DEVELOPMENT OF STRUCTURAL STRENGTH FINITE-ELEMENT ANALYSIS TECHNIQUES*

V.D.Chuban**
Central Aerohydrodynamics Institute (TsAGI)
Moscow, USSR

Abstract

A development and application of the FEM are surveyed briefly on the basis of works performed at both the TsAGI and the USSR aircraft industry. Main attention is being given to tendencies in developing the finite element method software and in extending its field of application. Information about the MARS system built up at TsAGI and widely used in practice is given. Some examples of using MARS are reported including a solution of Buran statics/dynamics problems.

Introduction

The Finite Element Method formulated as such in early 1950s goes now through its vigorous growth period. A number of FE codes is available which allow engineers and scientists to perform static/dynamic analysis and structural optimization. A set of special-purpose systems has been intended to prepare data for the FE analyses and to process the results obtained. These systems are highly interactive and hold a dialog in terms of geometry images, say, a surface, a line, etc. The application of the pre/postprocessors results in dramatic reduction of the working hours required to perform the analyses thus doing the analyst profession more attractive. At present, the FEM is becoming the prime tool in structural strength analysis at all stages of airframe design projects. A practical dimension of FE models is as great as several tens of thousands and the entire airframe can be subjected to analysis. An exact prediction of force resultants at interfaces of subassemblies and the tendency toward a reduction of working hours for projects make it possible to look from a new point of view at the necessity of static tests. As is known, entire airframe static testing can now be replaced by the testing of subassemblies subjected to the external forces from FE analyses.

Also, the finite element method theory received a large development effort. The variational principles have been summarized and generalized, and hybrid elements have appeared; they include cracks, cut-outs, various nonlinearities, etc.

The FEM is applicable to conditions of large displacements, postbuckling deformational response, plastic deformation, forced/nonlinear vibrations.

It is a reality of present days that the FEM comes into a new stage of its development where three lines can be distinguished.

The first line concerns with the formulation and solution of structural strength problems with a desire to allow for all possible features of the physical processes under investigation. Particularly, nonlinearities in structural behavior are to be considered; these are the local skin buckling/postbuckling behavior, the cracking, etc. To solve such problems will be possible only after improving the FEs themselves (so that they were inherently nonlinear) and at providing the algorithms for embedded, multi-level models with an automated coupling on the meshes of different levels. Moreover, the analysis system must offer the possibility to use not only FEM but also other methods at each level of the hierarchy. Say, local strength problems for cracked structures can be successfully solved using the Boundary Element Method. Those problems obviously require computers of enormous speeds.

The second line concerns with introduction of FEM (and appropriate design programs) into a daily practice of designers. It is already being performed due to availability of low-cost PCs. The programs must not rely upon any pre/postprocessing, and the designer must be given full independence of any finite-element things so that he (or she) dealt only with a structure proper, and both the pre/postprocessing and FEM were "concealed" in a design code.

Wide prospects are seen owing to appearance of transputers and vector processors developed to the level of the usual plug-in cards for PCs.

The third line can be briefly characterized by the phrase "symbiosis of computer and software". As an example, let us note the experimental special-purpose computer named Finite Element Machine (NASA/Langley). This device is known to be a set of processors for a hardwared solution of the essentially new problems being too difficult for usual, general-purpose computers to solve.

Summarizing, it is worth to note that the FEM, throughout its existence, was steadily decreasing the working hours required for the solution of just most complicated problems.

**Chief of the Subdivision for computational structural mechanics

MARS: multi-purpose structural analysis/optimization system

The system is being under development in TsAGI since 1977. At present, it numbers some about 300,000 instructions. Since year 1981, the system is being applied to a structural analysis in aerospace industry as well as in automotive, construction and other industries. MARS makes it possible to analyze stress-strain states at thermomechanical loads and to determine normal modes and frequencies.

Load-bearing structures can include trusses, beams, bending plates/shells, composite structures, stiffened thin-walled members; as well, 3D problems can be solved. The possibility exists of optimizing the structures subject to stress and displacement constraints.

Developing the MARS system was being performed with regard to current achievements in numerical methods and operating systems. Thus, the following concepts are laid as a basis of the system itself and employing it:

1. A multitasking/multiuser operation capability for providing a collective and simultaneous work of structural engineers.
2. A shareable data base ("archive") with complete management system included into command language options.
3. Well-developed command language analogous to the known high-level algorithmic languages and based on unified syntax rules for both common-usage and problem-oriented instructions.
4. Dialog/batch options for all user-specified tasks.
5. Subsystems for all stages in pre- and postprocessing and main computations.
6. An adaptability to specific computer/peripheral features.
7. Options for checking the principal results.

The system shows an improved effectiveness when use is made of geometric modelling programs, for instance, the MARS/SGM for which MARS has interface routines. At present time, the user engaged in analyzing the complicated load-bearing structures is given the opportunity to use MARS/SGM for preparing the geometric model with a thickness distribution, an assignment of materials, a specification of loads, etc. These few data are then converted into huge, detailed information for the MARS to perform main calculations.

The system possesses a state-of-the-art finite element library including both the classical isoparametric FEs and special-purpose hybrid elements (with a hole, a crack, etc.). A material anisotropy and a dependence of characteristics on temperature are envisaged. Fig.1 shows some of curvilinear isoparametric FEs and the dot-element being very useful at modelling the lumped masses. Every element can have 2 to 5 nodes at every edge, thus being capable to provide a high-guality modelling.

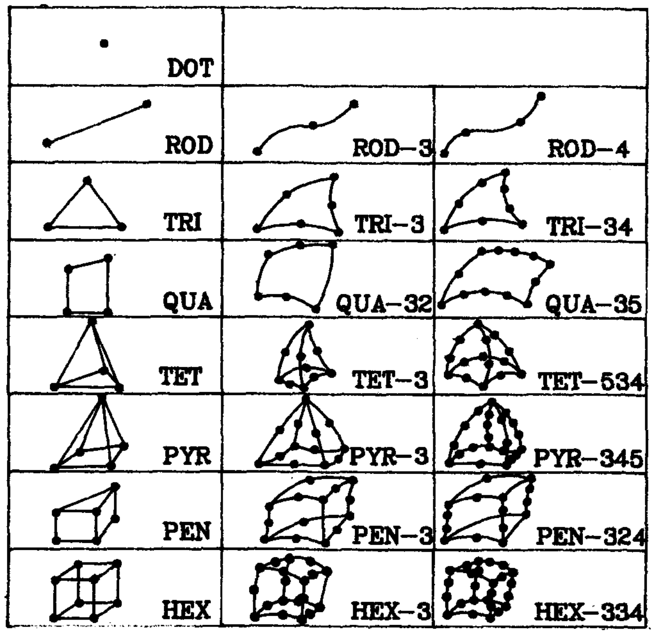

Fig. 1

The stiffness and mass matrices for one- and two-dimensional FEs are formulated using the fact that 3D integrals can be calculated through, first, an integration over a beam cross-section or a shell thickness, and, second, a subsequent integration over a length or a surface area., respectively. That is, in the case of an one-dimensional FE:

$$K_1 = \int_l D_1^T k_s D_1 dl, \quad k_s = \int_A D_s^T H D_s dA$$

$$M_1 = \int_l B_1^T m_s B_1 dl, \quad m_s = \int_A B_s^T \rho B_s dA$$

and those of a two-dimensional FE:

$$K_2 = \int_s D_2^T k_p D_2 ds, \quad k_p = \int_z D_p^T H D_p dz$$

$$M_2 = \int_s B_2^T m_p B_2 ds, \quad m_p = \int_z B_p^T \rho B_p dz$$

where K_i and M_i are the i-dimensional FE stiffness and mass matrices, respectively (i=1,2), k_s and m_s are the one-dimensional FE stiffness and mass matrices referenced to unit length, k_p and m_p are the two-dimensional FE stiffness and mass matrices, respectively, referenced to unit area; other things are well-known quantities.

Such an approach applied to a computation of the characteristics per unit length/area, respectively, makes it possible, in turn, to use powerful ideas of the FEM. With this, the possibility of taking account of an arbitrary warping due to torsion and transverse shear for cross-sections of arbitrary shape/topology appears.

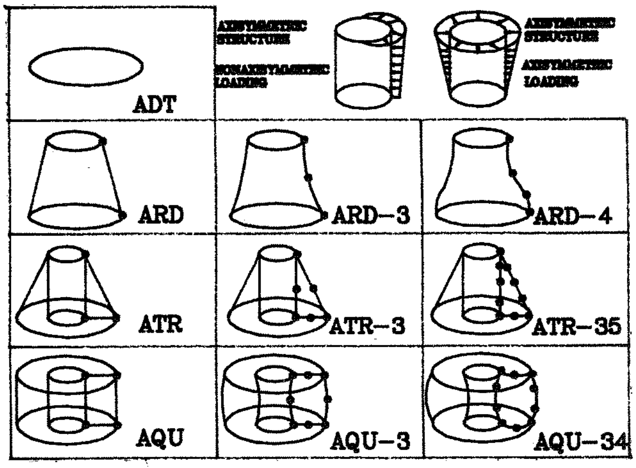

Fig. 2

The axisymmetric isoparametric FEs represented in Fig.2 enable structural engineers to analyze axisymmetric structures under any load and to obtain both axisymmetric and non-axisymmetric vibration mode shapes.

The special-purpose FE library represented in Fig.3 is implemented only partially. Some elements (with a load-free or pin-loaded hole, with a crack) took already part in practical analyses, some others (e.g., the element with corrections for the postbuckling behavior of a skin) are tested on toy problems, the remaining ones are under investigation.

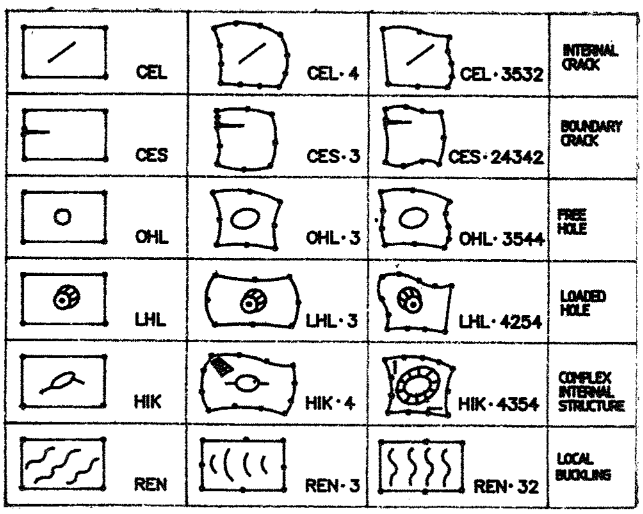

Fig. 3

As an example of application of the special-purpose FEs, let us consider the analysis of a large structure with an access hole closed by a bolted cap.

The analysis have been carried out using the hybrid finite element which comprises a sheet and a bolt of specified diameter. A bolt/sheet interaction was assumed to observe the cosine contact load distribution hypothesis. This assumption have given an exact analytical solution describing a stress state of the infinite sheet, and the hybrid finite element formulation have made it possible to couple these stresses and the nodal displacements. Here, a contact area was proposed to depend on the bolt force direction. The appropriate iterative procedure converges after 3 or 4 steps.

Structural statics

MARS system offers the opportunity to use the substructuring technique in both describing a finite-element model and solving various problems. The solution of the static equilibrium equations is performed through the multilevel decomposition where substructures are being step-by-step coupled into a hierarchical set of superelements. This is based on the quoted minimum degree ordering.

Let us consider the application of MARS to a static analysis of airframes. As is known, analyses of a global stress state occupy most place in computational investigations. By now, they are based, as a rule, on membrane models where the total number of state variables is as great as 20-30 thousands. Usually, a model covers the airframe as a whole, and there always exists a desire to describe the airframe as finely as possible, including all frames, ribs, stringers, etc. However, in itself this does not provide a validity of the results obtained since the model may comprise severely curved and/or prolate FEs, the FEs used can turn out to be of insufficiently high degree of approximation, the stiffness matrix may, for some reasons, be ill-conditioned; unfortunately, all these things were encountered more than once. An experience shows that only a careful, all-round checking of a model at every stage of project works can give a confidence in a validity of the results.

It is an important feature of MARS that the user is given a variety of options for inspecting the correctness of a solution to a problem, at all stages. Say, MARS envisages (i) a calculation of the actual rank of a finite element stiffness matrix and (ii) a checking for the possibility to displace a finite element as a rigid body without a strain energy accumulation. The equilibrium equation solution step can be subjected to verification on the basis of values of errors between the forces/moments applied and the reaction forces/moments at support points. With stress/strain fields computed, the user can check them by making MARS to intersect the structure by a user-specified plane and to calculate both the corresponding stress resultants and external forces and moments. An extent of accordance between them is a reliable figure of merit for the analysis; as for the units of wing/fuselage type this is of high clearness.

MARS has already been used for analyzing the statics of a number of airframes. Figs 4 and 5 represent complete finite-element models of Buran and Tu-204 transport airplane. These models have been created using experimental data and were used to support the static testing of the parents' structures.

Such an application of the FEM in experimental and analytical investigations

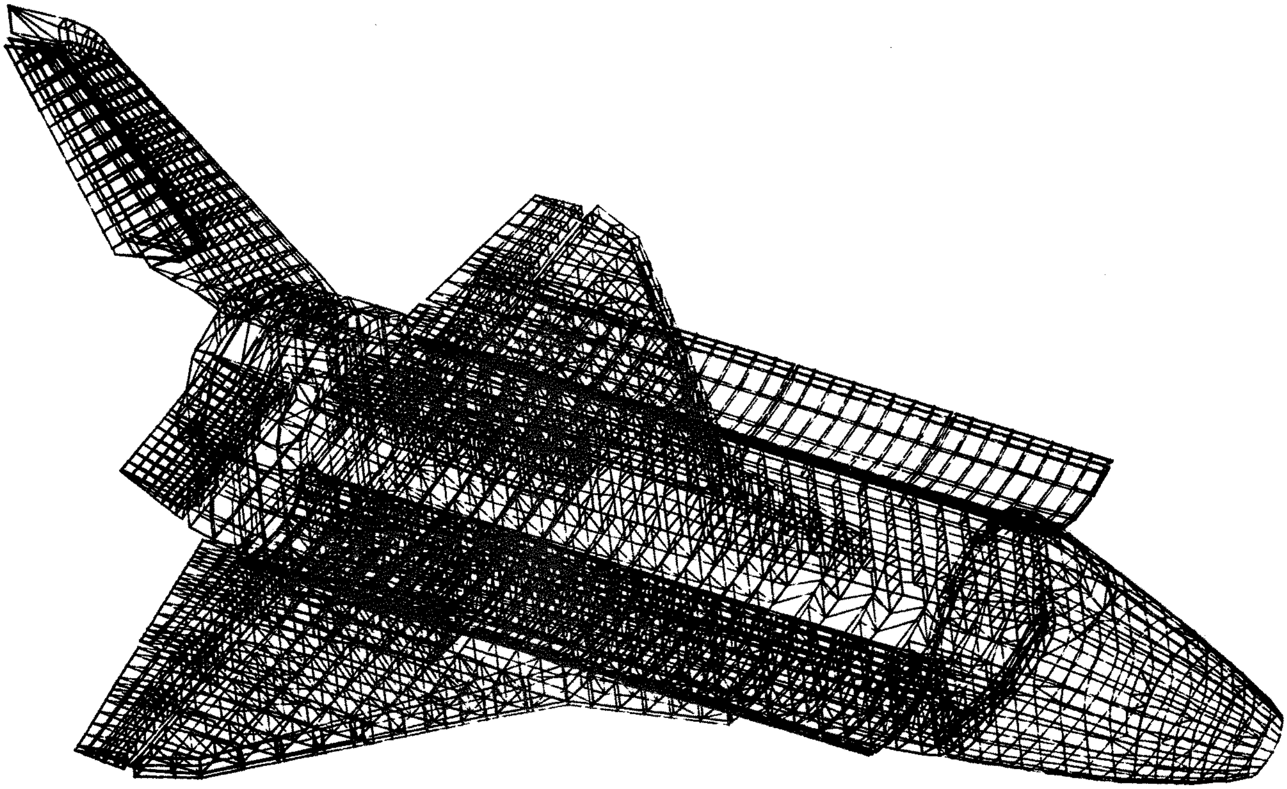

Fig. 4

is now becoming very important because of two circumstances.

On the one hand, there is a set of aerospace structures being unique and of tremendous cost. When testing those structures, no premature failure can be allowed, in no event.

On the other hand, there is a tendency to perform the static tests for every airframe subassembly separately. In this case the unit to be tested should be coupled with an auxiliary structure simulating the removed part of the main structure and possessing its own stiffness which, in general, is not equal to a true value. FEM can be very effective in predicting an actual state of stress at an actual stiffness distribution.

An analytical support of the experimental studies must be capable of predicting reliably the test results, and, in turn, the correctness and quality of the FE model must be checked against the test results. One should envisage some options for modifying the model.

So, Buran was successfully tested using only a part of its airframe - a body section and one half of a wing; the second half of the wing was taken into account only approximately. For the purposes of the analytical support, we created, first of all, an incomplete finite-element model in which the body section was represented well, including all particularities of an attachment system. Several analyses and modifications were performed to achieve a good correspondence between analytical and experimental results. Subsequently, this model was extended to a level of a structure as a whole (Fig.4), was used in a number of computational investigations, and will serve in future.

Let us consider other pattern. Fig.6 depicts the Buran's nose fairing - a shell of sophisticated shape, made of a composite material, and loaded thermally. Such subjects are of great interest because of complexity of a math model and the finite elements to be used (here, isoparametric bending shell elements of third order approximation, that is, 4 nodes at each edge). Analyses showed a critical stress level at shell attachment zones. An experiment conducted by the same time has revealed microcracks at these zones thus confirming the computed data though indirectly.

Analysis of dynamic behavior of an elastic structure

MARS performs a modal analysis by means of multilevel dynamic decomposition of both stiffness and mass matrices. This process, as well, consists in elimination of internal degrees of freedom of each substructure and is based on approximating the generalized displacements of internal nodes. That approximation can, in general, be performed as two versions:

a) an expansion in terms of static deformation modes,

b) an expansion in terms of normal vibration modes of a substructure fixed at its external nodes.

In the latter case, coefficients at the normal modes are considered as non-nodal generalized displacements - the generalized amplitudes of the substruc-

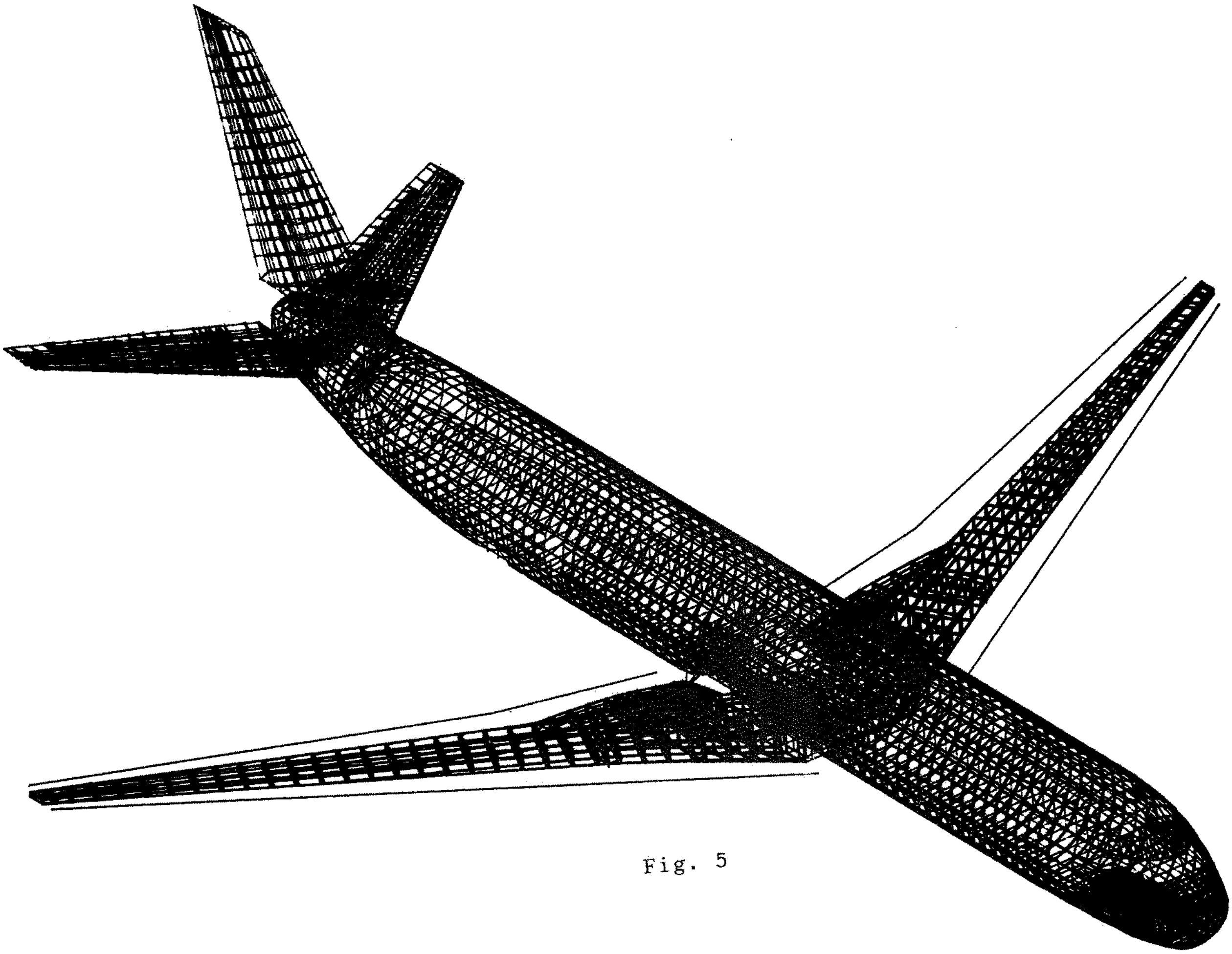

Fig. 5

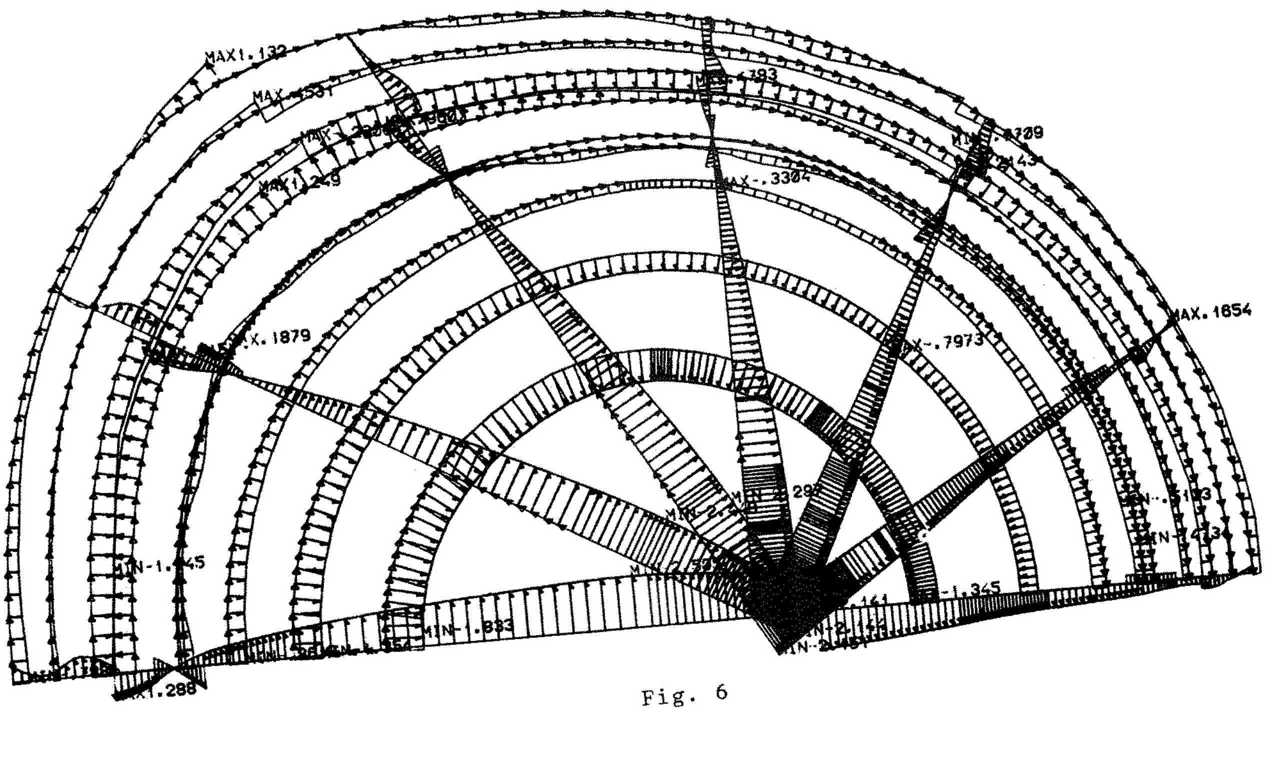

Fig. 6

ture. Actual displacements of the external nodes and the generalized amplitudes form the complete set of the substructure degrees-of-freedom, and the stiffness/mass matrices should be reduced to this set.

The first kind of approximation results in the so-called static decomposition. The latter makes the substructuring technique applicable to the modal analysis and lowers significantly the dimension of the eigenvalue problem, with the accuracy of modelling the dynamic behavior being kept high enough.

Nowaday helicopters raise questions of providing the dynamic strength, decreasing the fatigue stresses/vibration level, etc. An experience in designing the helicopters shows that all the questions caused by a dynamic behavior of an airframe must be considered and answered at the drawing-board stage since a "debugging" of a just finished aircraft becomes often a long, expensive process and is accompanied by a great deal of designer's efforts and an inferior performance. These problems are solvable at the drawing-board stage if use is made of a mathematical modelling and a refined dynamic model.

As an illustration of applying FEM to those problems and structures, Fig.7 shows results for the Mi-34 helicopter. The model (8160 unknowns) was created using a traditional technique. The helicopter contained units and equipment of significant weight, a part of them being located in a fuselage: an engine, a main reduction gear and a shaft, main rotor blades, a tail propeller and a reduction gear, an instrument panel, seats and pilots, a fuel tank, etc. These units have been modelled separately, with account being taken of a kinematics and an actual manner of attachment. We use a spectral shift to obtain 50 lowest frequencies for this FE model.

Comparing the analytical results versus ground modal test data showed a satisfactory agreement in both the frequencies and modes.

MARS system has been used in an analysis of Buran's dynamics. A model had 5000 variables which has allowed us to reliably determine natural frequencies over the range of 0 to 15 Hz and the corresponding normal modes, see Fig.8. The spectrum turned out to be very dense, with many frequencies being close to each other. Analyses have revealed various unusual modes of low frequencies, e.g., axial in-plane vibrations of a vertical stabilizer, transverse vibrations of frames and payload bay doors. The results obtained have been utilized in improving the Buran's dynamic model; they were later confirmed by full-size modal test data.

Development prospects

Concerning further development of FE codes for the structural analysis and optimization, we should note the necessity to integrate subsystems of (a) computing the nodal forces, (b) analyzing the aeroelastic phenomena, (c) describing the airframe from the designer's point of view, (d) evaluating the margins with respect to the strength, the buckling, the service life, and the load-carrying capability under prescribed criteria for the airframe (as opposite to a FE model), and (e) designing interactively the airframe.

At present days, programmers begin to introduce options for the detailed design of frames, stiffened plates, spars, ribs, etc. These systems are being created for designers to use; they would operate on PCs, have developed interaction means, quickly treat all candidate structural concepts, have direct informational communication with a versatile FE system (such as the MARS). Here, a designer should be given the possibility to describe a subject of inquiry in terms of structural elements: a spar boom, a stiffened plate, a corrugated wall, etc. For every structural member, margins with respect to strength/buckling requirements are calculated, account being taken of the entire set of the criteria established for each structural member. With that idealization, stress-strain states should be computed using special-purpose, hybrid FEs which are in one-to-one correspondence to the structural members. Such FEs can possess an arbitrary number of edges, be anisotropic. In formulating the FEs, use is made of the third, enlarged form of the potential energy functional which takes into account the forces mutually balanced inside a finite element:

$$P = \tfrac{1}{2}\int_{\Gamma} U_p^T T d\Gamma + \int_{\Gamma} D^T U_e d\Gamma \qquad (1)$$

where U_e is the approximated displacement vector at the boundary Γ; U_p and T are, respectively, the displacements and forces resulting from the self-equilibrium stresses in the FE.

To determine them, the fourth-order partial differential equation is applied:

$$\sum_{k=0}^{4} \alpha_k \frac{d^4\Phi}{dx^k dy^{4-k}} = 0 \qquad (2)$$

which generalizes the Airy equation to the anisotropy conditions.

Formulation of those finite elements requires a great deal of independent solutions to Eq.(2), and the problem becomes too difficult for developers. In this connection, use is made of an additional program based on the REFAL language and being capable of performing analytical transformations. A resulting program generates the required sets of independent vectors U_p and self-equilibrium forces T.

The approach outlined has already been implemented at compiling the attachment frame design system which functions on a 32-bit PC.

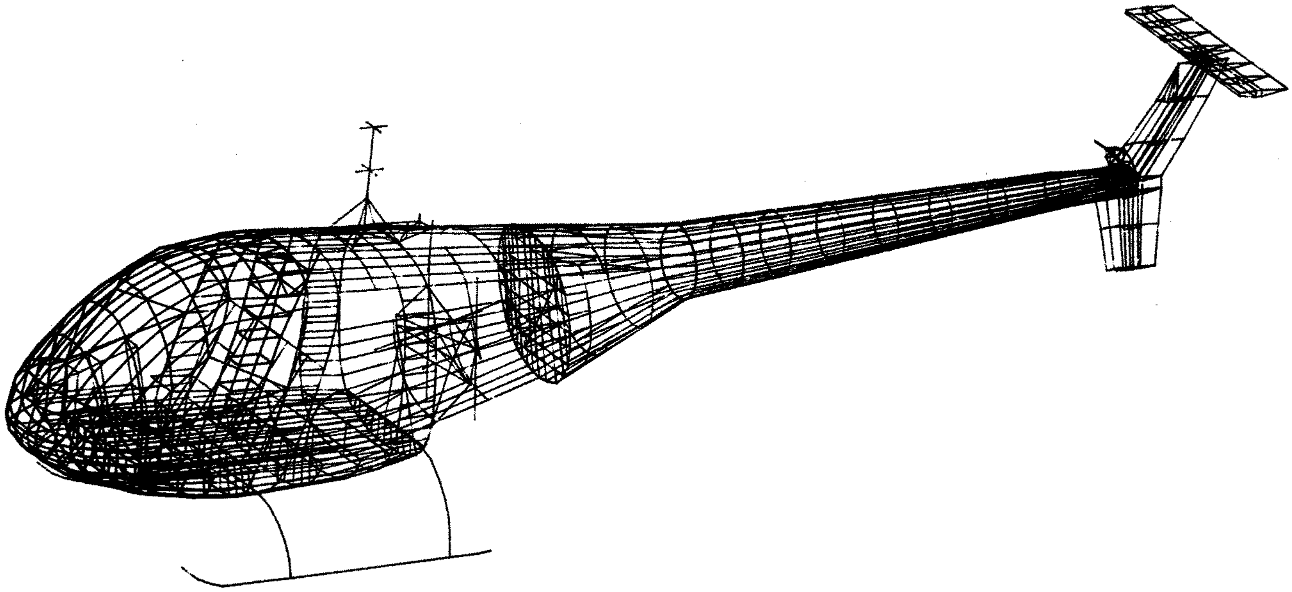

Fig. 7

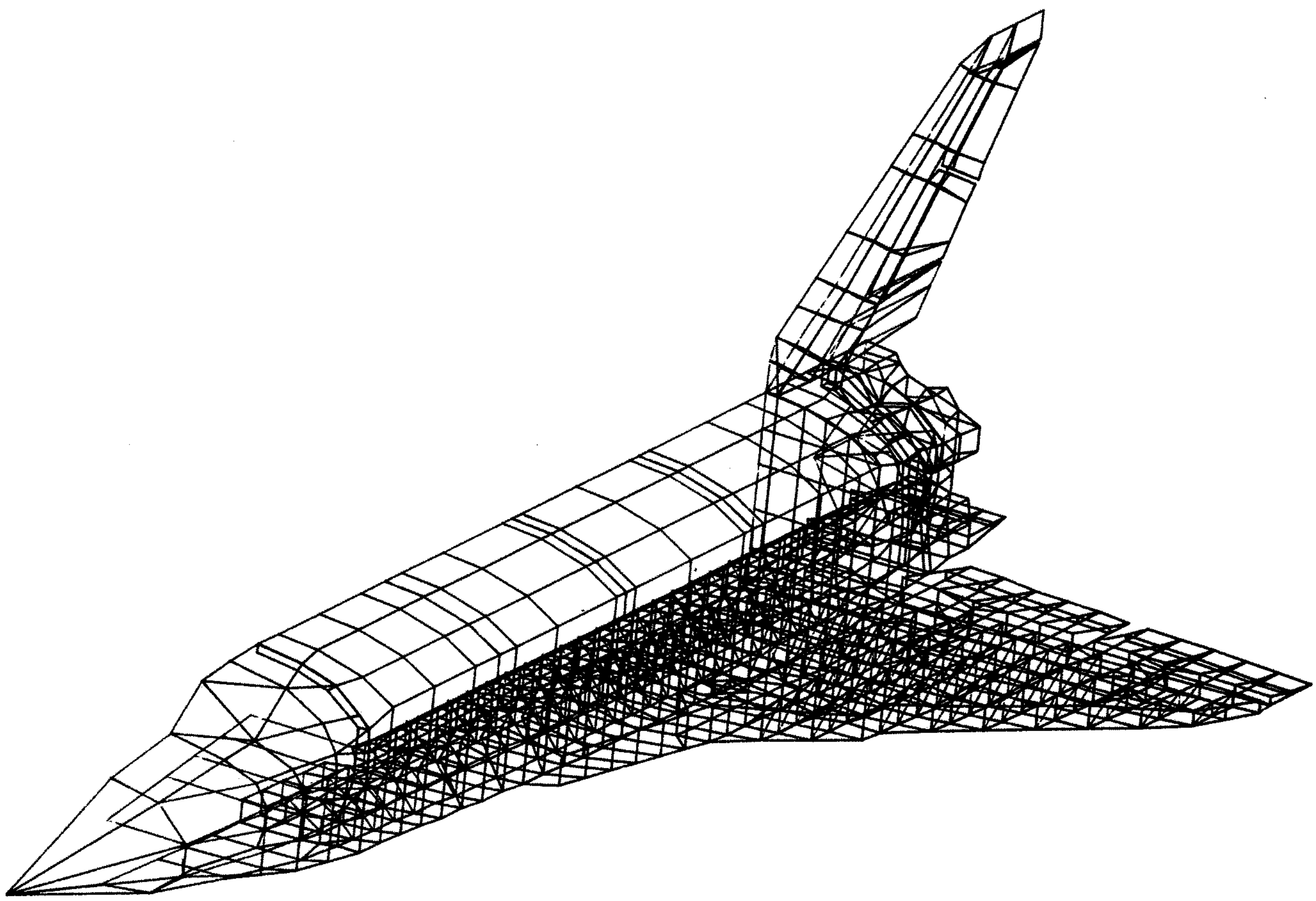

Fig. 8

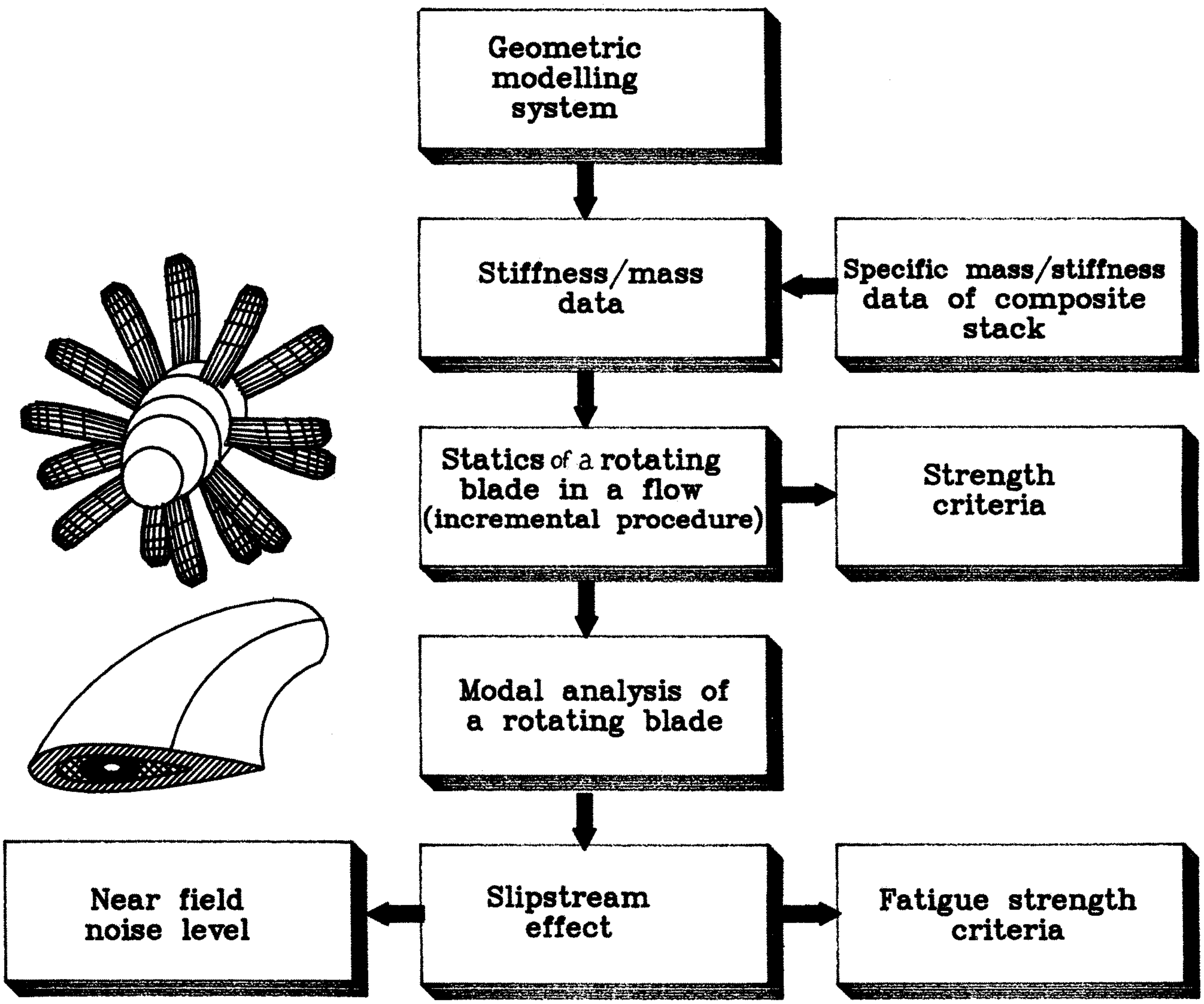

Fig. 9

During several last years, much attention is being paid to developing the economic prop-fans equipped with swept blades made of composites. Compared to the usual application of the FEM, the problems involved require accurate consideration of nonlinearity in deformational responses. Thus, design of the structures mentioned must include an integrated analysis of (a) a blade/flow interaction, (b) the structural strength,(c) the life-time available, (d) a noise level, etc. (Fig.9). For doing this, an isoparametric curved discrete beam element has been developed allowing us to accurately analyze the geometrically nonlinear deformation and the transverse shear strains. The element permits users to introduce cross-sections of arbitrary shape/topology, to assume an unrestricted warping, to compute large rotations/displacements, and to take an account of a material anisotropy. Here, the deformed state is analyzed on the basis of the updated Lagrange formulation:

$$(K_L + K_\sigma + K_p)^{(n)}\,{}_\Delta U^{(n+1)} = P^{(n)} - S^{(n)},$$

where, for nth iteration, $K_L{}^{(n)}$ is the linear stiffness matrix, $K_\sigma{}^{(n)}$ is the geometric stiffness matrix, $K_p{}^{(n)}$ is the external displacement-dependent force stiffness matrix, $P^{(n)}$ is the vector of external nodal forces, $S^{(n)}$ is the vector of internal nodal forces.

For solving the dynamic problems (the aeroelastic stability evaluation, the amplitude-frequency curve prediction, the analysis of stress levels under oblique stream conditions), equations of small vibrations are applicable after a linearization at the static equilibrium conditions. Here, an additional geometric stiffness due to the equilibrium state stress can be taken into account. Thus, one can write the following matrix equations:

$$M\,{}_\Delta\ddot{U} + (C+D)\,{}_\Delta\dot{U} + (K_L + K_\sigma + K_p + B)\,{}_\Delta U = {}_\Delta P$$

where M is the mass matrix, C is the gyroscopic matrix, D is the aerodynamic damping matrix, B is the aerodynamic stiffness matrix, ${}_\Delta P$ is the external excitation vector.

Based on the approaches outlined, the first version of the composite-material swept blade design system is implemented. An experience of using this system have shown this line of inquiry to be promising and further developments to be necessary.