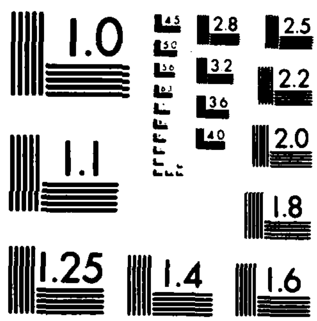
AD-A135 611 EXPERIENCE IN USING A FINITE ELEMENT STRESS AND VIBRATION PACKAGE ON A MINICOMPUTER(U) ADVANCED ENGINEERING LAB ADELAIDE (AUSTRALIA) S P PENNIMENT 1/1

UNCLASSIFIED 1982 F/G 12/1 NL

END
DATE
FILMED
1 84
DTIC



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS 1963-A

12

AD-A135 611

ACADS NATIONAL CAD/CAM CONFERENCE

HELD AT COMTEC '82

EXPERIENCE IN USING A FINITE ELEMENT STRESS AND VIBRATION PACKAGE ON A MINICOMPUTER

S.P. Penniment

Advanced Engineering Laboratory
Defence Research Centre Salisbury

83 12 09 070

AR-003-688

ACADS NATIONAL CAD CAM CONFERENCE
HELD AT COMTEC '82

EXPERIENCE IN USING A FINITE ELEMENT STRESS AND VIBRATION PACKAGE ON A MINICOMPUTER

S.P. Penniment

Advanced Engineering Laboratory
Defence Research Centre Salisbury

SUMMARY

This paper describes the use of a finite element analysis package running on an in-house PDP11/60 minicomputer at the Advanced Engineering Laboratory. Case studies are presented to illustrate the approach taken to problems involving stress level and deflection estimation and free vibration analysis.

INTRODUCTION

The structural design and analysis of military equipment, in particular airborne equipment, forms a significant part of the work undertaken by the Advanced Engineering Laboratory (AEL) at the Defence Research Centre Salisbury. Typically projects involve various kinds of equipment, such as optical, electronic, laser and radar supported and protected by a structure which is subjected to loading of several types, for example, the aerodynamic, heat and vibration loads that act on a store mounted externally on an aircraft flying at high speed and low altitude. The design of complex military equipment can involve specialists in many disciplines for long periods of time and major structural deficiencies discovered only after the production and subsequent testing or operation of a prototype can necessitate significant redesign in other areas besides the structure. Early analysis and evaluation of proposed structural designs with respect to common design criteria is clearly required. Hand calculations of structural stress, deflection and vibration modes, while providing essential guideline information to the designer suffer the following limitations. Firstly when dealing with complex geometric shapes, loading cases and support conditions, simplifying assumptions are required and these can affect the accuracy of the result. Secondly, even in cases where handsums yield an accurate answer there is not always sufficient time available to analyse all significant load configurations or work in an iterative manner analysing a number of proposed designs fully until an optimal solution is achieved. With the advent of high speed digital computers economical solutions to the classical mathematical equations governing structural behaviour are possible. The finite element method (FEM) is currently the most widely used technique of obtaining these solutions.

/2

THE FINITE ELEMENT APPROACH

In FEM computations the physical continuum is replaced by a discrete model consisting of a finite number of nodes connected by elements. The response between the nodes in the elements can be expressed as a function of the response at the nodes. The functional relationship between the two responses is approximated by various interpolation functions or shape functions. The type of function depends on the complexity of the problem at hand. This discretization reduces the original differential equations of the continuum to a set of algebraic equations which can be solved much more readily on digital computers.

Finite element experience at AEL has been predominantly concerned with linear elastic static deflection and stress estimation and normal mode computation for mechanical structures. The key equations for these computations are presented below.

1. Static Equilibrium

$$K\ (X) = (F)$$

where K is the stiffness matrix
(X) is a column vector of nodal displacements
(F) is a column vector of forces acting on the nodes.

2. Dynamic Response

$$M\ (\ddot{X}) + C\ (\dot{X}) + K\ (X) = (F(t))$$

where for natural frequencies W_n

$$(K - W_n^2\ M)\ (X_n) = 0$$

where M is the nodal mass matrix
C is the damping matrix
$F(t)$ is a forcing function dependent on time.

A thorough treatment of FEM is beyond the scope of this paper and the reader is referred to texts on the subject such as reference 1.

FEM IMPLEMENTATION AT AEL

FEM analysis for recent AEL projects has involved the use of a system of programs developed by the Aerospace and Mechanical Engineering Department of the University of Arizona known as the Graphics Orientated Interactive Finite Element Time Sharing Package (GIFTS). This package has been running on a PDP11/60 minicomputer.

Unlike many other FEM packages, GIFTS consists of a collection of fully compatible special purpose programs operating on a set of files on disk known as the Unified Data Base (UDB) containing all the data to describe the problem at hand. Each computational step such as the assembly of the stiffness matrix is initiated by running the appropriate program from the GIFTS library. The following is a list of the major GIFTS library programs with a brief description of what each program does.

Bulkm A three dimensional model generation program incorporating an automatic mesh generation capability. This program is used to specify material properties and the physical attributes of the elements (e.g. the thickness of a plate element or the diameter of a circular beam element), generate the key nodes, form lines that pass between key nodes and finally generate sequential strings of beam and bar elements lying on, and element grids (3D surfaces) bounded by, previously defined lines.

Editm A model generation program without automatic mesh generation capability intended for the creation of simple FEM models or for the modification of models after a Bulkm run.

Bulkf This program scans the element list and assigns freedoms to nodal points based on the class of problem, the type of elements connected to the nodes and their geometric orientation. The program will, for example, only allow freedoms in the plane of a grid of membrane elements, as the elements are not capable of supporting loads out of their plane.

Bulklb A bulk load and boundary condition generation program. Loads can be applied to key nodes, lines or grids, masses can be applied and inertia loading produced. Node freedoms can be suppressed or released. Prescribed displacements may be specified for key nodes, lines and surfaces. Temperatures can be applied to lines and surfaces.

Editlb This program provides local modifications to load and boundary conditions applied by bulklb, or generates simple loading on models. Loads are applied to nodes one at a time or to strings of nodes.

After running the above programs the model with static and kinematic boundary conditions has been prepared and checked ready for the execution of the following computational modules.

Optim A stiffness matrix bandwidth optimization program. Although GIFTS was designed to handle problems without size or bandwidth restriction, it is important to reduce the bandwidth of the stiffness matrix before decomposition because of run time considerations. Optim may be called several times in a row, until the best node numbering scheme has been achieved.

Stiff Computes the element stiffness matrices and assembles them into the master stiffness matrix.

Decom Introduces kinematic boundary conditions, and decomposes the stiffness matrix using an out of core Cholesky method.

Defl Computes deflections from the current loading conditions and the decomposed stiffness matrix. If temperature loads were applied to the model, thermal forces will be calculated and added to the current applied loads before solution.

Stress Computes element stresses based on current deflections.

Result A result display program. Deflections and stresses can be plotted or tabulated in various forms.

Advanced graphics features have been incorporated in GIFTS. The FEM model may be checked visually by displaying it from any angle, in any scale, rotation of the model about any cartesian axis is possible. A "boxing" command facilitates the viewing of subregions of the model in isolation including sectioned views. Selective viewing in which only elements of a particular type are shown is possible. Cross-section views of various element types are possible with the cross sectional area, second moment of area and other parameters calculated for beam and bar elements. Translational and rotational nodal freedom patterns can be displayed, as can plots showing load vectors acting on model nodes.

A comprehensive range of result displays is provided. Deflected shape plots can be used for example,to view the deflection profile of a support arm subjected to a static load in order to decide on the positioning of additional stiffening. Deflected shape plots can also be used to display vibration mode shapes. Stress contour plots enable regions of high stress gradient in plate bending models to be highlighted. Separate plots for the top, middle and bottom surfaces of plates are available. Various beam stress display capabilities exist, including bending moment and shear stress diagrams and plots at any beam section showing shear and axial stress distribution over the section in percentage of yeild stress for the beam material. Once an overall impression of where stress or deflection critical areas exist in a model it may be required to determine accurate results for all nodes in these areas, for this purpose tables listing nodal deflections and stresses are available.

Elements fully supported by GIFTS include the following classes:

1. Rod elements. Two and three noded versions are available. These elements are loaded axially and can be used to analyse two and three dimensional pin jointed trusses or to model stiffeners acting in conjunction with membrane elements.

2. Beam elements. Standard beam sections such as hollow and solid circular, hollow and solid rectangular, I, T, Z, channels and angles are available. All beam elements include the effect of non-symmetric bending, shear strains, torsional rigidity, position of the shear centre and allow for various points of attachment at their ends.

3. Membrane elements. Various elements with differing numbers of node points are available in this type. Elements that have constant strain and linearly varying displacement distribution, or linearly varying strain and quadratic displacement distribution, or quadratic strain with cubic displacement distribution are available. These elements can be used in the analysis of two-dimensional elasticity and in stiffened membrane problems, they may also be used to model three dimensional shell problems where the shell bending stresses play a minor role in the behaviour of the structure.

4. Plate bending elements. A three node triangular element and a four node quadrilateral element which is a simple combination of two three node triangular elements in which the stress values are averaged, are provided. These elements include both bending and membrane effects. The bending part assumes a linear variation of internal plate moments, including the twist.

5. Cosub element. These elements are in effect substructures composed of many elements and nodes. Cosub elements are generated during separate runs in which their stiffness, mass and other properties are generated and stored in a set of separate disk files. At the time of assembly of the cosub elements to the main structure only the master nodes on the cosub boundary lines are assembled. The displacements of dependent and internal nodes are kinematically constrained to those of the master nodes, ie once the displacements of the master nodes are fixed those of the dependent nodes are found by interpolation.

Two dimensional stress field problems that can be modelled by a combination of plate bending and beam elements have been the most frequently encountered class of problem since the introduction of FEM design analysis at AEL. The following case studies are presented to illustrate the approach taken at AEL to several typical problems.

CASE STUDIES

a. Aircraft Antenna Mount Design

The RAAF required a new blade antenna to be mounted on Mirage IIIO type aircraft. To attach the blade antenna to the aircraft it was decided to replace an existing access panel that was secured to solid airframe structure by another panel strengthened to accept the antenna. (See fig.1) Worst case loading on the antenna occurred in practice as the aircraft flew at very high speed at low level with some sideslip, this produced in addition to other smaller load components a large pressure loading acting in a direction normal to the blade plane.

A 5 mm thick aluminium alloy mounting plate design was proposed after consideration of several alternative designs, but before any prototype plates were manufactured or load testing performed, it was necessary to estimate the stresses and deflections that would occur in the loaded plate. The plate had a long central cutout to accommodate the electrical connections to the antenna and a curved shape with

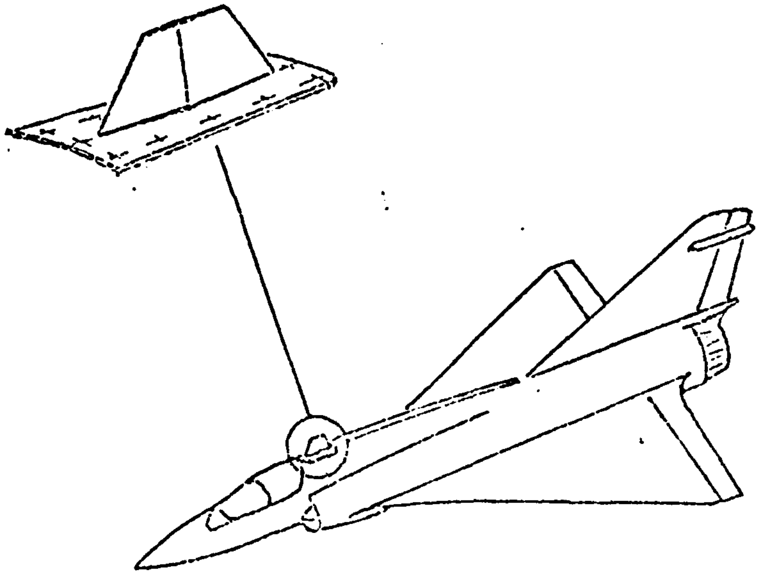
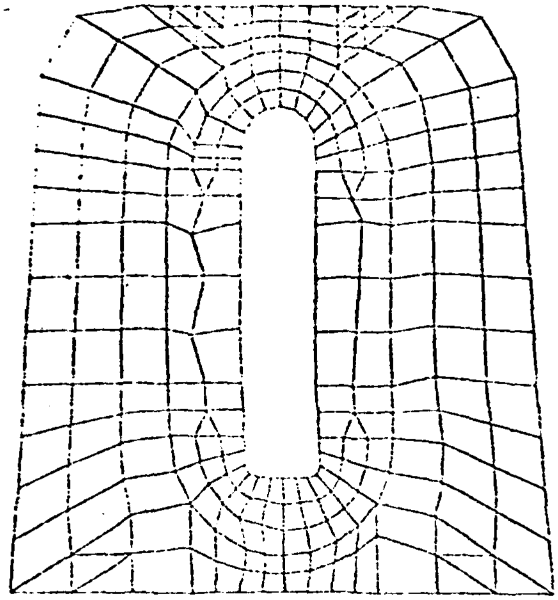


FIGURE 1. INSTALLATION OF A BLADE ANTENNA ON A MIRAGE IIIO AIRCRAFT

a different radius of curvature at each end. FEM analysis was chosen for this task because any hand calculation that did not involve oversimplification of the physical situation would have been very time consuming.

The finite element model for this analysis (see Fig.2) consisted of 4 node quadrilateral and 3 node triangular plate bending elements. To form this model several key points lying on the same plane were joined by grid boundary lines and the automatic mesh generator produced the internal grid details. Once the entire grid had been formed it was "wrapped" onto a predefined imaginary conical surface to represent the actual mounting plate shape. Predicted high stress areas in the regions around the curved ends of the central cutout were modelled with relatively small elements in an effort to more accurately estimate the stresses there.



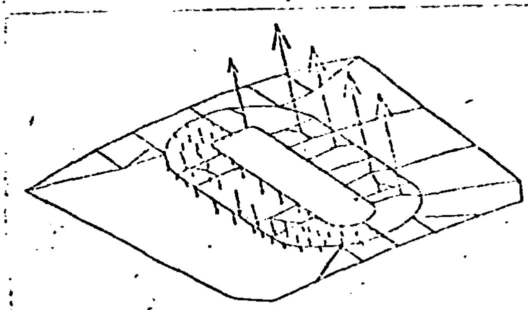


FIGURE 2. ANTENNA MOUNT FEM MODEL

FIGURE 3. LOADING PATTERN ON ANTENNA MOUNT MODEL

The antenna had been tested by the manufacturer to loads exceeding our requirement and so was not included in this analysis, instead the loads exerted on the plate by the antenna when subjected to the critical side pressure load were calculated by hand and programmed into the analysis as a static load. Load vectors applied to the model are shown in Fig.3, on one side of the antenna there is an evenly distributed pressure loading acting down on the plate, while on the other side of the antenna there are large concentrated loads pulling up on the plate at the screw points. Simply supported plate edges were assumed in this analysis.

Results for this analysis are presented in Figs. 4 and 5. Fig.4 is a contour plot of the outside fibre stress in the plate, showing high stress gradients in the predicted areas. Fig. 5 shows the deflected shape of the plate.

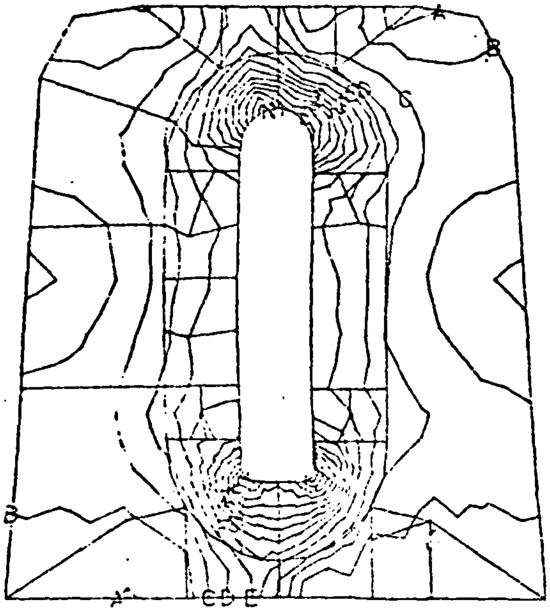


FIGURE 4. STRESS CONTOUR PLOT FOR ANTENNA MOUNTING PLATE

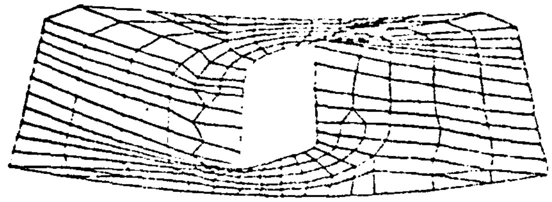


FIGURE 5. DEFLECTED SHAPE PLOT FOR ANTENNA MOUNTING PLATE

b. Fuselage Mounted Pod Design

Fig.6 presents a proposed design for the main structural section of a pod to be mounted on an aircraft fuselage. The design consists essentially of an aluminium alloy cylindrical canister, 1535 mm long, with an outside diameter of 430 mm and skin thickness of 5 mm. Relatively stiff bulkheads are located at each end of the canister and there is a cutout to provide access to internal electronics positioned at one end. The canister is suspended below the aircraft using two suspension lugs with two swaybraces provided to prevent lateral movement of the canister during flight. Optical equipment requiring accurate alignment with the aircraft fuselage is mounted from the front bulkhead and so in addition to the usual strength and rigidity requirements for an aircraft store such as this, there is a requirement for a very stiff load path between the forward bulkhead and the points where the store attaches to the aircraft.

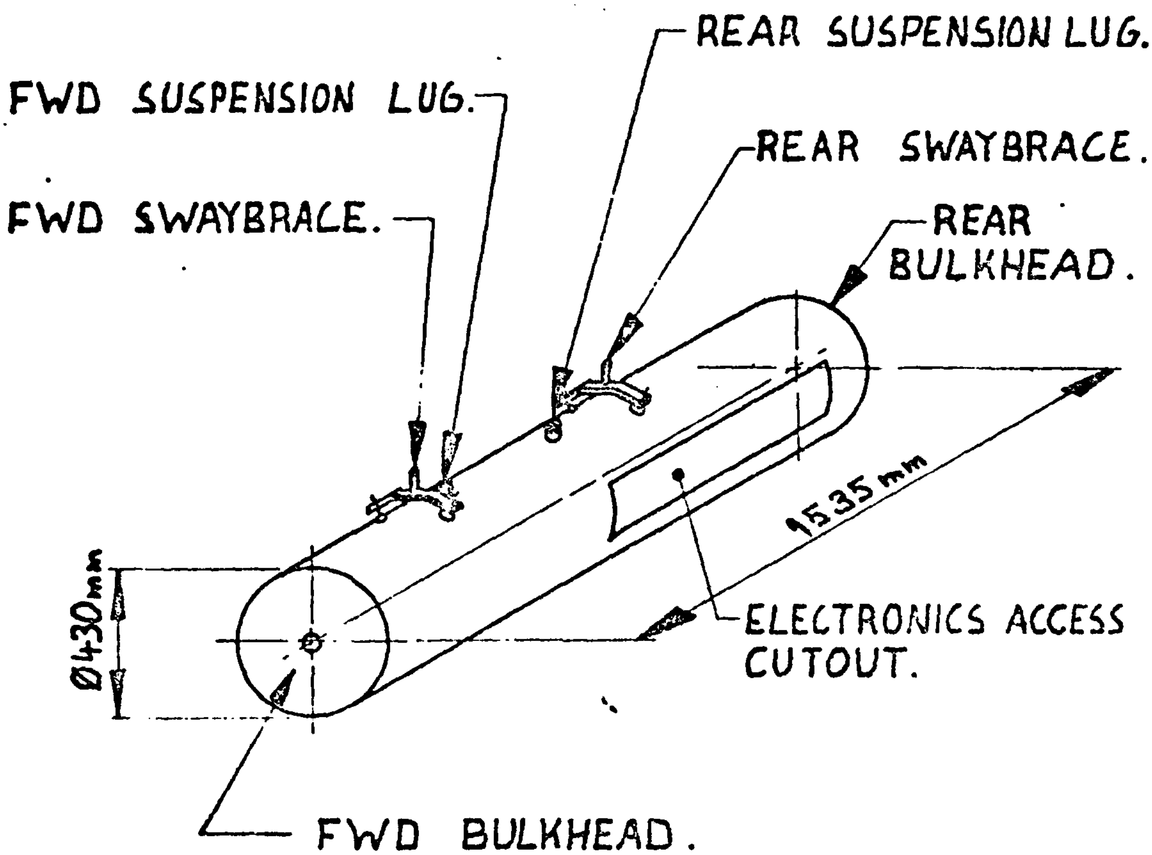


FIGURE 6. SCHEMATIC VIEW OF FUSELAGE MOUNTED POD

The initial model constructed from four node quadrilateral bending elements and used to obtain baseline design information is shown in Fig.7. All internal pod equipment is mounted from either the front or rear bulkhead, with the weight distributed evenly on each side of the bulkheads so that when the pod is subjected to a vertical acceleration, the moment load, M_x, acting on each bulkhead is small compared with the vertical inertial forces. To simulate the effect of these inertia forces on the canister the model was loaded with an evenly distributed down load acting around the circumference of each bulkhead as shown in Fig.8, while restraint consistent with the lug and swaybrace system was imposed.

Results of the baseline analysis are presented in Figs. 9 and 10. Fig.9 shows that the rear of the pod will deflect a greater distance than the front of the pod as expected since the suspension lugs are closer to the front bulkhead. Fig.10 illustrates the warping action in the front bulkhead caused by insufficient framing around the electronics access cutout at the rear of the pod.

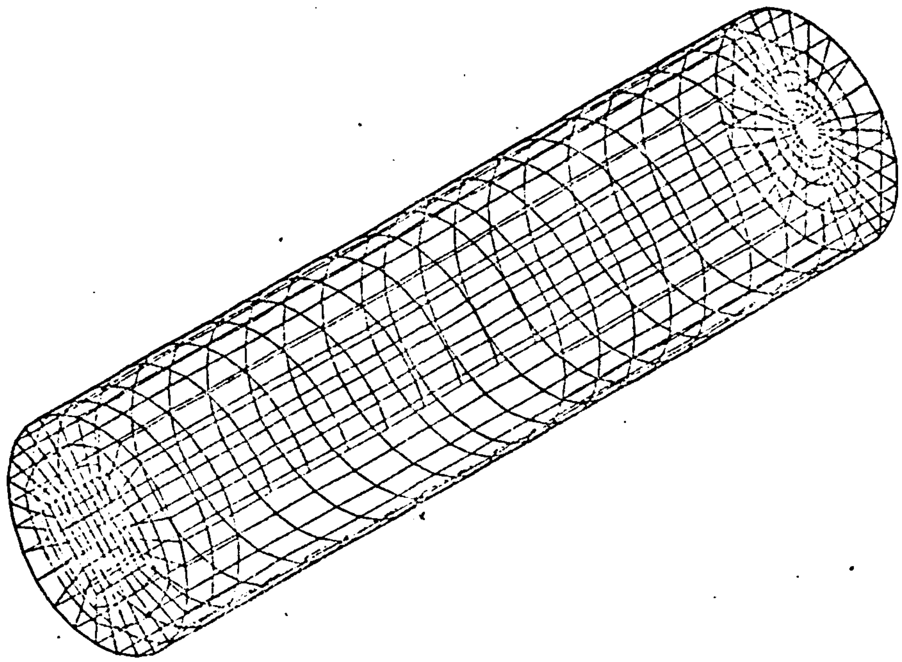


FIGURE 7. BASELINE ANALYSIS POD MODEL

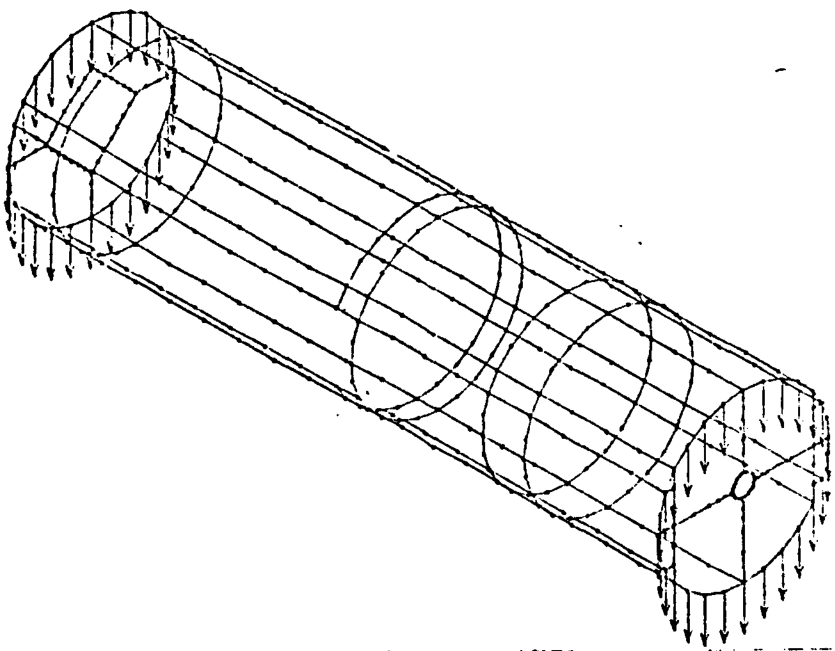


FIGURE 8. LOADING APPLIED TO POD MODELS

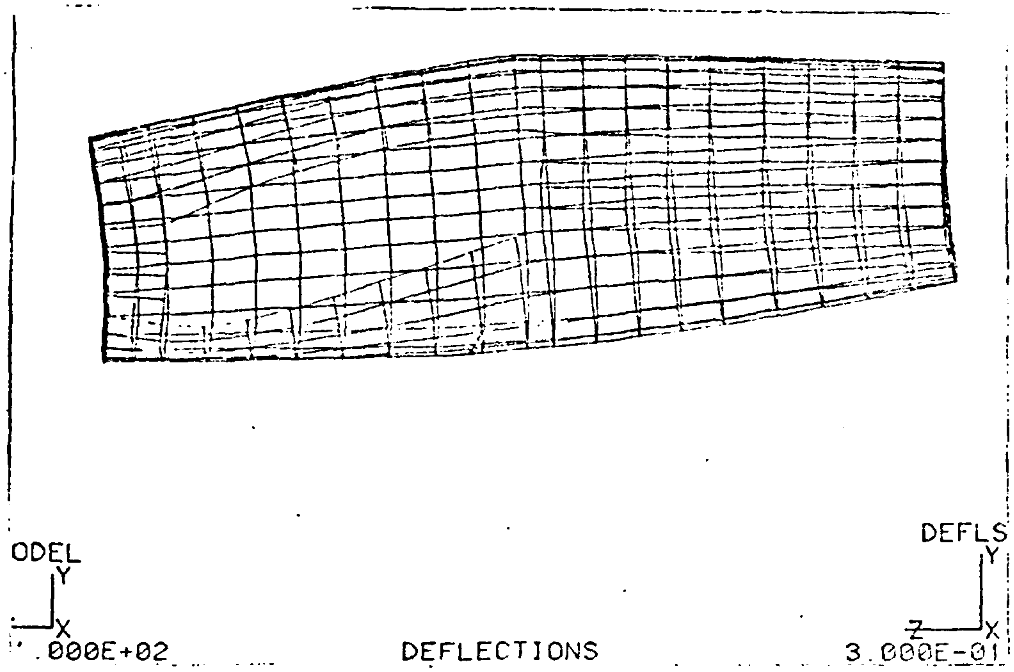


FIGURE 9. SIDE VIEW OF DEFLECTED BASELINE ANALYSIS POD MODEL

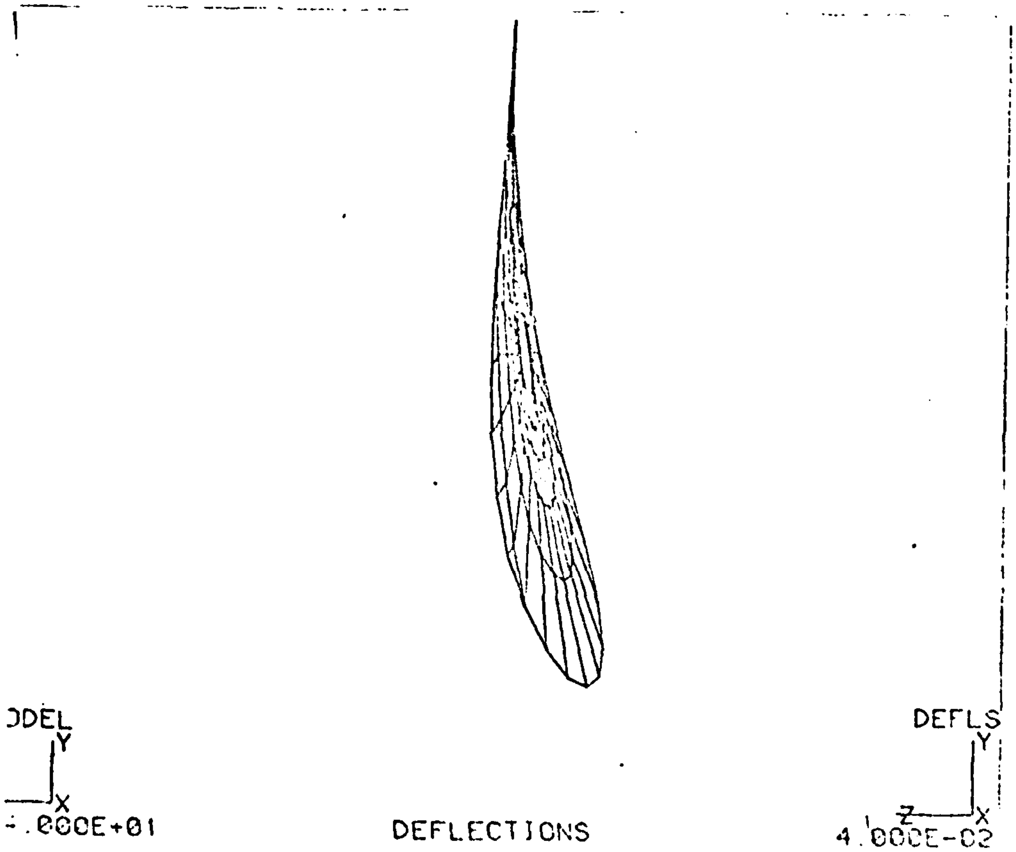


FIGURE 10. SIDE VIEW OF FRONT BULKHEAD DEFLECTED SHAPE - BASELINE ANALYSIS

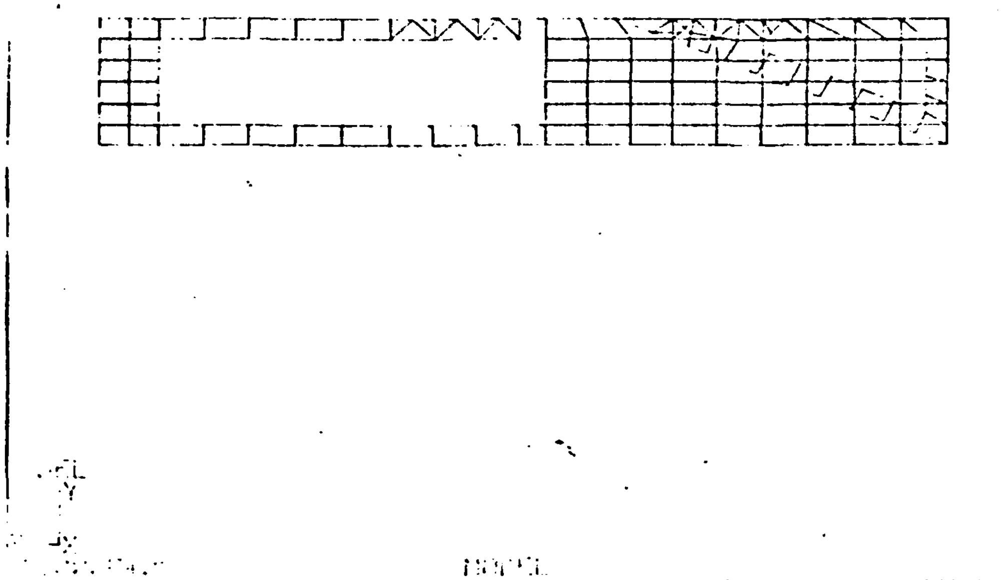


FIGURE 11. MODEL INCORPORATING TRIANGULAR BRACES TO SUPPORT FRONT BULKHEAD

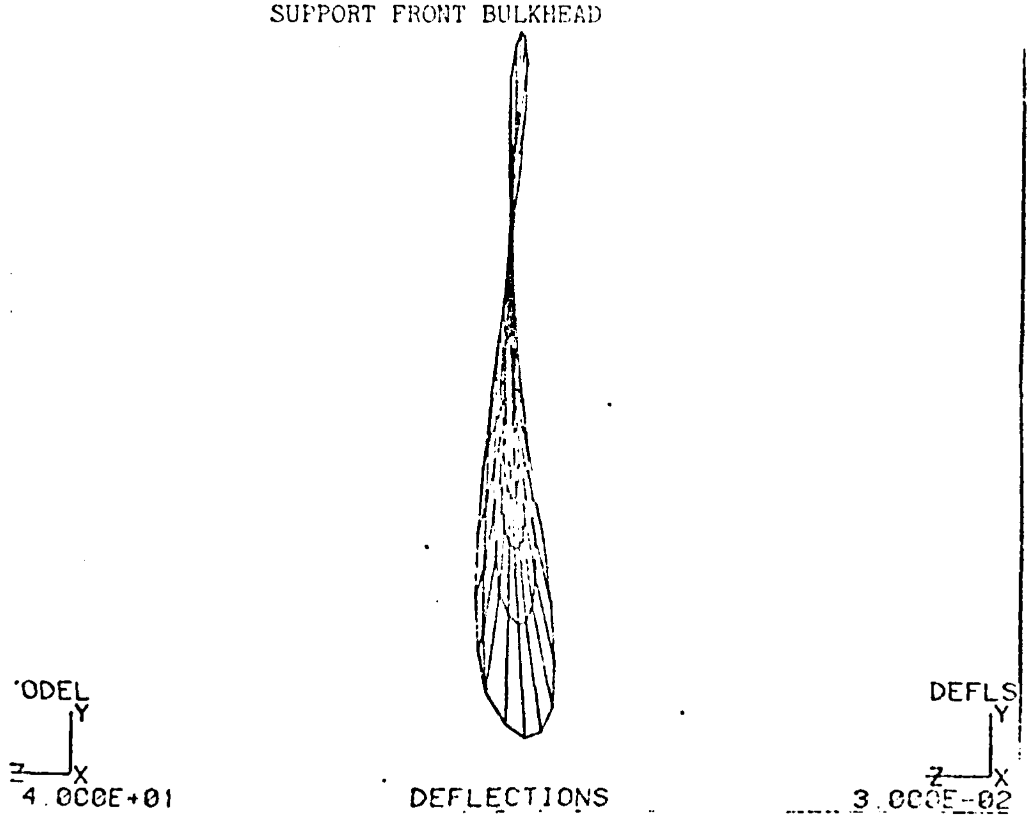


FIGURE 12. SIDE VIEW OF FRONT BULKHEAD DEFLECTED SHAPE FOR MODEL INCORPORATING TRIANGULAR BRACES

In order to stabilize the front bulkhead, it was suggested that two triangular braces, one on each side of the pod, be incorporated in the design. The braces were to attach along one edge to the canister in the suspension lug region and along another edge to the forward bulkhead. The braces are shown incorporated into the model in fig.11. The deflected shape of the front bulkhead of the model incorporating triangular braces is shown in fig.12. A change in mode and a reduction in magnitude of the bulkhead deflection is apparent.

The triangular brace design had a number of disadvantages related to non-structural aspects of the pod design and so other methods of reducing the front bulkhead deflection under load were investigated resulting in a final design incorporating heavy framing members around the cutout in the rear of the pod and stiffening hoop rings attached to the inside of the canister. As various design features were proposed for the pod structure they could be modelled and compared using the deflected shape plots as shown in figs 9, 10 and 12 and tables of nodal deflections. In this project FEM analysis has been included as an integral part of an iterative design process rather than simply a post-design checking aid and it is in this role that the full potential of the method can be realized.

c. Radar Support Leg

Fig.13 presents a side view of a FEM model that represents one of two main support legs for a shipboard radar assembly. Each leg was essentially a hollow, curved, tapered cantilever beam cast using aluminium alloy. At the top of each leg was a bearing housing through which the loads from the supported radar assembly were introduced to the leg, at the bottom the legs were bolted to structure directly attached to the deck of the ship. It was required to estimate the low order vibration mode shapes and frequencies for a support leg.

Four node quadrilateral plate bending elements were used to model the majority of the leg with three node triangular bending elements being used where necessary to stay within element aspect ratio limits and maintain sufficiently small element size in high stress gradient areas (as some investigation of the stress pattern in these areas was required). All nodes lying on the base plane were constrained in all six degrees of freedom. As is common practice in this type of design, one leg reacted practically all of the vibration load imposed by the mass of the supported equipment in the direction of the bearing housing axis except for friction effects in the bearing slide of the other leg. Vibration inertia loads in the other two axes were reacted in equal proportions by the two legs. To model this behaviour a useful technique in which the same node can have different masses assigned to it in each of three orthogonal axes was applied. /14

The leg reacting all of the equipment inertia load in the direction of the bearing housing axis was the one chosen for analysis as it had the lowest frequency modes due to the extra effective mass at the end of the cantilever structure.

In general the equilibrium equation for a vibrating system can be expressed in the following matrix differential equation form as noted earlier:

$$M\ (\ddot{X}) + C(\dot{X}) + K\ (X) = F(t)$$

In order to analyse the free response of the support leg a non-damped system was assumed as this will not affect the mode shapes or frequencies significantly, but it does simplify the above matrix differential equation. It was also assumed that no forcing terms were present, which is consistent with a free response analysis, with these assumptions the equilibrium equation was reduced to the following form:

$$M\ (\ddot{X}) + K\ (X) = 0$$

The general solution of an equation of this form can be written as:

$$x = \bar{x}e^{iwt}$$

the real part of which represents a harmonic response as

$$e^{iwt} = \cos Wt + i \sin Wt$$

substituting we find that W can be found from the equation

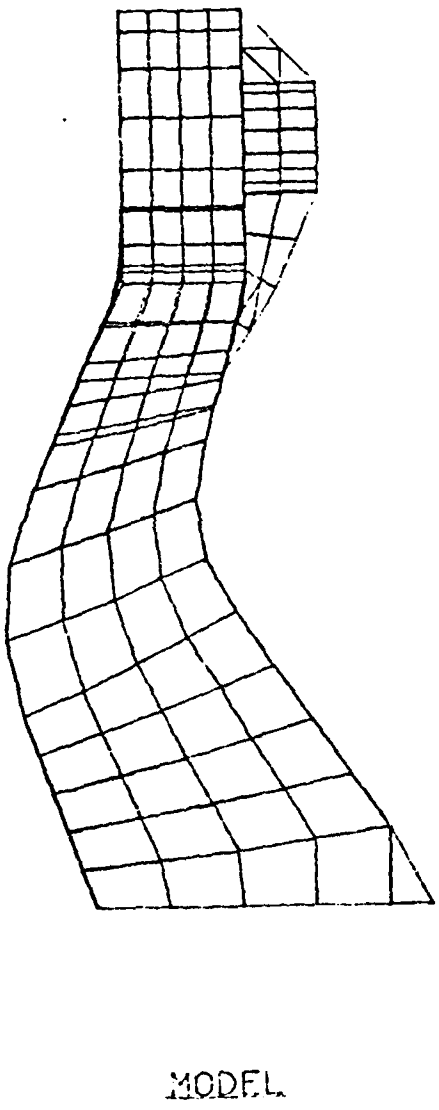
$$(-W^2M + K)\ \bar{x} = 0$$

non zero solutions for this eigenvalue problem require a zero valued determinant for the above equation.

$$\left| - W^2M + K \right| = 0$$

If the size of the above matrices K and M is nxn then the zero determinant requirement will yield n values of W^2. In structural problems the matrices K and M are usually positive definite and so n real, positive, values of W, the natural frequency of the structure can be found. Relative proportions of terms in the vector $\bar{x}$, corresponding to each natural frequency can be determined resulting in mode shape answers.

Programs to perform the type of analysis outlined above are available in GIFTS and were used in the analysis of the radar support legs. Fig. 14 presents a side view of the deflected shape of the first mode occurring at 41.9 Hz (compare with the undeflected shape in fig.13).



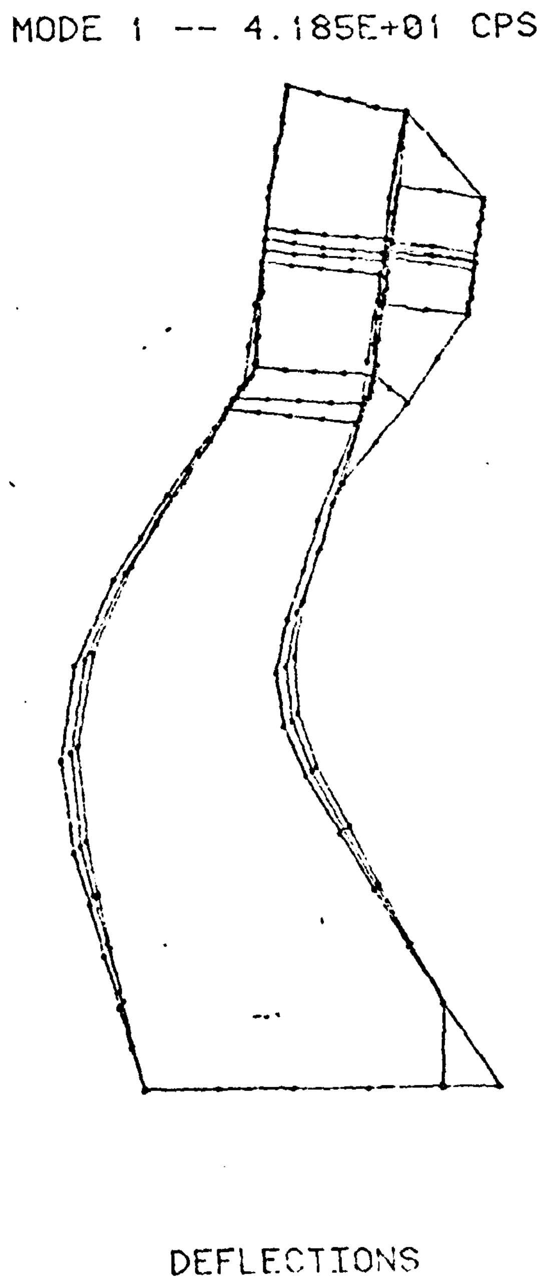


FIGURE 13. UNDEFLECTED SUPPORT LEG

FIGURE 14. DEFLECTED SHAPE OF FIRST MODE

RESULT CREDIBILITY

In addition to obvious mistakes which can occur when attempting to solve engineering problems using computers, errors of several distinct kinds can affect a FEM analysis. Round off errors may grow during the analysis of structural systems involving a large number of nodal degrees of freedom with corresponding large matrix sizes. At AEL we use a 16 bit word PDP11/60 minicomputer and so this source of error is of concern to us. When analysing large systems such as the fuselage mounted pod a technique is used in which the stiffness matrix is saved before decomposition takes place and then once a solution for the deflection vector (X) in the following equation

$$K\ (X) = (F)$$

has been computed following decomposition of the stiffness matrix K , the deflection vector (X) is premultiplied by the original stiffness matrix K and a result for the nodal force vector (F) is computed. This computed force vector is then substracted from the original load vector to give the residual forces which indicate the amount of error. For the fuselage mounted pod this error was small. Errors will occur when the element mesh is not fine enough to accurately model the structural continuum, this can occur if coarse elements are used to span regions in which high stress gradients occur rendering high local stresses undetectable in the result printout or if insufficient node points are included in a model to detect a high order vibration mode curve. In order to avoid this type of error structural design experience is essential. Intrinsic errors exist in finite element analysis for example, the displacement method of structural analysis yields an upper bound on stiffness when conforming elements are used, the result approaches the correct stiffness assymptotically as the element mesh is refined, but there is usually an economic limit on the degree of mesh refinement possible and some error has to be tolerated.

Hand calculation checks of FEM analyses have been performed at AEL for simple structures such as straight cantilever beams, (vibration mode shape and frequency calculations) and space frames (frame member stress and deflection calculations). Agremeent between the computer results and hand calculation results has been found to be reasonable in those simple cases, however it has not been feasible to check FEM results for complex structures in this way. When possible, full scale testing of complex structures should be performed to verify or point out deficiencies in FEM analysis, this has been done to a limited extent at AEL for example in the determination of vibration mode shapes of a radar dish.

CONCLUDING REMARKS

The operation of a finite element analysis package on an in house PDP11/60 minicomputer has proved to be successful in providing the Advanced Engineering Laboratory structural designers with a powerful computational aid. Analysis of large problems with over 5000 degrees of freedom has been possible as the GIFTS package uses overlayed program modules and matrix solution routines that make use of minimal core space, however long turn around times for large problems have been experienced, with most of the computer time being spent in the formulation and decomposition of the stiffness matrix. Element selection range has been a limitation, however, GIFTS has the ability to generate and display more element types than it has analysis capability for, and can be used as a pre- and post- processor for larger analysis programs. Future developments may include the use of GIFTS to create and display FEM models interactively on an AEL minicomputer while the long matrix operations are handled by a batch program running on a faster mainframe and interfaced with GIFTS.

LIST OF REFERENCES

1. Zienkiewicz O.C. (1977) - The Finite Element Method
   3rd Edition, McGraw Hill, London.

2. H.A. Kamel, M.W. McCabe (1973) - A Graphics Orientated Interactive
   Finite Element Time Sharing Package (GIFTS)
   University of Arizona.

3. Venkayya V.B., Tischler V.A. (1978) - "Analyze" - Analysis of
   Aerospace Structures with Membrane Elements
   Air Force Flight Dynamics Laboratory, Wright-Patterson
   Air Force Base OH.