Proceedings of the Institute of Acoustics

QUALITY ASSURANCE OF STRUCTURAL FINITE ELEMENT MODELS

J.D. McVee

Admiralty Research Establishment, Hydrodynamics & Structures Dept, Dunfermline, UK

ABSTRACT

A methodology is presented for carrying out major linear quasi-static structural analysis of naval surface ships using the superelement technique incorporated in the SESAM69C finite element computer program. Attention is drawn to quality assurance procedures undertaken at each stage of the analysis. In particular the application is described of a recently developed interactive graphic quality assurance processor which has proved to be particularly effective in ensuring the integrity of the structural model. The use is highlighted of other batch and interactive graphic utility programs, including the FEMVIEW results postprocessor.

INTRODUCTION

The last ten years have witnessed an accelerating proliferation of structural mechanics software implemented on increasingly available hardware configurations ranging from supercomputers to microcomputers, collated by Mackerle[1,2] and associated modern interactive colour graphics devices, as discussed by Croll[3] and Bailey et al[4], at continuously improving price/performance ratios. This, coupled with the widespread use, typically illustrated by Beames[5],of computer aided design systems many of which as described by Baillot et al[6] have either embedded or interfaced finite/boundary element modellers, solvers and postprocessors has led to international concern regarding the general quality of structural analysis. Amongst others, this has been crystallised in the UK by the formation of a National Agency for Finite Element Methods and Standards (NAFEMS) and in the USA by, for example, the American Institute for Aeronautics and Astronautics (AIAA) Standards Committee on Structural Analysis.

Mair[7,8] has articulated the wide ranging objectives of NAFEMS and its recent activities concerning the promotion of competent and accredited finite element practice; the development of benchmarking tools for the assessment of linear static, dynamic and non-linear elements and systems, and the clarification of reliable standards and requirements relating to the evolving art of data exchange between computer aided design and finite element systems. In association with the AIAA, MacNeal and Harder[9] have proposed a standard set of problems to evaluate the accuracy of linear finite elements subjected to static loads. Nagtegaal[10] has elaborated on quality assurance procedures employed by the vendors of a major non-linear finite element code which has to satisfy the requirements of the nuclear industry worldwide. Primarily from a users point of view, Boros[11] has discussed the quality assurance of proprietary software and proposed an international system for registration reflecting three software categories. Ball and Barlow[12] have described an implementation of practical quality assurance procedures, based on the NAFEMS philosophy, for commercial finite element applications using proprietary software in an industrial environment.

Feldsen[13], under the auspices of the Acoustical Society of America, is pursuing the quality assessment of numerical codes and the development of relevant benchmarks for underwater acoustic propagation. Finite element methods are now in routine industrial use[14] for structural, thermal, magnetic and acoustic cavity analyses. Widespread application is expected during the next decade to fluid analysis[15] including fluid-structure interaction[16] and the related problem of acoustic radiation from elastic structures[17]. Modern transducer design involves the more demanding solution of the interrelated structural, electric potential and fluid equations[18]. Although the following remarks address the theoretically simpler but non-trivial problems associated with relatively complicated elastic structural models, they are applicable to the basic treatment of acoustic models.

The purpose of the present paper is to outline a methodology for accomplishing complicated linear finite element analysis of naval structures which utilises the venerable but well established and reliable multi-level superelement technique implemented in SESAM69C[19]. Examples are provided from structural analysis of surface ships subjected to a given longitudinal hull girder bending moment distribution based upon a static balance in a design wave. Attention is drawn to certain quality assurance procedures applied at various stages of the analysis.

In particular the application of an interactive graphic quality assurance processor is described. This processor has been recently developed at ARE, Dunfermline, to take advantage of state of the art colour graphics terminal firmware. It can very rapidly manipulate and interrogate finite element model databases to provide three dimensional shaded surface, hidden line and wire-frame images which have proved invaluable in ensuring the integrity of the structural data and boundary conditions of finite element models. Additionally the use of several batch processors and interactive graphic utilities for rigorous results validation and presentation using FEMVIEW[20] is highlighted.

METHODOLOGY

The proposed methodology falls into nine distinct but interrelated phases namely: a thorough consideration of the aim of the analysis; structural idealisation; finite element discretisation; model datageneration; quality assurance of the datagenerated model; computed solution; quality assurance of results; some reanalysis using much of the original model and finally the production of results in a useful form suitable for presentation.

Aim of the Analysis

Knowledgeable judgement is required at the outset to seriously equate the aim, type and complexity of the analysis being considered, including the resulting size of the model, with the availability of suitable finite element and pre and postprocessing programs and resources in terms of experienced staff and adequate computing power and disk space. At this stage it is also often necessary to use prototype models to evaluate element behaviour and analysis techniques or to develop, and properly test, program adjoints to reliably aid with model generation or results evaluation and presentation.

Structural Idealisation

Obviously the extent of the model must be determined along with the loading and associated boundary conditions which can often utilise, or assume, planes of structural symmetry and loading symmetry and/or antisymmetry. It is also important to ensure adequate definition of scantlings, geometry, details of cut outs and material elastic constants especially those of any fibre reinforced plastics.

All the plated structure within the model can be idealised using quadrilateral or triangular elements with thickness equal to the average thickness of the actual structural plating. Frames, longitudinals, girders and carlings which stiffen ships hull and deckhouse structure have been represented either by corresponding single beam elements where judged necessary or by grouping them together to form lumped beam or uniaxial bar elements. In the course of deriving stiffened plate elements, Mukhopadhyay and Satsangi,[21] have confirmed that for typical orthogonally stiffened ships plating, lumped beam idealisations can provide accurate stresses in the plating and the stiffeners. Alternative representations which involve smearing the stiffeners throughout equivalent orthotropic plating have not been adopted since appropriate consideration must also be given to incorporating the effective breadth of plating acting in bending with the stiffeners and, if necessary, membrane shear lag effects. Corrugated bulkheads can be modelled as plane orthotropic plates.

A clear superelement strategy must also be arrived at. Economies in the total number of degrees of freedom (dof) in the model can be obtained by careful representation of planar structural components such as hull bulkheads and discontinuous decks by suitably supported membranes rather than thin shells.

Finite Element Discretisation

Clearly the types of finite element used along with their granularity influence the expected accuracy of the stresses and deflections obtained. Familiarity has been gained with the lower order membrane and bending elements currently available in SESAM and MSC/NASTRAN. Accuracy studies have been carried out by McVee and McLachlan[22] and the elements have performed satisfactorily in a wide range of applications.

The ubiquitous stiffeners dominate the discretisation in regions of fine mesh density and prohibit the global use of automatic meshing techniques. Moreover the common application of such procedures as unions of separate patches generates trapezoidally shaped elements which MacNeal[23] has proved to either lock or fail a C_0 patch test when subject to in-plane bending for the lower order elements available in the aforementioned systems as illustrated by Robinson[24]. Nygard[25] and MacNeal and Harder[26] have proposed modifications by including in-plane nodal rotations as degrees of freedom in the element formulation.

In any event quadrilateral elements should be kept as rectangular as possible and triangular elements only used where unavoidable. Rigid elements rather than triangular or distorted quadrilaterals should be used to effect mesh grading.

SESAM enables the finite element model to be composed of assemblies of first level superelements, known as SELs, each of which can be a membrane, shell or solid module. A multilevel superelement capability enables up to ten levels of superelements to be used with only the first level SELs composed of basic modules. The practical advantages in modelling, datageneration, computation and reanalysis to be gained from an efficient user-orientated implementation of the superelement technique are well known and such facilities now exist in many finite element systems. In particular, the recently developed, second generation SESAM hull and ship analysis system[27] includes interactive first and higher level structural and hydrodynamic wave loading pre-processors and interactive post-processors capable of statistical analysis of global response variables.

Frigate Discretisation Figure 1 illustrates a relatively small but complex 5000 dof frigate model described by McVee[28]. The aim of the analysis was to provide adequate definition of the force interaction between the hull and deckhouse and to determine the average stress distribution throughout the regions of obvious interest along the hull deckhouse interface, particularly at the deckhouse ends. A centre line plane of transverse structural and loading symmetry was assumed in this model which omitted minor hull bulkheads, the funnel and masts. Where necessary it incorporated the structurally weakest option. It was composed of three levels of SELs comprising 56 first levels distributed throughout seven second level assemblies each of which represented a complete longitudinal section of the ship. The response of the hull alone was readily achieved by deleting the first level SELs from the relevant four second level assemblies and solving the resulting 4000 dof model on the fourth level.

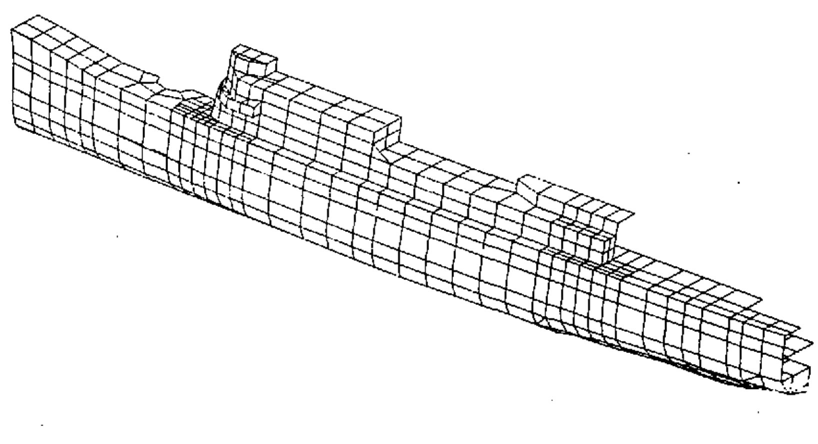


Figure 1. FEMVIEW plot of symmetric frigate model with typical coarse mesh discretisation suitable for load path analysis.

In generating the first level SELs using the SESAM69 batch processors which were developed primarily for ship structures and amenably cope with the multifarious beam elements representing stiffeners, care was taken to minimise the bandwidth in each SEL especially where nodal linear dependency was invoked. The higher level SELs were assembled with a strict geometric tolerance specified at coincident supernodes and using a string numbering system so that, if necessary, judicious manual numbering which took account of any linear dependencies enabled a near optimum bandwidth to be obtained at each level.

Analysis of the results quantified the significant contribution of the deckhouse to the effective midship section modulus and for the particular configuration of deckhouse considered, the relative increase in the longitudinal and shear stresses in the vicinity of the deckhouse ends. In addition the load distribution and hard spots where the deckhouse sides and ends intersect with supporting hull bulkheads were identified. The attendant stress concentration factors can only be theoretically obtained by successive mesh refinements which in the limit requires detailed solid modelling including the weldment at the hardspot. Excellent agreement was found however with experimentally measured 01 to 1 deck stress ratios. It was also demonstrated that classical solutions derived by Caldwell[29] and others to the perennial naval architectural problem of hull deckhouse interaction are inferior to those possible from suitable finite element models in the scope and accuracy of the resulting design information.

VSTOL Carrier Discretisation As shown in Figure 2 a comprehensive 38,000 dof model was developed to examine several novel structural design features incorporated in the first of class Vertical and Short Take Off and Landing (VSTOL) Cruiser discussed by Honner and Andrews[30]. This large scale discretisation had five major aims namely to obtain the stress distribution in way of groups of closely spaced openings, especially in the ship's sides, flight deck and the asymmetrically situated starboard island superstructure; to examine the effect of transverse asymmetry of the structural form on the global response; to determine the contribution to longitudinal rigidity of the hangar bulkheads which only exist above the lower accommodation deck; to quantify any need for compensating structure adjacent to the large bays in the

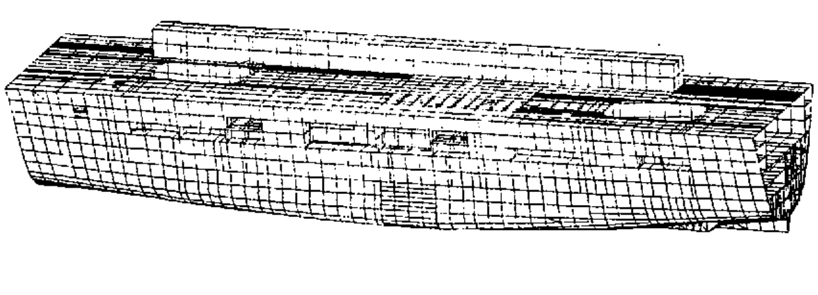


Figure 2. FEMVIEW plot of cruiser model.

hull sides and finally to ascertain any necessity for pillars to support the double bottom structure especially to docking loads. The model consisted of three levels of SELs comprising 89 first levels distributed throughout four second level longitudinal sections of the carrier. An indication of the degree of structural asymmetry is reflected in the fact that the idealisation necessitated 75 unique SELs. A reanalysis to compute the response of the structure including the pillars was readily obtained by incorporating 8 suitable identical first level elements in the third level assembly. In addition local refined mesh analysis, using displacements from the global analysis, were carried out adjacent to certain openings in the hull sides, flight deck and the island.

Detailed examination of the results indicated that:-

(a) although pronounced interactions occurred in the stress fields both adjacent to the original hull side openings of unequal height and between the openings originally proposed in the superstructure, minimal modifications to the local scantlings and corner details resulted in satisfactory design stress levels. Other adjacent openings, including those in the flight deck, were not design critical. Relatively high general stress levels exist in the island superstructure.

(b) the effects of transverse asymmetry of the structure could not be isolated from shear lag and natural discontinuities and perturbations in the global stress distribution which varied considerably from simple beam response.

(c) as expected, perturbations occurred at the extremities of the island and around cut-outs in the hangar bulkhead longitudinal shear stress distribution which otherwise closely followed the static balance shear force.

(d) there was no requirement for extensive general reinforcement of the structure surrounding the large bays in the hull sides.

(e) owing to the small length to width ratio of the machinery compartments, the limited number and type of pillars considered and the relatively low docking loads, pillaring to support the double bottom structure was ineffective.

Model Datageneration

Once the superelement hierarchy has been established, it is necessary to carefully derive from the relevant structural drawings accurate and detailed model layouts for each basic first level SEL. These describe the proposed finite element mesh and define the idealised plating thicknesses, stiffener types, material properties, supernodes, linear dependent freedoms and any nodal boundary constraints and loaded nodes. Similar clear and unambiguous diagrams showing nodes, supernodes, dependencies and boundary conditions are necessary for each higher level assembly.

Computer disk based data file creation and editing and output file interrogation is accomplished via an editor customised for SESAM69C use with specialised keypad features. Split screen rulers reflect the specific 80 column input formats and output is displayed in 132 columns. Efficient extraction and comparison of parts of data and/or output files is possible on the same screen.

First Levels Based strictly on the model layouts, suitably encoded input data is produced for each first level SEL. Beam data definition is automatically obtained by the use of a batch utility program which computes the relevant input data including cross sectional areas and out of plane inertias incorporating effective breadth of plating from given stiffener types, spacing and plate thicknesses. Between the datageneration iterations usually required to obtain the defined mesh use is made of various interactive program adjoints which utilise incomplete disk resident datageneration output files to automatically update the input data. It is thus possible to readily modify the input data to include shifts and scaling of nodal positions, nodal linear dependencies, radiused corners, circular penetrations, uni and bi-directional deck camber and to redefine the mesh positions and number of beam elements representing specific stiffeners. Moreover triangular and quadrilateral plate and beam elements can be easily inserted or removed. Output of generalised nodal forces and element principal stresses are requested as standard for each first level SEL. Quadrilateral element out of coplanarity is restricted to be no greater than 5°.

After each first level has been successfully datagenerated and thereby satisfied the checks inherent in SESAM69, it is subjected to a rigorous quality assurance datacheck using a powerful interactive graphic processor as described below. At this stage the idealised SEL scantlings are again directly compared with the structural drawings. Any necessary changes are then made to the SEL and the quality assurance process repeated until verification is obtained with the model layouts. Separate, appropriately scaled, computer plots are produced for each first level SEL showing unambiguously node and element numbering, plate thicknesses and beam cross sectional areas, supernodes and boundary conditions, and the mesh alone.

Higher Levels For each second level SEL the constituent first level datasets are concatenated into one file headed by encoded assembly data. In a procedure analogous to that used for first level datageneration, interactive program adjoints are utilised to match the lower level supernodes to the higher level nodes, to generate linear dependencies and to insert nodes into the assembly. Other interactive utilities exist which are readily capable of extracting the assembly node coordinate table from the datacheck output file, switching on or off constituent first level datacheck output directives and extracting or replacing the first level datasets in the concatenated input data file.

Quality assurance of the assembly data is facilitated by the use of interactive preprocessors which are capable of automatically determining constituent SEL axes systems relative to the axis of the highest level assembly, listing assembly transformations, calculating transformed first level coordinates and comparing the positions of first level supernodes. The admissibility of linear dependencies are graphically examined. Another processor is used to check and optionally automatically correct specified boundary constraints throughout the hierarchy. After it has been rigorously confirmed that the assembly corresponds with the model layout, higher level plots are obtained showing node numbers, supernodes, linear dependencies, boundary conditions and the unified constituent SEL mesh.

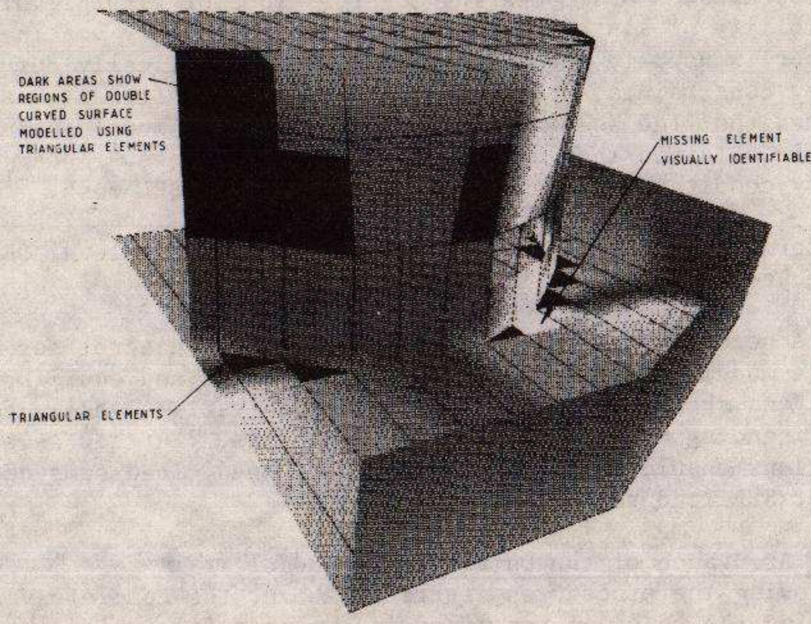
The intermediate assemblies and finally the highest level SEL are generated and checked in a similar fashion.

Loads A separate batch preprocessor evaluates, discretises and integrates into the model data, the applied vertical loads and equilibrating end bending moments and shear forces derived from given vertical shear forces and longitudinal bending moments at the ship ordinates. Further selective datagenerations enable a load audit to be obtained, load sums checked and the relevant SEL load vectors plotted.

Quality Assurance of the Datagenerated Model
By following the procedures described above, the complete validated input data describing the model is contained in a single file which is used to produce a binary datageneration restart database for the model. The accompanying complete datageneration output file is subjected to a final interrogation using the quality assurance preprocessor. Moreover the relevant assembly output is stored in a FEMVIEW database via an in-house interface and SEL concatenation utility. This enables batch generation of hidden line removal plots to be obtained for the complete model and point source shaded images of constituent higher level SELs to be hard copied from Tektronix 4129/4236 terminals. Back-up compared copies of the validated input and corresponding complete datageneration restart database, output file and FEMVIEW database are archived.

Interactive Graphic Quality Assurance Processor This host driven program has been written in FORTRAN 77 to take full advantage of the recent developments in high performance colour graphic terminals which incorporate a three dimensional local viewing system. Such devices enable extremely rapid hardware manipulation including rotation, translation, partitioning, zoom and pan of multiple light source shaded surface, hidden line and wireframe images. They are also capable of almost instantaneous switching between the shaded surface, hidden line and wireframe rendition of the high resolution raster screen image.

The processor is designed to facilitate rapid and thorough interactive interrogation of a datagenerated SESAM69 finite element model for possible errors. It automatically flags inadmissible linear dependencies and can also be used to interactively provide updated model input data. Visualisation is aided by colour coding the nodes in blue, beams in red, triangular elements in green and quadrilateral elements in white or grey. In particular, missing elements such as those indicated in Figure 3 can be more readily determined



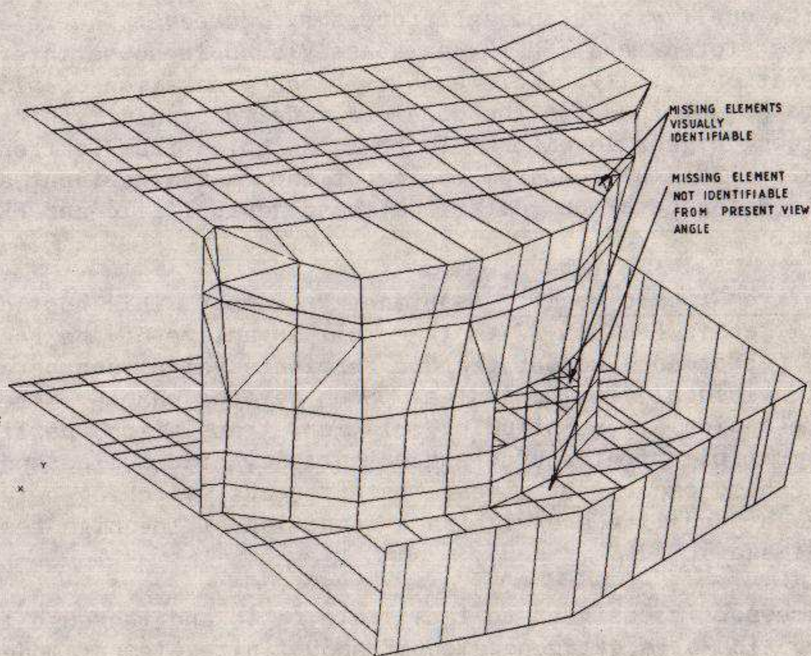


Figure 3. Monochrome reproductions from hard copy of colour shaded surface image and colour coded hidden line removal view of bridge SEL.

by manipulating shaded surface images rather than hidden line removal views. Regions of plating modelled by triangles are also clearly visible.

As shown in Figure 4a, the menu has a tree command structure, with help functions at main and subsidiary levels, and ten primary directives which are briefly elaborated on as follows:-

VIEW changes the displayed image by either adding or removing the selected entities as defined in Figure 4b from the displayed image. In addition the displayed SEL axis system can be translated to any screen position including any selected node within the model, or turned off.

SHOW highlights chosen entities by causing the relevant parts of the image to blink.

LOOK interrogates nodes, beams, quadrilaterals and triangles by cursor selection. Image rendition is automatically changed to give the most efficient use of the command. For a node the number, co-ordinates and datageneration definition are given. Distances between two consecutively selected nodes or a number of nodes relative to a specific node can be optionally displayed. For an element the number, type and nodal connectivity are given along with plate thicknesses or beam cross sectional areas. Annotated selected mesh and element topology data can be automatically produced.

FIND locates, highlights by blinking, and interrogates specified single or ranges of nodes and elements within the image. Unattached nodes are flagged, made visible and highlighted. The information displayed is similar to that obtained by LOOK. In addition if the element is a six dof beam the inertias and the number,co-ordinates and datageneration definition are also given and the guiding node made visible but not highlighted.

LDEP enables rapid and accurate checking of linear dependencies. Dependency definitions are displayed and the linearly dependent and dependent nodes made visible with the independent nodes flashing.

SNOD highlights specified supernodes and displays their co-ordinates and datageneration definition.

BCON displays boundary conditions of highlighted specified nodes and displays their co-ordinates and datageneration definition.

STAT extracts SEL statistics including number of nodes, elements, linear dependencies, supernodes, boundary conditions, filename and SEL number.

DCOM allows the experienced user to alter the shaded surface image display by changing the shaded surface reflectivity coefficients, the number of active lightsources and their position, intensity and mobility.

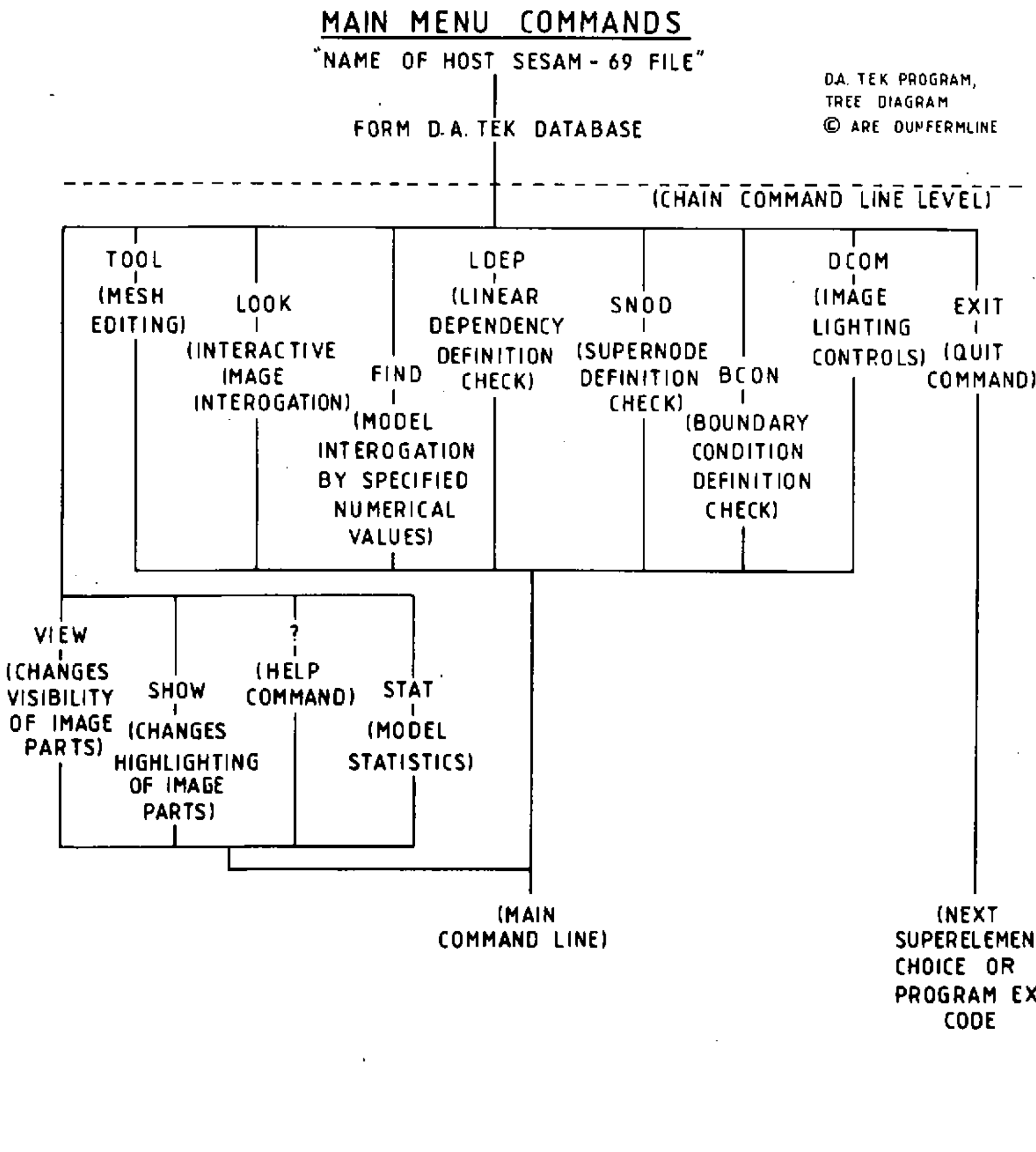


Figure 4a. Main menu commands.

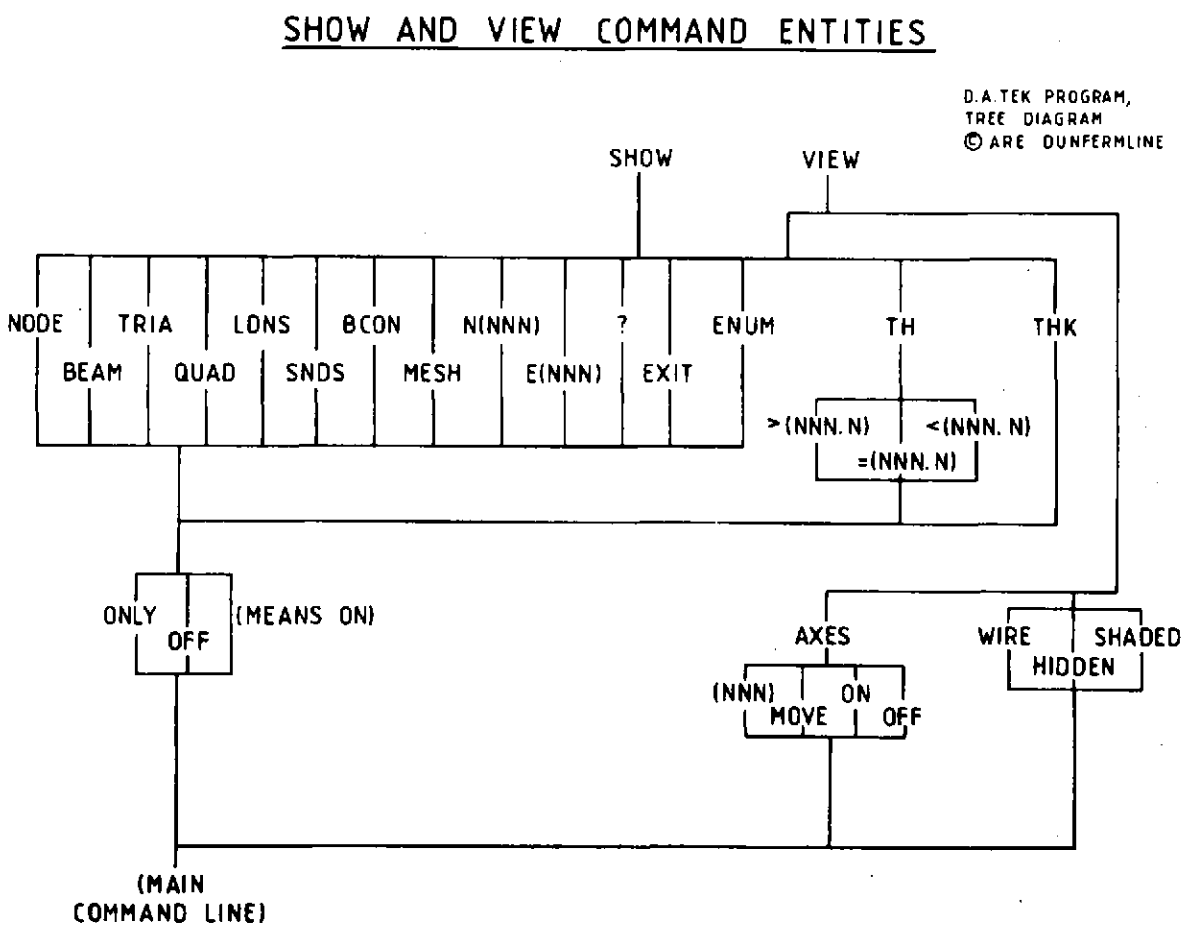


Figure 4b. SHOW and VIEW command entities.

TOOL facilitates correction of errors by invoking graphically interactive mesh editing facilities for inserting elements, shifting nodal positions and generating admissible linear dependencies. Each utility automatically updates the screen image, the internal database and, optionally, the input data file to reflect the current model state.

Computed Solution

Using a compared copy of the datageneration restart database, reduction of the first, intermediate and highest level SELs is carried out either separately or in groups and the reduced SELs stored in a complete structural model restart database which is immediately backed up.

If any singularities occur during this process, they are rectified by making the required adjustments to the necessary subset of data extracted from the complete input data file. This input data subset is independently datagenerated, quality assured and rereduced. Once the reduction is successful the complete data file, corresponding model layouts and datageneration and reduction output files are updated along with the packed datageneration and structural model databases. Double back-ups of these files replace those previously archived.

When the final equation solution of the highest level SEL has been obtained, the reduction output file is carefully inspected for near singularities, all diagonal decomposition ratios of less than 10^{-2} investigated and, if considered necessary, the relevant input data modified to affect correction as described above. In general the equations are considered to be numerically well conditioned if diagonal decay is nowhere less than 10^{-3} and 10^{-4} respectively for basic and higher level SELs.

Quality Assurance of Results

After ascertaining that the highest level displacements have no obvious irregularities and are of the same order of magnitude as those produced from an analogous simple beam idealisation, the entire model is retracked. Double back-ups are taken of the resulting binary postprocessing plot file and the retrack output file containing nodal displacements, generalised reaction forces at constrained nodes and supernodes, plate element centroidal membrane and inner and outer fibre bending components and principal and von Mises stresses, axial stresses and mid-length bending moments in beam elements, and generalised element nodal forces for every first level SEL in the model.

Postprocessing utilities are used to check that the individual and resultant generalised reaction forces at constrained nodes, linear dependencies and supernodes are satisfactory and that the computed bending moment at or near midships is consistent with the applied bending moment. Computed Euclidean norms of the ratio of residual and applied generalised nodal forces within each SEL are inspected to ensure that they are less than 10^{-4}. If not, the element forces within the offending SEL are summed, plotted and the equilibrium checked at each node in the SEL. Maximum displacements, element and nodal forces, principal stresses and stress components are automatically sorted and examined. Displacement plots obtained from the binary postprocessing file of assembly and individual SELs give a visual check on the structural deformations. Irregularities discerned during any of these checks are investigated, rectified as described above and the process repeated until the results are deemed to be satisfactory.

Reanalysis
Structural alternatives and remeshed regions of the original model are incorporated as necessary into modified existing or additional SELs and a reanalysis carried out using the procedures described above. The relevant input data, model layouts, output files and corresponding databases are considered as separate entities and their back-up files do not replace those created for the original analysis.

In certain instances a very useful SESAM69C facility can be invoked which enables existing SEL supernodal displacements to be automatically applied to obtain the response of only the modified SEL alone.

Presentation of Results
A suite of specially developed utilities are used to translate stresses and nodal forces into a form suitable for inclusion, via the interface, into the previously created FEMVIEW assembly database which is immediately backed-up. Rasterised hard copy is obtained of colour filled hidden line removal displays of element stresses and contour or vector nodal forces. In addition numerical values of stresses are plotted. Another utility is used to interactively annotate the FEMVIEW screen images and the corresponding hard copies and plots.

CONCLUSIONS

A proven methodology has been described for carrying out linear quasi-static finite element structural analysis of complicated naval surface ships. This is accomplished by using the superelement facilities available in the SESAM69C finite element computer program together with multifarious batch and interactive graphic pre and postprocessing program adjoints and the FEMVIEW results postprocessor.

Experience has shown that

(a) the application of efficient interactive graphic quality assurance procedures using a recently developed specialised intermediate preprocessor is exceptionally effective in ensuring integrity of the structural model.

(b) rigorous results validation procedures which rely on customised postprocessing utilities are mandatory.

(c) interactive colour postprocessors, such as FEMVIEW, are invaluable for the detailed analysis and presentation of results.

QUALITY ASSURANCE OF STRUCTURAL FINITE ELEMENT MODELS

The principles involved and program adjoints described are, with suitable modifications, applicable to similar finite element analysis using one of the many programs now available with well developed superelement capability.

DISCLAIMER

Any views expressed in this paper are wholly those of the author.

REFERENCES

[1] Mackerle J. (1986), MAKEBASE, An On-line Information Retrieval System for Structural Mechanics, Computers and Structures, Vol.24, pp.977-983.

[2] Mackerle J. (1988), Finite Element Codes for Microcomputers, Computers and Structures, Vol.28, pp.797-814.

[3] Croll B. (1988), The Benchmark Dilemma - An Expert's Guide, Computer Graphics World, Vol.11, pp.59-61 and 70-71.

[4] Bailey B.C., Hajjar J.F. and Abel J.F. (1986), Toward Effective Interactive Three-Dimensional Colour Postprocessing, Engineering Computations, Vol.3, pp. 90-98.

[5] Beames C.J. (1987), 3D Modelling and Model Management Applied to Ship Design, Vol.19, pp.560-565.

[6] Baillot J.F., Finifter D. and Kamel H.A. (1984), Integrating Cad-Cam Tools in the Design and Construction of Ships and Off-shore Structures, pp.221-236, Proceedings of Ship Structure Symposium '84, Arlington, Virginia, USA, 1984.
The Society of Naval Architects and Marine Engineers, New York.

[7] Mair W.M. (1985), The Objectives of the National Agency for Finite Element Methods and Standards, Computers and Structures, Vol.21, pp.875-879.

[8] Mair W.M. (1987), The National Agency for Finite Element Methods and Standards - Background and Status, pp.1.1-1 to 1.1-12, Proceedings of International Conference on Quality Assurance and Standards in Finite Element Analysis, Brighton, England, 1987.
NAFEMS, National Engineering Laboratory, Glasgow.

[9] MacNeal R.H. and Harder R.L. (1985), A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Elements in Analysis and Design, Vol.1, pp.3-20.

[10] Nagtegaal J.C. (1987), Quality Assurance for ABAQUS, Finite Element News, Issue No.6, pp.39-44.

[11] Boros L.J. (1987), Quality Assurance in the Computer Age, Computers and Structures, Vol.26, pp.495-498.

[12] Ball J.M. and Barlow J. (1987), Implementation of Finite Element Quality Assurance in a Commercial Industrial Organisation, pp.5.2-1 to 5.2-19, Proceedings of International Conference on Quality Assurance and Standards in Finite Element Analysis, Brighton, England, 1987. NAFEMS, National Engineering Laboratory, Glasgow.

[13] Feldsen L.B. (1988), Sound Numbers for Computing Sound, p.1 and Benchmarks-Purpose and Perspectives, pp.395-396, in Computational Acoustics: Wave Propagation (Ed. Lee D., Sternberg R.L. and Schultz M.H.), Proceedings of the 1st IMACS Symposium on Computational Acoustics, Newhaven, CT, USA, 1986. Elsevier Science Publishers B.V., Amsterdam.

[14] Supercomputer Applications in Automotive Research and Engineering Development (Ed. Marino C.), Proceedings of the 2nd International Conference on Supercomputing Applications in the Automotive Industry, Seville, Spain, 1988. Cray Research Inc, Minneapolis.

[15] What Every Engineer Should Know about Finite Element Analysis (Ed. Brauer J.R.), 1988. Marcel Dekker Inc, New York.

[16] Everstine G.C. (1981), A Symmetric Potential Formulation for Fluid-Structure Interaction, Journal of Sound and Vibration, Vol.79, pp.157-160.

[17] Lamancusa J.S. (1988), Acoustic Finite Element Modelling using Commercial Structural Analysis Programs, Noise Control Engineering Journal, Vol.30, pp.65-71.

[18] Hunt J.T., Knittel M.R. and Barach D. (1974), Finite Element Approach to Acoustic Radiation from Elastic Structures, Journal of the Acoustical Society of America, Vol.55, pp.269-280.

[19] Egeland O. and Araldsen P.O. (1974), SESAM-69 - A General Purpose Finite Element Method Program, Computers and Structures, Vol.4, pp.41-68.

[20] FEMVIEW version 4.0 User Manual (1986), FEMVIEW Ltd, Leicester.

[21] Mukhopadhyay M. and Satsangi S.K. (1984), Isoparametric Stiffened Plate Bending Element for the Analysis of Ships' Structures, Transactions of the Royal Institute of Naval Architects, Vol.126, pp.141-151.

[22] McVee J.D. and McLachlan L. (1984), An Accuracy Study of Some SESAM and MSC/NASTRAN Membrane and Plate Elements, in Accuracy, Reliability and Training in FEM Technology (Ed. Robinson J.) pp.202-210, Proceedings of the 4th World Congress on FEM, Interlaken, Switzerland, 1984. The Pitman Press, Bath.

[23] MacNeal R.H. (1987), A Theorem Regarding the Locking of Tapered Four-Noded Membrane Elements, International Journal for Numerical Methods in Engineering, Vol.24, pp.1793-1799.

[24] Robinson J. (1984), Application of Haggenmacher's Membrane Skew Tests, keynote lecture, Proceedings of the European MSC/NASTRAN Users Conference, Munich, West Germany, 1984.

[25] Nygard M.K. (1986), The Free Formulation for Nonlinear Finite Elements with Applications to Shells. Report No.86-2, Division of Structural Mechanics, The Norwegian Institute of Technology, The University of Trondheim.

[26] MacNeal R.H. and Harder R.L. (1988), A Refined Four Noded Membrane Element with Rotational Degrees of Freedom, Computers and Structures, Vol.28, pp.75-84.

[27] The SESAM Hull and Ship Manual (1988), Veritas SESAM Systems, Oslo.

[28] McVee J.D. (1980), A Finite Element Study of Hull-Deckhouse Interaction, Computers and Structures, Vol.12, pp.371-393.

[29] Caldwell J.B. (1957), The Effect of Superstructures on the Longitudinal Strength of Ships, Transactions of the Royal Institute of Naval Architects, Vol.99, pp.664-681.

[30] Honner A.F. and Andrews D.J. (1982), HMS INVINCIBLE The First of a New Genus of Aircraft Carrying Ships, Transactions of the Royal Institute of Naval Architects, Vol.124, pp.1-13.

Authors' Addresses

Dr. R. J. Brind, A.R.E.(N), Portland, Dorset, DT5 2JS.

Mr. J. R. Dunn, School of Electronic & Electrical Engineering, University of Birmingham, P.O. Box 363, Birmingham, B15 2TT.

Miss. A. Gallaher, A.R.E.(N), Portland, Dorset, DT5 2JS.

Dr. D. J. W. Hardie, A.R.E.(N), Portland, Dorset, DT5 2JS.

Dr. P. C. Macey, PAFEC Ltd., Strelley Hall, Main Street, Strelley, Nottingham, NG8 6PE.

Dr. J. McVee, A.R.E., St. Leonards Hill, Dunfermline, KY11 5PW.

Dr. B. V. Smith, School of Electronic & Electrical Engineering, University of Birmingham, P.O. Box 363, Birmingham, B15 2TT.