if all the elements meeting at a node are co-planar plate elements, the modified system of simultaneous equations of Kd = P will not have a unique solution due to zero stiffness coefficient relating to θz. To overcome the singularity problem of the structure, a non-zero value, say 1, at the position of diagonal coefficients of stiffness matrix relating to each θz degree of freedom of plate elements is set. All the other coefficients relating to the θz in the appropriate rows and columns are left equal to zero.

In this paper, the algorithm of the developed computer programme is given. This programme can be used for the analysis of skeletal/continuum space structures. The programme was initially developed for its use on main frame computers PRIME 740 and subsequently has been modified to be employed on personal computers. For the validity of results obtained from this method, a number of structures have been modelled with this programme and a standard packaged known as LUSAS. The two types of results are found identical. Further, the results of this programme have been compared with experimental observations made on a composite triple layer bridge-grid model. It is gratifying to note that two types of results are found to have good agreement with each other. The listing of the FORTRAN-77 computer code is open and is available from the authors on request.

THE PROGRAMME

The programme is organised in thirteen FORTRAN-77 subroutines each of which performs one or more than one specific tasks and linked with the other subroutines through the main programme as shown in the flow diagram, Fig. 1. The flow diagram shows the

seven major steps involved in the computer programme together with the corresponding subroutines which perform the tasks. These steps are briefly described below:

1. The required informations are generally arranged in a data file, which are transferred automatically to the computer during the execution of the programme through the open and read file statements. Subroutine INPUT requires member connectivity, geometrical and material properties of the structure, the loading and support conditions, and the eccentricity of members in horizontal and vertical planes.

2. Development of primary stiffness matrix 'K' is performed in the subroutine ASSM, which involves a number of operations including its initialisation in a 2D rectangular arrays "SS" by calculating half-bandwidth (Brebbia & Ferrante 1979).

   The member stiffness matrices are formed in order of members. The control is directed automatically to that portion of the subroutine which deals with the skeletal or continuum component of the structure. First, all the skeletal members (if any) are taken in turn and then shell elements (if any) are dealt with in succession.

   I. Each member is taken in turn. The element stiffness matrix "SM" and transformation matrix "ROT" are set-up in two separate 2-D arrays of the same size of 12x12. The matrix SM being in local co-ordinates is converted to structure axes, by carrying out the multiplication ($ROT^T.SM.ROT$) using a subroutine TRANSF. Then the coefficients of member stiffness matrix are posted into the relevant positions of the stiffness matrix 'SS'. A subroutine ELASS carries out this job by decomposing the member stiffness matrix and storing its diagonal and upper diagonal elements into the appropriate positions.

   II. As for the skeletal members, each continuum element is taken one by one. Dealing first with the plane stress part of the element, the matrix A(8x8) is generated using the polynomial function. Matrix A is then inverted by using a subroutine INVERS.

   The matrix $\int B^T DB dV$ is given in explicit form and is temporarily stored in the same array in which skeletal member stiffness matrix is kept but occupying a space of 8x8. Then the plane stress matrix is formulated by carrying out the remaining multiplication operation of $A^{-T}\{\int B^T DB dv\}A^{-1}$ by calling a subroutine TRANIT. This matrix is stored in an auxiliary 2D array (24x24). The member stiffness matrix due to plate bending action is formulated in the same way as for the plane stress action and is stored below the coefficients of plane stress matrix. However the matrices A and $\int\int B^T DB dx dy$ are of order 12.

   A subroutine ROTATE arranges the coefficients in the auxiliary array, which have both membranal and flexural actions, in appropriate form as required by shell element. At this stage, the element stiffness matrix of shell is in the local co-ordinate system which is transformed to the global co-ordinate system with the help of subroutine TRANSF by using the a procedure similar to the skeletal members but the order of array ROT is equal to 24 in this case. Further, the eccentricities in the

members are taken care of by generating eccentricity matrix in the array ROT by again using the subroutine TRANST. Finally the member stiffness matrix is posted to the appropriate locations of the global stiffness matrix by adopting the procedure similar to the skeletal members.

3. External constraint conditions are implemented by employing subroutine BIND.

4. Subroutine DETERM is used to solve the system of linear simultaneous equations using Gaussian elimination technique for the symmetric banded matrices. The unknown nodal displacements are determined and control is passed to the next subroutine.

5. Subroutine FORCES is employed to calculate the element forces. The nodal displacements related to a member are transformed to local co-ordinates and the member stiffness matrices are developed in the local co-ordinate system. Then the generalised forces at the nodes of each skeletal member is determined and stored in a one-dimensional array.

   The matrix A for the in-plane action of each plate element is developed and inverted with the help of subroutine INVERS. The formulation of $DBA^{-1}d_e$ is required for the determination of in-plane generalised forces. Similarly, the member forces due to out-of-plane action are determined. Subroutine MULPLY is used to perform multiplication operations involved in obtaining forces and subroutine FORT is used to arrange the resultant matrix according to appropriate degrees of freedom of the shell element. Further, the effect of eccentricities are considered accordingly for the generalised forces in the shell elements. The moments due to the plate bending action are converted into bending stresses by using relationship given by Ugural and Fenster (1981). Further, principal stresses are also calculated by using the basic elasticity concepts.

6. The reactions are determined at the support points of the structure and overall equilibrium of the external and internal forces is verified.

7. The output results are stored in a result file.

RESULTS AND CONCLUSIONS

The programme was debugged and a number of skeletal and composite space structures were analysed with it and a standard package LUSAS. The results obtained from the two programmes for these structures were found almost identical. Further, for the validation of the programme developed, a model of square on diagonal composite triple layer bridge-grid was tested under uniformly distributed and concentrated types of loading. The model was of a highway bridge with a scale of 1:10 whose top deck was represented by an aluminum sheet bolted to the top layer of triple layer skeletal frame made of brass tubing. It was 5 panels wide and 25 panels long each being a 17 cm square. Further its depth was kept equal to 20 cm. The strain and deflection gauges were used to measure strains and deflections at critical points. The measured deflections and strains were found to have good agreement with those predicted by the programme, Figs 2-5. The maximum discrepancy of

the experimental deflections from the predicted ones were only 3% and 5% for uniformly distributed and concentrated types of loading, respectively. The bottom layer strains had maximum discrepancy of 13% for the two types of loading.

There is no size limitation of the computer programme as such and the size of actual problem that can be handled depends upon the computer memory and judicious use of node numbering. The programme is also time efficient even when operating on a personal computer.

REFERENCES

Brebbia, C.A. and Ferrante, A.J. (1979), Computational Methods For the Solution of Engineering Problems, Pentech Press, London.

Coates, R.C., Coutie, M.G., and Kong, F.K. (1980), Structural Analysis, Van Nostrand Reinhold (UK) Co Ltd, 2nd. edn., UK.

Ugural, A.C. and Fenster, S.K. (1981), Advanced Strength and Applied Elasticity, The SI Version, Elsevier North Holland. Inc.

Zienkiewicz, O.C., Parekh, C.J., and King, I.P (1968)., "Arch dam analysed by a linear finite element solution program", Proc. Symp. Arch Dams, ICE, London, 19-36.

Zienkiewicz, O.C. and Cheung, Y.K. (1964), "The finite element method for analysis of elastic isotropic and orthotropic slabs", Proc. ICE, 28, 471-488.

Zhao, H.L., Wu, Z.Z, and Liu, J.L. (1984), "Experimental study, theoretical analysis and design of sheet-framed space structures", In: Nooshin, H., ed., 3rd Int Conf. on Space Structures, Elseiver Applied Science Publishers, London.