ICAS-84-4.7.2

COMPUTERIZED METHODS FOR ANALYSIS AND DESIGN OF AIRCRAFT STRUCTURES

B. Fredriksson
SAAB-SCANIA AB
S-581-88 Linköping, Sweden

ABSTRACT

The developments made to utilize computerized methods for analysis and design of aircraft structures are described. The paper discusses the integration of data and methods between geometry, aerodynamics, loads, structures etc to form a computer aided engineering environment. The paper is concentrating on structural analysis and sizing. Methods to rationalize the finite element internal loads calculations and to increase the quality of the work is discussed. Methods have been developed to generate local spectra from the finite element result for fatigue and fracture analysis. The rationalization and quality effect of this is discussed.
New design criteria and requirements of high performance,light weight and economy implies introduction of new methods for analysis. The paper discusses the developments made in some advanced topics like combined contact and crack problems and structural optimization.

The combined use of mini- and supercomputers as well as the twofold effect of rationalization and quality by using supercomputers are discussed.

I. INTRODUCTION

Computer capacities available today make it possible to utilize numerical methods efficiently. Computer software are also developing with increasing intensity. Systems for advanced nonlinear analysis and for Computer Aided Design (CAD), Computer Aided Manufacturing (CAM) and Computer Aided Engineering (CAE) are rapidly introduced on the market. To obtain maximum benefit in the use of such systems great care should be taken to system solution. It will effect the whole organisation and the way of working.

New design criteria like damage tolerance design requires introduction of new methods both in stress analysis and fracture mechanics. All new methods are well suited for programming and many require high computer capacity. Requirements of high performance, light weight and economy implies both introduction of new material like fibre composite and introduction of new methods like structural optimization.

To be competitive when introducing a new product it is necessary to be able to develop it in a short period of time. To this purpose computerization of methods for the development could be very beneficial. The effect of computerization and the use of high performs computers is twofold. First it will increase the quality of the work. More detailed studies could be done at an early stage in the project and it is possible to utilize more advanced methods. Secondly it will rationalize the work. Even if more detailed studies are made it is done in a shorter period of time. It is however important to remember that the computerization will generate a lot of more information that should be handled by the engineer.It is thus important that the system or systems include efficient data processing facilities. The utilization of computer graphics is very important.

The following example from internal loads analysis and sizing of an aircraft indicate the rational for developing computerized methods. For internal loads analysis today the finite element method is almost exclusively used. Table 1 is an example of a rough estimate of the number of data involved. The engineer has to search in this data for all loadcases to find the envelope of sizing values. The number of points for sizing varies of course very much. For medium sized transport and military aircraft the range might roughly very as indicated in Table 2. For all these points the engineer need the stresses and the local spectrum to be able to size for static loads, fatigue and fracture.

The paper dicusses the above mentioned topics and the effort made to utilize computerized methods and high performance computers.

FE- Model ——— Some data

Model		
Nodes	Elements	Unknowns
10 000 – 50 000	10 000 – 40 000	50 000 – 150 000

Result / Loadcase		
Stresses	Reactions	Displacements
30 000 – 100 000	2 000 – 6 000	50 000 – 120 000

Number of loadcases	
Total	Active for sizing/point
50 – 500	5 – 10

Table 1 Rough figures for typical aircraft internal loads model.

Sizing —— Some data

Number of points		
Statics	Fatigue	Fracture
5 000 – 15 000	5 000 – 15 000	100 – 500

Table 2 Rough estimates of number of points for sizing in an aircraft.

II. INTEGRATION AND COMPUTER AIDED ENGINEERING

Computer Aided Design, CAD is today a rather well establish method and a lot of software exists on the market. The definiton of the term CAD varies however a lot. The CAD systems have mainly been drafting systems and facilities for the broader term design has not generally been included. Today with efficient geometry systems and possibilities for geometric modelling the systems are rather efficient also for design. Including in the system also computerized methods for stress analysis and sizing the design part is rather complete. Such a system is sometimes also called a Computer Aided Engineering, CAE-, system. To be complete such a system should also include the project study phase and the manufacturing part as shown in Fig 1. Involved in such a system are different types of computers from mini- to

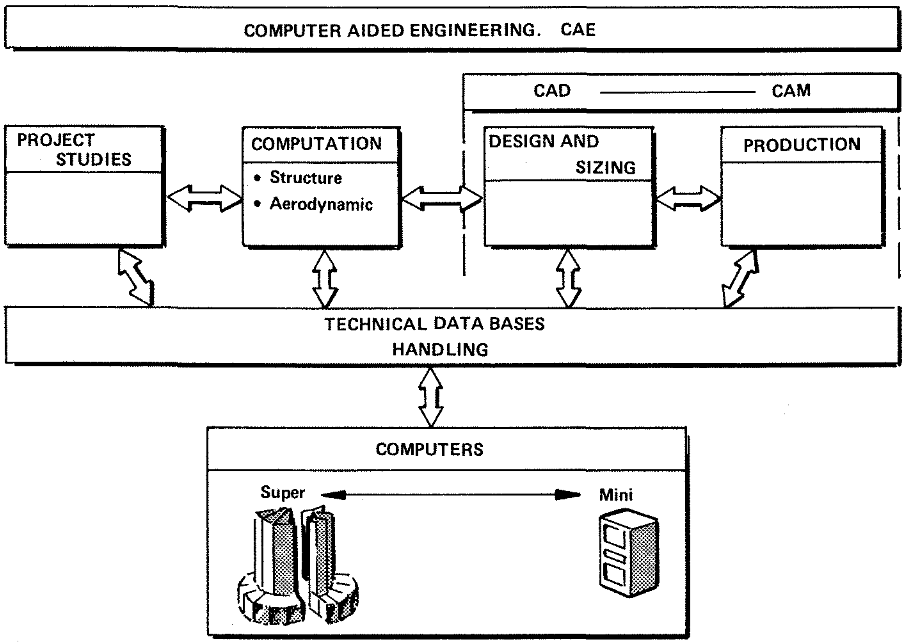


Fig 1 Illustration of CAE.

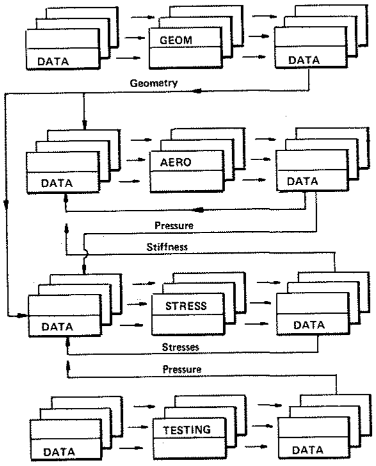


Fig 2 Example of data flow.

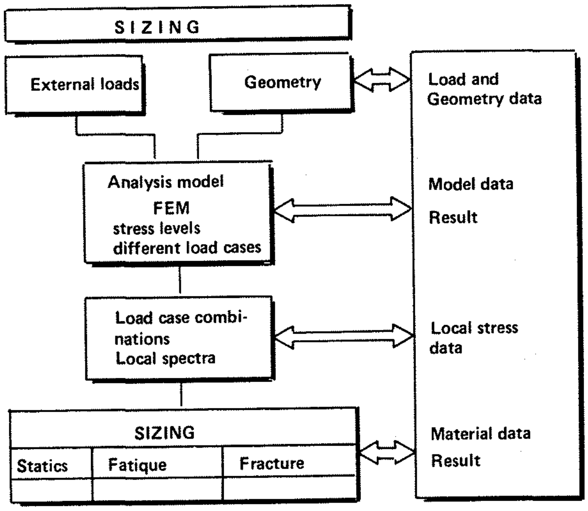


Fig 3 The sizing process.

supercomputers. A lot of data should be communicated between different databases and different computers. When designing such a system much care should be taken for an efficient solution of data communication both from a speed and safety (reliability and responsibility of data) point of view.

At Saab-Scania rather much effort is made to obtain an efficient solution of the engineering work by computerization of methods. The present paper is concentrating on the computerization of the structural analysis and sizing part. See Fig 1.

The basis for the sizing work is the finite element (FE-) model for internal loads analysis. The FE-model need geometry information from the geometry system and loads from aerodynamic calculations and/or wind tunnel measurements. It is a lot of information to pass between the different disciplines and integration of this process could be very beneficial. An example of data flow is shown in Fig 2. Developments have been made to facilitate this communication and examples of applications will be discussed in more detail in the next section. For the datacommunication and handling we are using the relational data base system (RDBS) ORACLE. The RDBS has its main advantage in the flexibility and ease of defining data structures and relations. The main disadvantage is the inefficiency of handling large sets of data like for instance flexibility matrices that should be communicated between stress and aerodynamics. For instance a matrix data type is presently missing. We have introduced the data type FILE which means that the large data sets are on files and the RDBS could be used to administrate the files. We are then missing however the relational search possibility of the RDBS. For reasons of efficiency the information should not be to much centralized. The data should be supplied with information enough so the receiver understands and can use the data. This is easier done for local than for central databases.(1).

The integration described above is between the different disciplines like geometry, aerodynamics, loads, stress etc. On the lower level integration and computerization of the sizing work has been made and is still going on. The principle of the system developed is described, in Fig 3. The basis for the sizing work is the FE-model for internal loads calculations which need the communication of data on the level described above. In the sizing work the data from the FE-model is used in all the different sizing programs for statics, fatigue and fracture. Systems have been developed to communicate this data efficiently and new methods for local spectrum generation from FE-results have been developed. Applications of this will be presented in detail in section V.

III. INTERNAL LOADS CALCULATION

For internal loads calculations almost exclusively the finite element method is used today. The models are getting more and more complex and thus resulting in more detailed information. See Table 1. For analysing a complete aircraft it is a necessity to utilize the substructuring technique. The reason is both computer efficiency and flexibility in work making it possible to easily work groupwise in parallell to develop the model. It is also beneficial in joint projects where different structural parts are modelled by different companies. Figure 4 shows the substructuring hierarchy of a typical aircraft FE-model and Fig 5 shows the FE-model of the JAS-aircraft. This figure indicate the need for pre- and postprocessing and efficient tools for computer graphics. For efficient modelling it is a necessity to communicate geometry and other data from the CAD-system to generate the mesh. It is important to have automatic procedures for generating the FE-loads from dynamic and aerodynamic analysis and wind tunnel testing and methods of introducing concentrated masses from systems and equipments. When the FE-model for hundreds of loadcases is solved the engineer needs a lot of tools for operating efficiently on all these data. He needs for instance software to search for extremes to find the envelope for sizing. He needs software to integrate stresses to obtain crossection forces for sizing. Software and hardware to present data graphically etc. Developments of a series of such software have been made. It is not possible to describe all and I will concentrate on some of the important ones.

One very heavy part in FE-modelling work is the generation of consistent nodal forces for pressure and acceleration loads. The pressure distribution is obtained from wind tunnel tests and/or from aerodynamic calculations. The pressure values obtained then is not obtained in the same points as the FE-nodal points. For reasons of flexibility in work and also for technical quality reasons it is important that this is not required. This means that to calculate the consistent nodal forces requires interpolations to finite element nodes or integration points. To this purpose a software for interpolation in surfaces was developed(2). The principle is briefly discribed in Fig 6. The pressure distribution from wind tunnel testing and aerodynamic calculations are given in defined points referred to specified lines. The data for each point is the x, y, z-coordinates and the pressure p. With the program INTIM it is now possible to define a new surface mesh with defined new points, MP and lines, ML. This results in a topologically rectangular mesh. The pressure and coordinate information from the original mesh is now used to interpolate for pressures in all the new points. The pressure and coordinate values of this mesh is now used to interpolate the pressure to the finite element integration points. For each integration point the program search for the neighbouring pressure points in the pressure surface to define the element to which the integration point belong. The pressure in the nodal points of this element are than used to interpolate to the integration points. Different numbers of integration points (from 3 x 3 to 10 x 10) could be used. The large number of integration points is used in coarse

FE-meshes where the pressure varies a lot over the finite element. Resultant force and its centre of action is calculated for control checking. The result is now input for the finite element program.

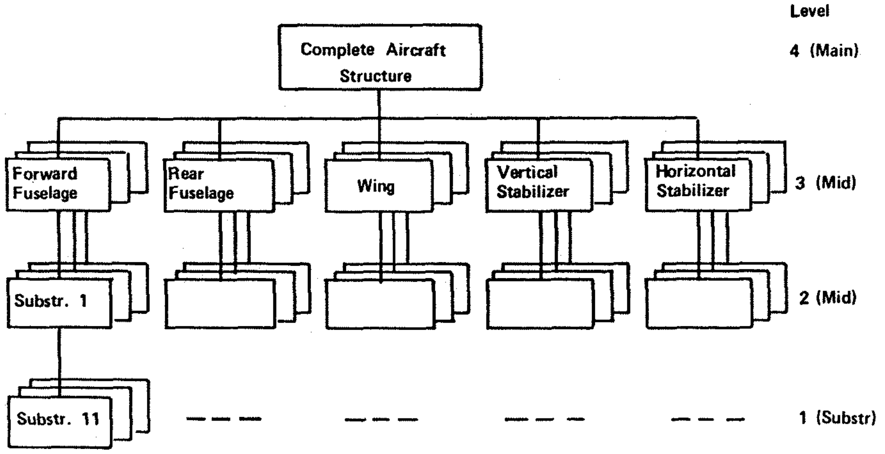


Fig 4 Substructurering hierarchy for an aircraft FE-model.

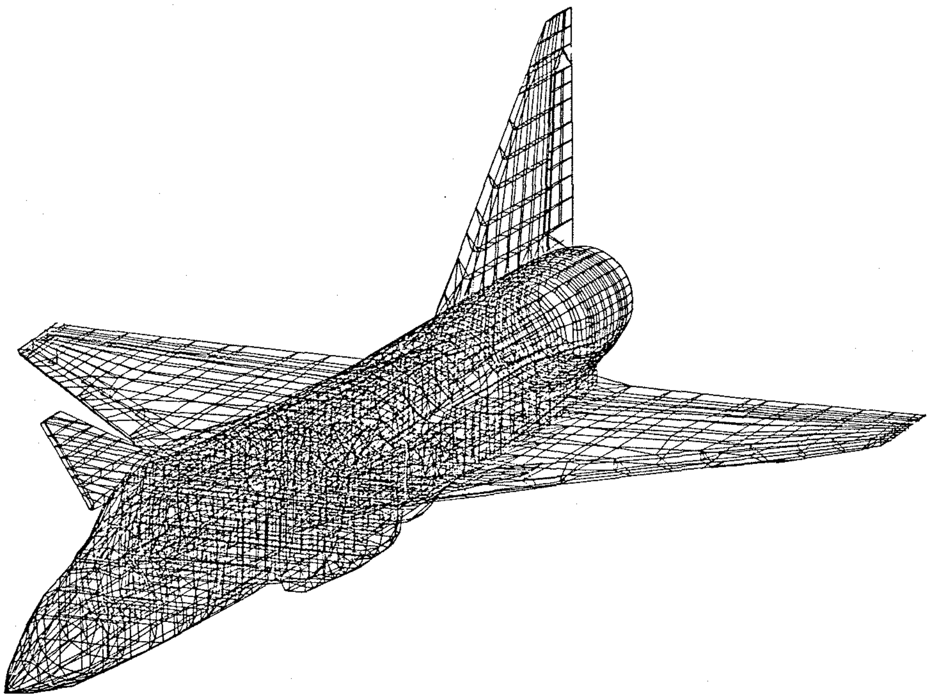


Fig 5 FE-model of JAS-aircraft.

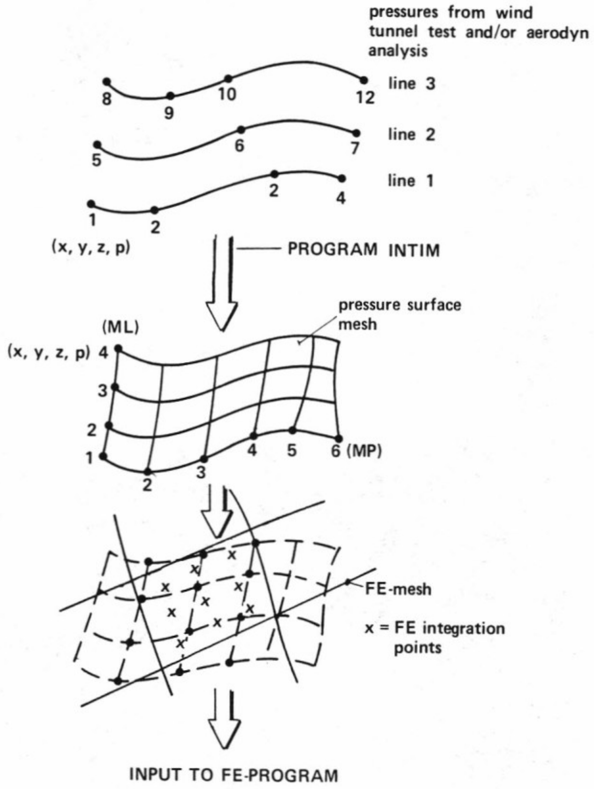


Fig 6 Interpolation and consistent FE-nodal forces.

Fig 7 shows a pressure surface for a wing part of the Saab Draken aircraft. The benefit of using this software to generate the forces is twofold. It will save a lot of manpower and time. It will also increase the quality of the work because it makes it possible for the engineer to include more loadcases then was previously possible in the same period of time. The timesaving is very high. As an example, to generate the pressure loads for the SF340 model (Approx 140000 degrees of freedom) took one manmonth per loadcase when not using this software. For the JAS FE-model we are utilizing this software and are able to generate the FE-loads for say 150 loadcases in less than 4 days.

To generate the inertia loads from dynamic analysis is even more timeconsuming and more difficult than the pressure loads. In an early stage of an aircraft project the dynamic characteristics of the aircraft is simulated with a beam model which might be successively refined with finite element model parts. Software has been developed to transform the accelerations from this model to the finite element nodes. Once the accelerations in the nodes are known it is with some modifications possible to utilize the same software as described above to compute the consistent intertia forces. Pressure surfaces are analogous to acceleration surfaces. There are however generally six acceleration components at each node. In addition to this way of computing dynamic loads it is also possible to use the complete FE-model and solve the dynamic eigenmodes. This is of coarse the most correct way of doing it and with an efficient eigenvalue solver and high performance

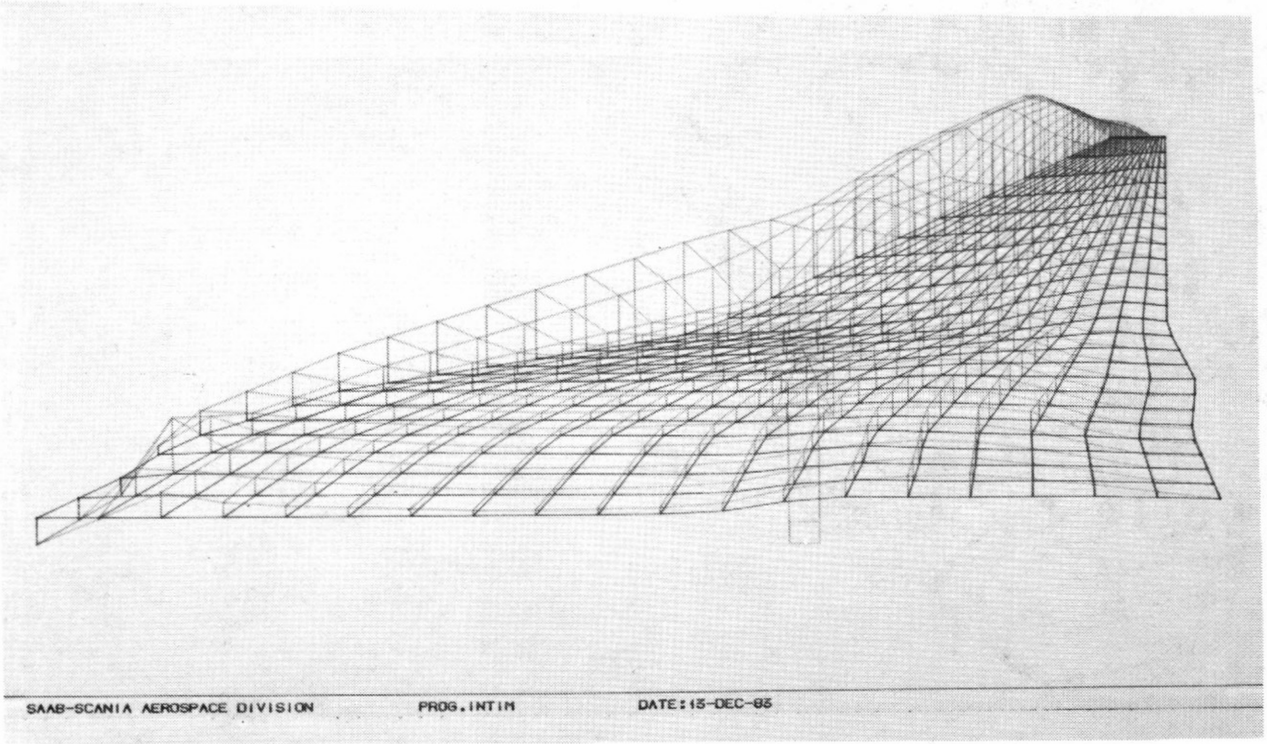


Fig 7 Pressure surface.

supercomputers it is fully possible to do so. It is presently done at Saab-Scania with the ASKA system on the CRAY1 computer. The Lansczos method for eigenvalue solutions are presently implemented in the ASKA program.

For aeroelastic analysis and also for optimization studies where aerodynamic efficiency is a constraint (See section IV) the flexibility of the structure is needed. To this purpose software has been developed$^{(2)}$ for generating flexibility matrices from the FE-model but in any specified mesh. The same basic software for interpolation as described above for pressure loads is used.

Utilization of computer graphics is important for beeing able to efficiently working with the FE-results. It is necessary to have software to search for extremes and envelopes among the large number of loadcases and also to present integrated results such as moment and shear force distributions on specified parts such as stringers, spars etc. At Saab-Scania Brandt and Person have developed software for colour graphics for presenting FE-models including superposition of results. It is based upon raster technique and presents an efficient way of utilizing hidden surface removals. On the models the results are plotted as isosurfaces and/or as information in vector form. Fig 8 shows the FE-model of the cockpit of SF340. Fig 9 shows the isostresses for a thin shell structure. The development of computer graphics for efficient handling and operation on data in databases is continuing and is judged to be very important for much of the stress work both in finite elements and in sizing work using comperterized methods. For further information on software for increased quality and rationalization of work in internal loads calculations the reader is reffered to Brandt$^{(2)}$.

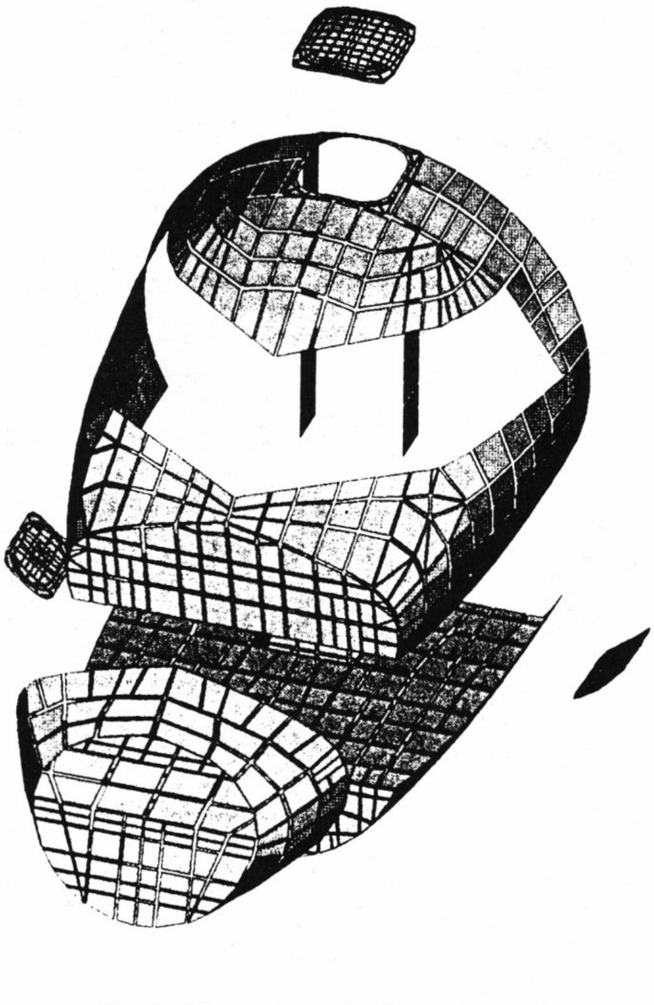


Fig 8 FE-model of SF-340 cockpit part of fuselage. Reproduced in grey-scale.

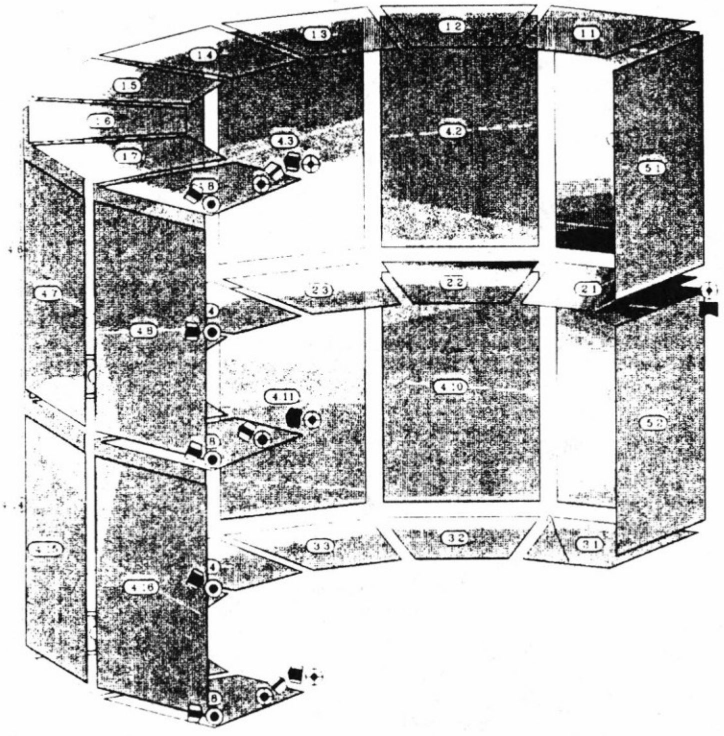


Fig 9 Stresses on FE-model reproduced in grey-scale.

IV. STRUCTURAL ANALYSIS - ADVANCED TOPICS

During the latest years the application of advanced computational methods have been made practically available through the high performance mainframe - and supercomputers. This has made it possible to perform for instance bird impact studies, nonlinear buckling studies, dynamic analysis of large models, elastoplastic analysis, contact and friction analysis, combined contact and crack analysis, structural optimization etc. At Saab-Scania we have developed and are developing methods making it possible to perform these types of advanced studies. Methods of contact and friction and combined contact and crack problems and methods for structural optimization will be discussed in some further detail.

Contact, Friction and Cracks

In aircraft structures joints of different types like bolted and riveted joints, lugs and bonded joints are important. The application of the new damage tolerance criteria, Mil A-83444 implies among other things study of combined contact and crack problems.

Globally it is important to obtain information about the stiffness of a joint in order to be able to use the right stiffness in the FE-model for internal loads distribution. We have for this purpose implemented a "finite joint element" making it possible to introduce stiffness coefficients from experimental measurements. The element is presently linear. It should be observed that the intrinsic stiffness introduced by the finite element mesh which varies from application to application should be taken into account(4) (5). Using the software for contact and friction studies developed at Linköping University we can analyse a single joint in detail to simulate the nonlinear behaviour when cycling the load(7). An example study is presented in Fig 10.

When studying in detail for instance lugs and bearings(8) (9) the contact problem is locally important to solve for correct stress distribution. This is also important when applying the damage tolerance design criteria for instance to different types of lugs where the redistribution of contact pressure during crack propagation influence the propagation rate. It is wellknown that coldworking might be beneficial for fatigue properties and coldworking of holes is used to increase the fatigue life of for instance lugs. To be able to quantify this effect an analysis tool for elastoplastic contact problems is needed.

The contact and friction problems presents a new class of boundary conditions which implies inequality constraints. The friction problem presents mathematically very difficult problems for which existence and uniqueness has not been generally proved. In recent years a lot of research work has been done on contact and friction problems. Please refer to Panagiotopoulos(10), Fredriksson et al(11) (12), Oden and Kikuchi (13) and Torstenfelt (14).

Considering two elastic structures A an B contacting each other we obtain contact boundary conditions which involves inequalities. The inequalities are introduced because points on either A or B approaching each other are not allowed to penetrate into the contacting material. At subsequent stages it is however possible that the contact is lost, if tensile stresses occur and there is no adherence capacity at the contact surfaces. If there for instance at the unloaded state is an initial gap h(x,y) or interference (negative gap) the differential normal displacement in A and B should satisfy

$$u_n^A(x,y) - u_n^B(x,y) \leq h(x,y) \text{ on } \Omega_c \qquad (1)$$

Ω_c = contact surface (unknown)

In addition the contact problem involves the equality constraints of prescribed displacements u_i and loads p_i. The solution of the elastic contact problem could be obtained from the principle of minimum potential energy Π with inequality constraints(15).

$$\text{Min } \{\Pi = \int_V (\tfrac{1}{2}\, \sigma_{ij}\, \varepsilon_{ij} - X_i\, u_i)\, dV - \int_\Omega p_i\, u_i\, d\Omega \qquad (2)$$

$$u_i = \bar{u}_i \text{ on } \Omega_u$$

$$u_n^A - u_n^B \leq h \text{ on } \Omega_c$$

σ_{ij} = Stress tensor, ε_{ij} = Strain tensor,

X_i = Body forces

u_i = Displacement vector, p_i = Surface loads

$V = V_A + V_B$ = Volume occupied by the structures,

Ω = Surface of the structures

Introducing the FE-formulation we obtain in matrix notations.

$$\text{Min } \{\Pi = \frac{1}{2}\ \underline{u}^t\ \underline{K}\ \underline{u} - \underline{P}^t\underline{u} \ \left|\ \begin{array}{l} \underline{u} = \overline{\underline{u}} \text{ on } \Omega_u \\ \underline{A}\ \underline{u} \leq \underline{h} \text{ on } \Omega_c \end{array}\right. \} \qquad (3)$$

This constitutes a quadratic programming problem with linear inequality constraints. The solution to this problem could be obtained from FE-programs by adding a module that automatically search for which contact nodepairs that are active in Equation (3). The conditions for checking whether the nodes are in contact or not are either the inequality condition in Equation (3) or if tensile normal stresses occur. Each changes of status means a transformation of the stiffness matrix and could be considered as a variable multipoint constraint transformation as described by Torstenfelt (14).

When studying frictional problems it is necessary to introduce a slip criterion and a slip rule to simulate the constitutive behaviour at the contact surface. Consider the contact stress space p_x, p_y and p_z. $p_z = p_3$ is the normal contact stress

Introducing the slip surface g for Coulomb friction we obtain

$$g\ (p_x, p_y, p_3) = (p_\alpha\, p_\alpha)^{1/2} + \mu p_3, \quad p_3 < 0 \qquad (4)$$

Slip could take place for g = 0.

Including friction means complex nonlinearities including for instance nondifferentiable terms when Coulomb friction is studied. The reader is refferred to Fredriksson et al (11), Oden and Pires(15) and Campos et al (16).

Stress intensity factor calculations by using the FE-method could be done in a number of ways. A detailed study and evaluation of different methods has been performed by Bartelds and de Koning,(28). An overview of methods and software is given by Fredriksson and Mackerle(29). An attractive and also accurate method is to use the relationship between the stress intensity factor K and the energy release rate G at a virtual crack extension.

$$K_1^2 = \frac{E}{1 - n^2} G \qquad (5)$$

n = ν (Poisson's ratio) for plane strain, n = 0 for plane stress.

Equation (5) is valid for mode I dominated cracks. At mixed modes it is not obvious how to separate the contributions from the different modes. The energy release based on the contribution from the different displacement components could be used. The method is applicable also to threedimensional cracks. Which value of n to be used in Equation (5) is then depending on where on the crack front the K-value is to be calculated. If the crack is closing, terms are added in Equation (2) due to contact and friction,(30). The calculation of G by using the FE-method is efficiently done by virtually shifting the nodes on the crack front. This shift then corresponds to a change ΔA in crack surface. The virtual shift of the FE-nodes means a change $\Delta \underline{K}$ in stiffness matrix. In matrix notations we then obtain from the FE-calculation.

$$G = \frac{1}{2} \underline{u}^t \left(\frac{\Delta \underline{K}}{\Delta A}\right) \underline{u} \qquad (6)$$

$\underline{u}$ is the displacement vector from the FE-solution prior to the virtual shift of the crack tip node. At the shift, only $\Delta \underline{K}$ has to be computed.
The methods for contact, friction and crack problems described above have been implemented in the ASKA FE-system. Fig 10 is an application on contact and friction. In Fig 11 the FE-model for stress intensity factor calculation of a threedimensional crack in a wing rudder attachment is shown. This includes also the analysis of the contact problems between lug and bearing and in the bearing.

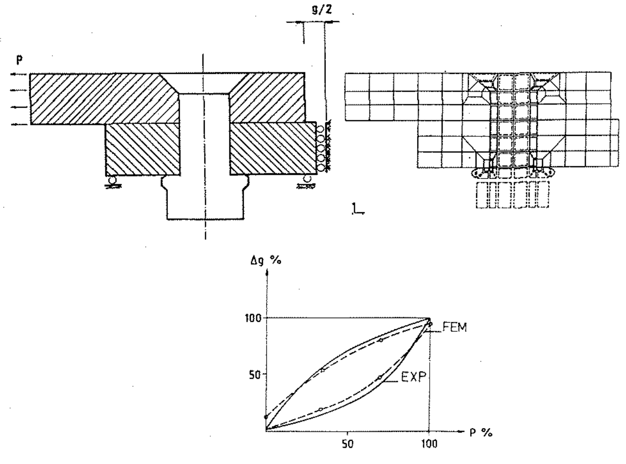


Fig 10 Bolted joint in single shear.

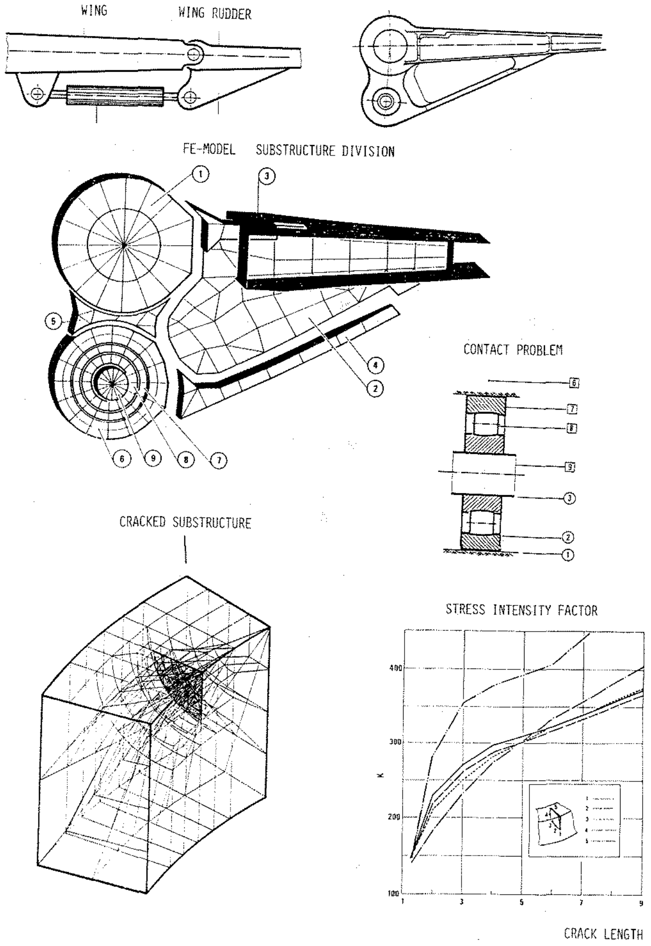


Fig 11 FE-model and stress intensity of a cracked rudder attachment lug.

The Boundary Element Method, BEM

In a cooperation with Linköping University the author has together with Andersson studied the BEM and especially the applicability to contact and friction problems (17) (18). It has its special merits one of which is the need to model only the boundary. This is especially attractive in contact problems where the nonlinearities takes place at the boundaries. A disadvantage is the nonsymmetry and nonbanded character of the matrices involved. Fig 12 shows a contact and friction study of a riveted joint. The problem is simplified to study a twodimensional state which means that no bending is assumed.

Structural Optimization

Requirements of high peformance and economy implies high utilization of materials and optimal design solutions. Structural optimization has become practically available during recent years and are applied more and more frequently in the design process (19-24).

One of the reasons for this is the high performance of the mainframes and supercomputers. An optimization study takes 5-100 times more computer time than a linear analysis of the same model. The dominating analysis tool in an optimization system is the FE-method.

We are presently in a program together with the Aeronautical Institute in Stockholm developing an optimization system. We are using the basic optimization routines developed by Fleury (19). The problem is mathematically defined as follows:

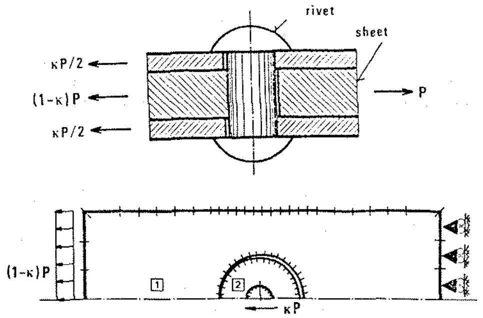


FIG 18.

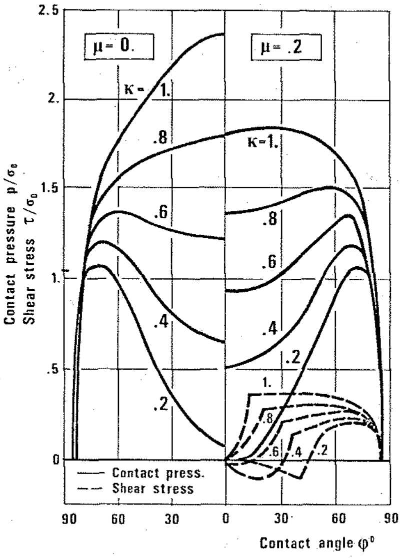


Fig 12 Riveted joint and BE-solution to the contact and friction problem.

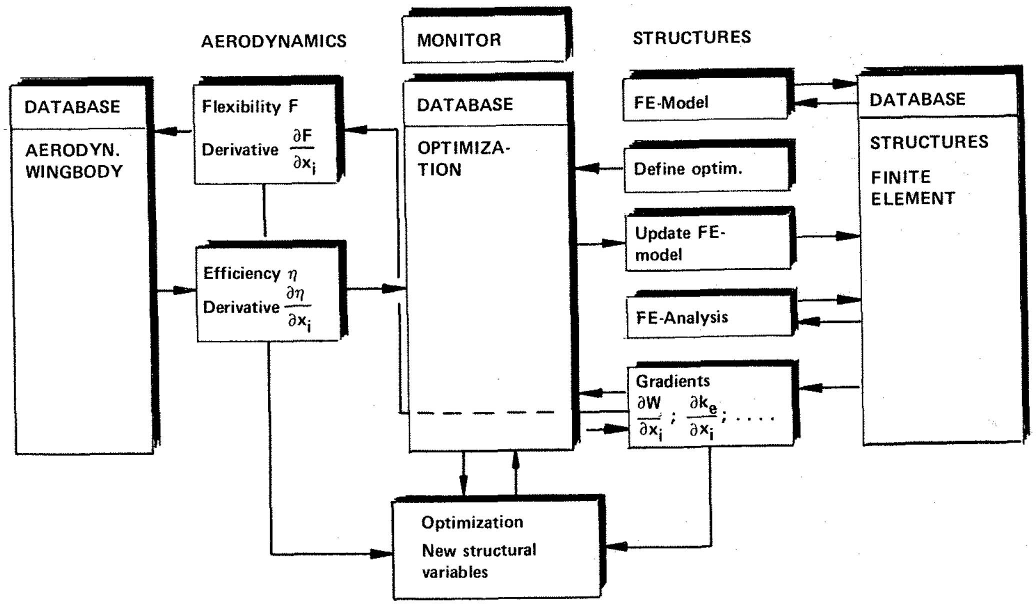


Fig 13 Optimazation system - brief flow chart.

Minimize the weight $W(x_i)$ which is a function of different design variables x_i such as thickness t, fibre direction ϕ in composite materials, coordinates c, crossectional area A etc. The minimization is subject to constraints on stresses or combinations of stresses f (σ) (Tsai, von Mises etc), displacements u, differential displacements Δu, flutter speed control surface efficiency η etc. In addition there must be side constraints which for instance due to practical manufacturing and handling reasons limit the design variables.

$$\text{Min } \{W(x_i) \left| \begin{array}{l} u(x_i) \leqq u_{max} \\ f(\sigma(x_i)) \leqq \sigma_{max} \\ \eta \geqq \eta_{min} \end{array} \right\} \qquad (7)$$

with side constraints

$$t_{min} \leqq t \leqq t_{max} \qquad (8)$$

$$\phi_{min} \leqq \phi \leqq \phi_{max}$$

The constraints on flutter speed and control surface efficiency implies integration of structural, aeroelastic and aerodynamic analysis. It is required that the software is modified to deliver the appropriate derivatives or gradients needed by the optimization routines to find the minimum. See Fig 13.

Generality of the methods requires the use of the FE-method as the analysis tool. To be practically useful it is important to be able to apply the optimization to models of different finite elements. One heavy part in developing an optimization system is to generate derivatives or gradients. This could be done in two ways, either analytically ones and for all by the developer and then use the formulas in the optimization program or numerically by the program. For instance the derivative of the element stiffness matrix k_e with respect to the design variable x is computed as

$$\frac{\partial k_e}{\partial x} = \frac{k_e(x + \Delta x) - k_e(x)}{\Delta x} \qquad (9)$$

To derive the derivatives analytically is difficult generally and very timeconsuming. Ones it is done it might however be computationally more efficient than deriving the derivatives numerically. It is however more beneficial to be able to use a large number of different elements in the optimization model. The method of numerically deriving the gradients makes it easy to include more elements. Totally this is than a more attractive way. The optimization system is at Saab-Scania developed by Brandt and Bråmå(26). We decided at Saab-Scania to use the method of nummerically deriving the gradients and had Esping(25) to develope the routines. Early tests also indicate that numerical stability is achieved and small enough increments could be used. Long enough computer wordlength is important. It is also shown practically that new elements could be implemented in the system in a number of hours rather than days(26).

When including composite fibre angles and coordinates for moving for instance stringers and spars as design variables the problem becomes nonconvex and the solution does not quarantee global minimum. What could be done then is to use a series of different initial states and take the lowest weight solution. If the same solution is obtained for different inital states the problem is probably convex and the minimum global.

Presently the optimization system is tested and continously developed. Included in the system are presently bar, membrane and 3D solid elements. Design variables are thickness, crossection area, coordinates and fibre angles. Constraints are displacements, differential displacements, different functions of stresses and strains. Linked variables could be used. The module for aeroelastic derivatives is presently developed at the Aeronautical Institute. This makes it then possible to use aerodynamic efficiency requirements as constraints.

V. FATIGUE AND FRACTURE

Sizing against fatigue and fracture are two of the most important objectives for the aircraft stress engineer. The basis for this work in addition to methods and data for fracture and fatigue analysis are the global load sequence and the finite element stress result. See Fig 3. One timeconsuming part of this work is to generate the local spectrum for the detail studied. From the global loadsequence a local sequence should be derived and from this a local spectrum based on some counting algorithm, e.g. Rain Flow Count. The local sequence is generated using the FE-element solution from the different loadcases included. The method of generating a local spectrum is very well suited for computerization. It will both rationalize the work and increase the quality. At Saab-Scania Persson(27) developed a method of generating the local spectrum from a global sequence. See Fig 14. This software makes it possible to rapidly generate a local spectrum for any detail studied. It means that more correct spectra could be generated for the details studied. It should be emphasized however that the stress result from the FE-solution might fluctuate and it might be better to use integrated values like moments and shear forces which is also available in the system. The rationalization effect of this software is extremely high. To generate a local spectrum manually takes some, say two weeks and using the present software takes, including input data some minutes.

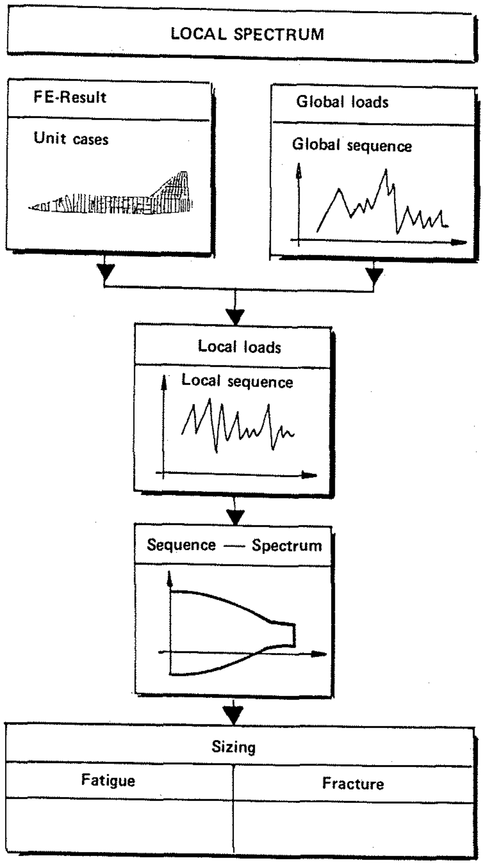
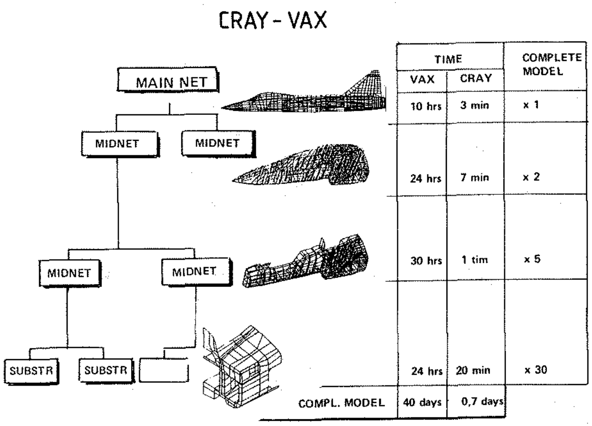


Fig 14 Local to global spectrum.

VI. MINI- AND SUPERCOMPUTERS

Computerization of the sizing and design work requires both high computational performance and a user oriented way of working with the computer. The combined use of minicomputers with there useroriented operating system for interactive work and supercomputers with high computational performance present a good solution to maximize the outcome of the computerization. At Saab-Scania we are for technical computations using this hardware configuration. The most heavy part in the process is the solution of the FE-model. The timesaving when using the CRAY1 computer as compared to the VAX/780 computer is very high. The multilevel substructure model (Fig 5) of the JAS Aircraft has been run on both these computers. On a VAX computer this model is solved successively in a series of substructure runs and it is also possible to use several VAX computers. If we sumarize the execution time (wall clocktime) for this model in our normal work load environment we obtain roughly the figures as given in Fig 15.



	TIME		COMPLETE MODEL
	VAX	CRAY	
	10 hrs	3 min	x 1
	24 hrs	7 min	x 2
	30 hrs	1 tim	x 5
	24 hrs	20 min	x 30
COMPL. MODEL	40 days	0,7 days	

Fig 15 Computer wallclock time - Comparision CRAY-VAX.

VII. CONCLUSIONS

Computerization of the sizing and design work has a twofold effect. It increase the quality of the work and it will rationalize the work. These are two important factors necessary to meet the requirements of high performance, economy and rapid project development. In addition to speeding up the work it increase the quality because new theoretical and numerical methods are made practically available. The speeding up of the work indirectly means also an increase of quality because more detailed studies could be made. Examples are the use of automatic procedures for aeordynamic and inertia loads and local spectra generation discussed in this paper. The computerization makes it possible to include loadcases that previously was impossible to include because of for instance project time schedule and economy reasons.

To be able to perform a high quality work at an early stage of the project is economically important. It should mean less changes at a later stage. If the stress engineers are able to do both static, fatigue and fracture sizing before signing the drawing this should imply less changes and the production cost should be lowered. Assume for instance that there are a design change which means that the FE-model has to be completely rerun. It is easy to understand what the timesaving as shown in Fig 16 means. As seen in Fig 17 it should also be easier to fulfill the timeschedule requirements for signing the drawings when using for instance the software for local spectrum generation.

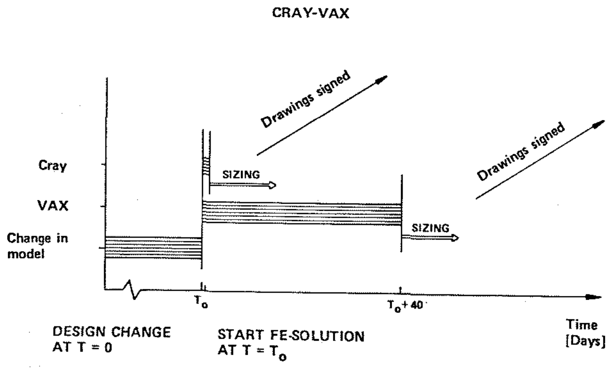


Fig 16 Effect of using the CRAY for FE-analys.

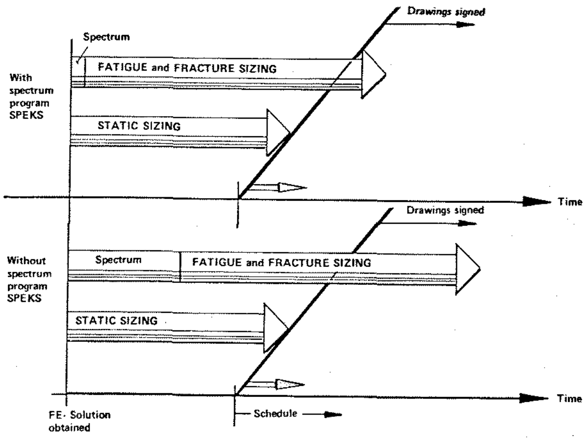


Fig 17 Effect on using software for local spectra generation.

ACKNOWLEDGEMENT

This review paper presents work going on at Saab-Scania, Aircraft Division. The work done is a result of a joint effort and all contributions are greatly acknowledged. Most of the work has been done at the Structural Analysis and Computer Programs group headed by Mr Jan Brandt. Mr Brandt and his coworkers Bo Persson, Stefan Carlsson, Torsten Bråmå, Jan-Erik Thomasson and Göran Rydholm are greatly acknowledged. On optimization the cooperation with Professor Börje Andersson at the Aeronautical Institute in Stockholm is also acknowledged. Part of the work presented has been financially supported by the Swedish Air Materials Administration and this is greatly appreciated. I am also greatful to Drs Bo Torstenfelt and Torbjörn Andersson at Linköping Institute of Technology, Division of Solid Mechanics for their cooperation on contact and friction and on the boundary element method respectively.

REFERENCES

(1) Langefors B., Theoretical Analysis of Information Systems, Studentlitteratur 4th ed. 1973, Sweden.

(2) Brandt J,. ASKA and Other Software Available at Saab Aerospace FEM Group. Internal Report KHS-83.38 Saab-Scania, Aircraft Division Linköping Sweden, 1984.

(3) Persson B., Brandt J. FEM-Presentation with Multicolour Raster Technique (in Swedish) Saab-Scania AB, Aircraft Division, Linköping, Sweden, 1983.

(4) Brandt J. Calculation of Force Distributions and Stress Distributions of Single Shear Single Row Joints Subjected to Tension or Shear (In Swedish). Saab-Scania Report FKH RU-3317, 1980.

(5) Edlund U. Linear FE-Element for Simulation of a Shear Loaded Fastener (In Swedish) Internal Report FKH R-3330, 1981 Saab-Scania AB, Aircraft Division Linköping Sweden.

(6) Torstenfelt B. Contact Problems with Friction in General Purpose Finite Element Computer Programs. Computers & Structures vol 16. 1983. No 1-4, pp 487-493.

(7) Eriksson, S-G. Twodimensional FE-Analysis of a Single Shear Bolt Installation (In Swedish) Internal Report LiTH-IKP-EX-225, Linköping University Linköping Sweden, 1980.

(8) Ansell H., Fredriksson B,. Cracks Interacting with Contact Forces - A Finite Element Study on Loaded Holes. Proc. ICAS Conf. Seattle August. 1982.

(9) Mason P.W., Balderes T., Armen H. The Application of Nonlinear Analysis Techniques to Practical Structural Design Problems. Computers & Structures vol 16, 1980. No 1-4 pp 549-562.

(10) Panagiotopoulos P.D. Nonlinear Programming Approach to the Unilateral Contact- and Friction -Boundary Value Problem in the Theory of Elasticity, Ing. Arch. vol 44, 1974 pp 421-432.

(11) Fredriksson B., Rydholm G., Sjöblom P. Variational Inequalities in Structural Mechanics with Emphasis on Contact Problems Proc. Int. Conf. of Finite Elements in Nonlinear Solid and Structural Mechanics, Geilo Norway, 1977.

(12) Fredriksson B. On Elastic Contact Problems with Friction. A Finite Element Analysis Diss No 6 Linköping Studies in Science and Technology, Linköping University, 1976.

(13) Oden J.T., Kikuchi N. Finite Element Methods for Constrained Problems in Elasticity. Int. J. for Num. Meth. in Eng. vol 18, 1982, pp 701-725.

(14) Torstenfelt B. Finite Elements in Contact and Friction Applications Linköping Studies in Science and Technology No 103, Linköping University, 1983.

(15) Oden J.T., Pires E.B. Numerical Analysis of Certain Contact Problems in Elasticity with Nonclassical. Friction Laws. Comp & Struct. Vol 16, No 1-4. pp 481-485, 1983.

(16) Campos L.T., Oden J.T, and N. Kikuchi. A Numerical Analysis of A Class of Contact Problems with Friction in Elastostatics.Comp. Meth. in Appl. Mech. and Eng. 34, 1982, pp 82-845.

(17) Andersson T., Fredriksson B., Persson BGA. The Boundary Element Method Applied to Two-dimensional Contact Problems Proc. 2nd Int. Seminar on Recent Adv.in Boundary Element Methods, Southamton, England, 1980.

(18) Andersson T. Boundary Elements in Two-Dimensional Contact and Friction Linköping Studies in Science and Technology. Diss. No 85. Linköping Iniversity, 1982.

(19) Fleury C. Optimization of Elastic Structures by Finite Elements. Aerospace laboratory ofthe University of Liege. Report SA-91, 1981.

(20) Ashley H. On Making Things the Best - Aeronautical Uses of Optimization J. Aircraft vol 19, No 1, pp 5-28.

(21) Esping B. Minimum Weight Design of Membrane Structures Report No 83-1. Dept. of Aeronautical Structures and Materials. Royal Inst. of Techn. Stockholm. 1983.

(22) Morris A.J., Bartholomew P., and Dennis J. A Computer Based System for Structural Design, Analysis and Optimization AGARD CP-280, paper 20, 1980.

(23) Schmidt L.A., Miura H. An Advanced Structural Analysis/Synthesis Capability - ACCESS 2. Int. J. Num. Meth. Eng. vol 12, 1978. pp 353-377.

(24) Wilkinson K., Markowitz J. Lerne E., and George D. FASTOP. A Flutter and Strength Optimization Program for Lifting Surface Structures. J. Aircraft, vol 14, 1977, No 6. pp 581-587.

(25) Esping B.J.D. Minimum Weight Design of Membrane Structures Using Eight Node Isoparametric Elements and Numerical Derivatives. Report No 83-5 Dept. of Aeronautical Structures and Materials. The Royal Institute of Technology Stockholm. 1983.

(26) Brandt J., Bråmå T. Private Communications.

(27) Persson B. Users Manual UNICOM/SPEKS (In Swedish) Internal Report Saab-Scania Aircraft Division Linköping Sweden, 1984.

(28) Bartelds G., de Konig A.U. Application of Finite Element Methods to the Analysis of Cracks. Reports NLR TR 78138U, 1978 and NLR TR 79118, 1979. National Aerospace Laboratory, NLR. The Netherlands.

(29) Fredriksson B., Mackerle J. Stress Analysis Software for Fracture Mechanics. Department of Mechanical Engineering. University of Linköping Sweden, 1980.

(30) Fredriksson B. Elastic Contact Problems in Fracture Mechanics. Fracture 1977, vol 3, ICF4, Waterloo, Canada, June 19-24.