PAPER Nr. : 23

The Development and Application of Finite Element Stress Analysis Techniques at Westland Helicopters Ltd.

by

M.L.W. Salzer and P.C. Wood

Westland Helicopters and Hovercraft Ltd

TENTH EUROPEAN ROTORCRAFT FORUM
AUGUST 28 — 31, 1984 — THE HAGUE, THE NETHERLANDS

THE DEVELOPMENT AND APPLICATION OF FINITE ELEMENT STRESS ANALYSIS TECHNIQUES AT WESTLAND HELICOPTERS LTD.

M.L.W. Salzer

and

P.C. Wood

Westland Helicopters and Hovercraft Limited.

Abstract

This paper discusses the role of the finite element method in the design - development cycle. No attempt is made to give a theoretical background but rather to outline the technique, method of implementation and areas of application.

In particular the pre- and post-processing software package WISDOM is examined and the steps involved in performing a finite element statics analysis.

It is envisaged that with more powerful computers and the acceptance of CAD solid modelling in design, that the time required to perform a finite element analysis will be reduced even further. Some of the improvements that are required to achieve this time reduction are also discussed.

1.0 THE FINITE ELEMENT METHOD - AN OVERVIEW

In the design of a given product, one may need to refer to several branches of engineering science. Yet, whatever function the device must perform, it will need to be structurally sound. Now it is obvious that if a product actually breaks it certainly will not have been adequate, but malfunction may also occur due to excessive distortion under load or to spurious response to some environmental disturbance such as temperature or vibration.

Evidently stress and deformation analysis is central to design and in this paper we are concerned with showing what a powerful aid to this work is offered by the finite element technique.

Traditionally the stress analysis of complex structures, such as helicopter gearbox casings, relies on the experience and skill of the engineer to deduce the critical load paths and reduce the structure to one which can be analysed. The approximations inherent in this approach may result in,

a) potential under-design requiring extensive time consuming and costly rectification programmes, and

b) over-design in certain areas with potential weight and cost penalties which may never be detected.

The finite element method provides a solution to these difficulties, in that the component can be represented by a mathematical model which yields stress and displacement data for given boundary conditions. The engineer therefore, has insight into the behaviour of the component or structure before manufacture and test, and can take corrective action if necessary.

In the finite element method the component is divided into a number of finite sized and conveniently shaped sub-regions or elements. (See figure 1). The behaviour of an individual element is governed by a simple function. However, by using a sufficient number of elements, an acceptable representation of the overall real situation is obtained.

Each element is defined in terms of a sequence of grid points, where a grid point is a coordinate defined in 3D space. Various element types have been formulated and are applicable to specific structures. Figure (1) shows a finite element model of part of a rotor hub assembly and the types of elements used.

Each element is assigned a material property type and therefore parametric studies may readily be carried out to establish the performance of various material combinations. Furthermore, components manufactured in composite materials can be analysed since material properties may be anisotropic. For example, the rotor arms shown in figure(1) have been modelled as composite material.

Once the model is completed, appropriate loads and constraints must be applied to represent the real environment. The finite element program reads the model data and associated boundary conditions, forming a set of simultaneous equations where the unknowns are the displacements at the grid points of the element mesh.

Having solved this set of equations for the displacements, the element stresses can be determined and also the reactions at the constrained grid points.

In addition the finite element technique is also employed in the areas of vibration, heat transfer, fluid flow and geometrically non-linear problems such as crash worthiness.

Although the finite element method has been known and used since the early 1960's, it is only within the past five years that computer techniques have evolved sufficiently to make routine analysis practical. The following sections describe how WHL has employed the finite element method and produced software to facilitate the analysis of a wide range of helicopter components.

2.0 AN INTRODUCTION TO WISDOM

In 1978, when commencing studies for a Sea King Replacement helicopter, Westland decided to apply Finite Element Analysis in earnest as part of the preliminary design activity associated with gearbox transmission. At this time no commercial finite element processor existed which met our requirements. It was necessary therefore, in parallel with the analysis, to develop a suite of pre- and post-processing programs for use on the mini-computers available at WHL with interactive graphics terminals,

a) to speed model preparation, input data preparation and checking, and

b) to simplify the scanning of stress and deflection data resulting from the analysis.

Development of these programs has resulted in a suite of programs called WISDOM (Westland Interactive Structural Dynamic On-line Modelling) which is now being used at Westlands, on helicopter structures, rotor hubs and blades, gearbox casings and internals.

In designing WISDOM, it was decided to adopt a condensed, unformatted data base, independent of any specific finite element program. The advantage of this approach is that only input/output routines need be written in order to interface WISDOM with any finite element program. (eg. NASTRAN, ASAS, etc.)

The data base is controlled by a master summary file. Programs in the WISDOM suite interface with this summary file and automatically obtain information on the model status. If data is generated by a program, then this fact is recorded on the summary file.

It is possible to have several model variations and the status of each model can be displayed at any time during the analysis.

User errors can be trapped at program level. For example, modifying a finite element model which has results files present is illogical, as any changes to the model topology invalidates the results.

WISDOM uses graphics extensively in model preparation and in post-processing of results and the steps involved in performing a finite element statics analysis are summarised below.

* Collect data, drawings, loads, etc.

* Generate finite element model

* Assign properties and materials

* Apply loads and constraints

* Solve for displacements and stresses

* Interpret results

The following sections elaborate on the above topics and the analysis of a typical Tail Rotor Gearbox will be used as an example to indicate the facilities in WISDOM.

2.1 WISDOM - SOLIDS MODELLER

In the majority of cases the finite element model represents the actual component or structure under analysis; it is a true three dimensional body. The difficulty is that the data which describes a components shape is usually two dimensional. The WISDOM solids modeller has been designed for flexibility, and shape information can be acquired from several sources.

* 2D information - Computer aided drafting tools (CADAM)
  Digitised (preliminary drawings)
  Direct from the keyboard (coordinates)

* 3D information - Computer aided surface modeller (SYSTRID)
  FE Models (other sources)
  Digitised (in the form of 2D planes)

Usually information about a component is two dimensional in the form of preliminary or detailed drawings. This data can be used in two ways.

Firstly, a cross-section or plane can be generated into the third dimension, as illustrated in figure (2). A large class of components have a basic shape which is definable in this manner.

Secondly, a wire frame can be constructed and used in the three dimensional working area to flesh out a complex shape.

In the example of a Tail Rotor Gearbox, detailed drawings were available from the CADAM data base, and figure (3) shows external views of the gearbox and a section along the shaft centre-line. The gearbox is a single stage, spiral bevel, splash lubricated, reduction gearbox. Power is transmitted to the tail rotor shaft via a splined coupling. The gearbox casing is an assembly of four magnesium alloy cast components. The input and output housings, and end plate all locate on the main wheel housing. The details shown in figure (3) were transferred to the WISDOM data base in a much reduced form.

The gearbox is comprised mostly of cylindrical components whereby a cross-section can be used to generate a complete housing. Merging the input and main wheel housings was carried out using the intersection option. The internal support bearings where formed using a combination of the rotate and extrude options. See figure (2).

Figure (4) shows the segments of the gearbox and the completed model. WISDOM is configured so that the whole gearbox may be processed or a segment. Generally the engineer is only interested in local regions of the model, and it has been found that processing a single segment provides better computer response.

In both the two- and three- dimensional working areas the operator is given the option of generating elements singularly or as sub-divided patches. If an element is badly distorted the user is warned, and the element will not be generated.

In addition to the external drawing/surface information a comprehensive set of geometric aids are provided which include:

* Radii
* Intersections - Line, Plane and Curved surfaces
* Ellipses
* Normals
* Plane cut / Radial cut
* Verify

The file handling routines allow model segments to be copied, deleted or combined to form the complete structure, and the following global translations are available.

* Scaler

* Translate

* Rotate

* Mirror

* Orientation operator

Orientation operator - By defining two rectangular coordinate systems a model segment may be positioned in its desired location relative to some base segment.

2.2 WISDOM - PRE-PROCESSOR

Before the model can be passed to the FE program, further data is required which describes the element properties, material types, the way in which the model is loaded and how it is constrained.

From figure (3) it will be seen that the rotor shaft and output spiral bevel wheel are effectively disconnected in the axial direction and power is tranmitted via a spline coupling. Therefore the two input loading systems can be separated into the rotor loads and the gear loading. Three load cases were considered, given below.

a) Bearing loads for a nominal input power.

b) Vertical rotor load plus moment

c) Horizontal rotor load plus moment

From the three load cases it is possible to generate a number of in-flight conditions, by the use of super-position. Figure (5) shows the constraints used in each load case and a typical bearing load distribution.

Although the Tail Rotor Gearbox contains no composite materials, Westlands employs composite materals in a number of structural components and in the stress analysis of such components it is important to align the element material direction with the fibre layup. Figure (6) shows how this problem is overcome in the WISDOM pre-processor.

Apart from the visual checks, the user can obtain a force/moment balance about any position, check the correctness of the element property/material information and also calculate the weight and inertia values.

The WISDOM data base is independent of any specific finite element program format, consequently the aforementioned facilities are appropriate to NASTRAN, ASAS, or for any finite element program.

It is worth noting that existing ASAS models can be read into WISDOM together with material and property information and output as a complete NASTRAN deck.

2.3 COMPUTE DISPLACEMENTS AND STRESSES

Having fully defined the model, the next step is to solve for the displacements and stresses. This involves generating the stiffness matrices for each element, assembling them to form an overall stiffness matrix for the structure, and inverting that matrix.

A number of finite element packages are used at Westland Helicopters for this purpose, and each has its particular merits.

The program NASTRAN is used in the analysis of a range of problems including structural vibration, heat transfer, stress analysis of dynamic components, buckling instability and non-linear problems such as the effects of bird strike.

In the analysis of airframe structures the program ASAS is used to obtain displacements, stresses and element loads. Other commercial packages in use are MARC, used for its comprehensive non-linear capabilities and ARGUS used in the analysis of composite components.

A few in-house programs have been generated for specific tasks, such as torsion and flexure of prismatic bars, plane stress/strain, etc. These programs are simple to use and are also employed as training aids.

2.4 WISDOM - POST-PROCESSING

Having run the finite element program, the results are translated into the WISDOM data base. In the case of the element stress data, this is averaged at the grid points. It is worth noting that the predicted stresses are discontinuous, that is, from one element to the next the stress values may be different. The magnitude of the stress difference can be used to derive an error term which indicates the validity of the stresses and therefore the adequacy of the idealisation.

The results can be processed in various ways. In the analysis of static problems where displacement is the governing criterion, a view of the deformed shape of the structure can give vital insight into its behaviour.

In the example of the Tail Rotor Gearbox, displaying the distorted shape as a series of animated plots indicated that the gear casing was undergoing a large rigid body movement due to the weakness of the supporting legs. Further, it also indicated problems with the internal bearing arrangment and showed large local bending of the casing wall at the junction of the bearing support. See figure (7)

Subsequent analysis of the stress results showed high stress levels in both these areas of large movement. Figure (8) shows a surface stress contour plot of the inboard bearings.

The model can be sliced on any plane or radial arc, and results data displayed as contour plots on the exposed internal faces. Figure (9) shows a section of the Tail Rotor Gearbox through the gear shafts and the local bending of the inboard bearings is clearly evident. It is also possible to obtain an animated plot of these sections if required.

Information can be produced in the form of graphs, where results data is displayed using a spline fitting routine. This is useful for reports and quantifying data. See figure (10).

In addition to the graphical options, any data held on the WISDOM data base can be displayed on the terminal screen and in the case of results information, this may be sorted, enabling the maximum and minimum values to be found directly. The user may print the page of information displayed on the terminal screen only, obviating the need for mounds of paper output.

Once the finite element analysis has been completed satisfactorily, the associated files are archived to tape.

2.5 EXAMPLES

A number of examples are shown in figure (11) and range from complete airframes and gearbox assemblies, to, two dimensional problems as in the case of a rotor blade.

3.0 FUTURE DEVELOPMENTS

The finite element method offers significant benefits to the complete design - development cycle, only as long as the results may be obtained sufficiently quickly. Although an enormous reduction in the period required for analysis has been made, further enhancements are required to enable the full benefits to be realised.

So that the reader can appreciate the present trends in the finite element method, the table below provides estimated time scales involved in the analysis of a 'large' structure for past, present and future systems. Dividing the analysis into individual tasks, we have,

Time period	Past	WISDOM	Future enhancements
Consideration of idealisation	1 wk	1 wk	-
Model generation	20 wks	1 wk	2 days
Model checking and pre-processing	4 wks	2 days	2 days
Solution	1 wk	1 day	on-line
Results interpretation	4 wks	1 wk	2 days
TOTALS	30 wks	3.5 wks	1 wk

The above figures are only approximate, however, it is clear that fast mesh generation and processing facilities are essential if finite element analysis is to be used as a design tool. Presently WISDOM provides this capability, but it is envisaged that with more powerful computers and the acceptance of CAD solid modelling in design, the finite element analysis time scale will be reduced even further. Some of the improvements that are required to achieve this time reduction are given below.

* Fast interfaces to CAD data-bases

* Improved auto-meshing techniques

* Improvements to terminal/computer access

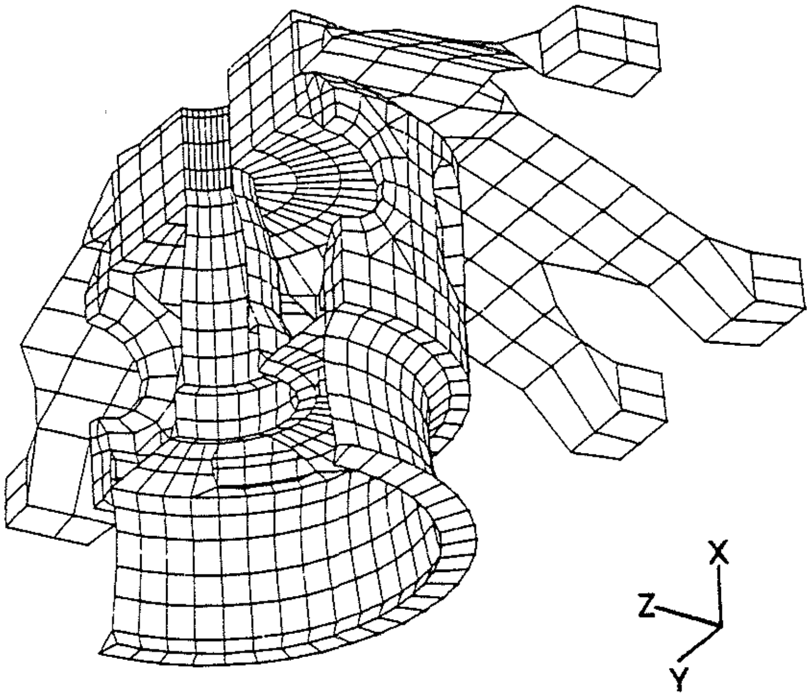
* Improved watchdog facilities both in generation and results interpretation.
  (Permits less skilled users)

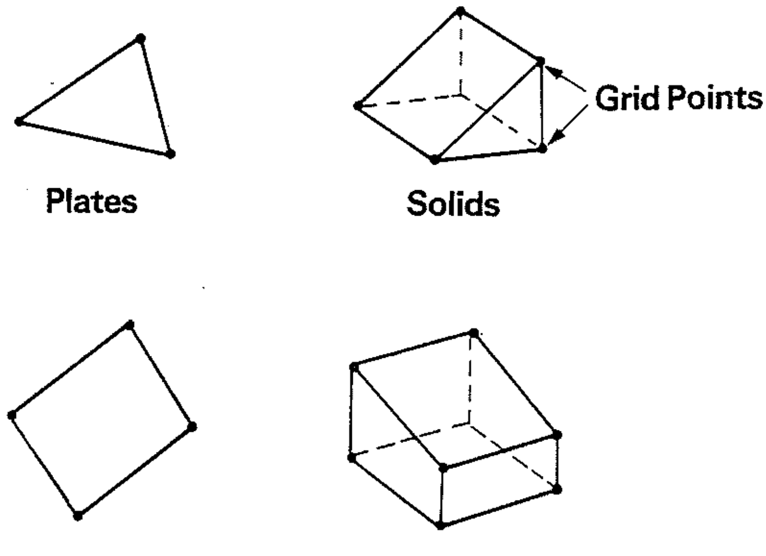
* Improvements in the interpretation and assessment of results.

4.0 Conclusions

Finite element analysis techniques can be successfully applied during the short timescales of project definition and preliminary design. This permits the analysis of the most complex of components, modelling static or dynamic behaviour, deflections, stresses, temperature distributions, model response, crack propagation, etc.

The finite element pre- and post-processor WISDOM, developed by WHL in parallel with real applications has proved of great benefit in the design of a wide range of helicopter components. These benefits will become more significant as CAD techniques generally are adopted and data bases generated in design are increasingly used in manufacture and development.





Element Types Used in the Above Model

Figure 1 Segment of Helicopter Rotor Hub

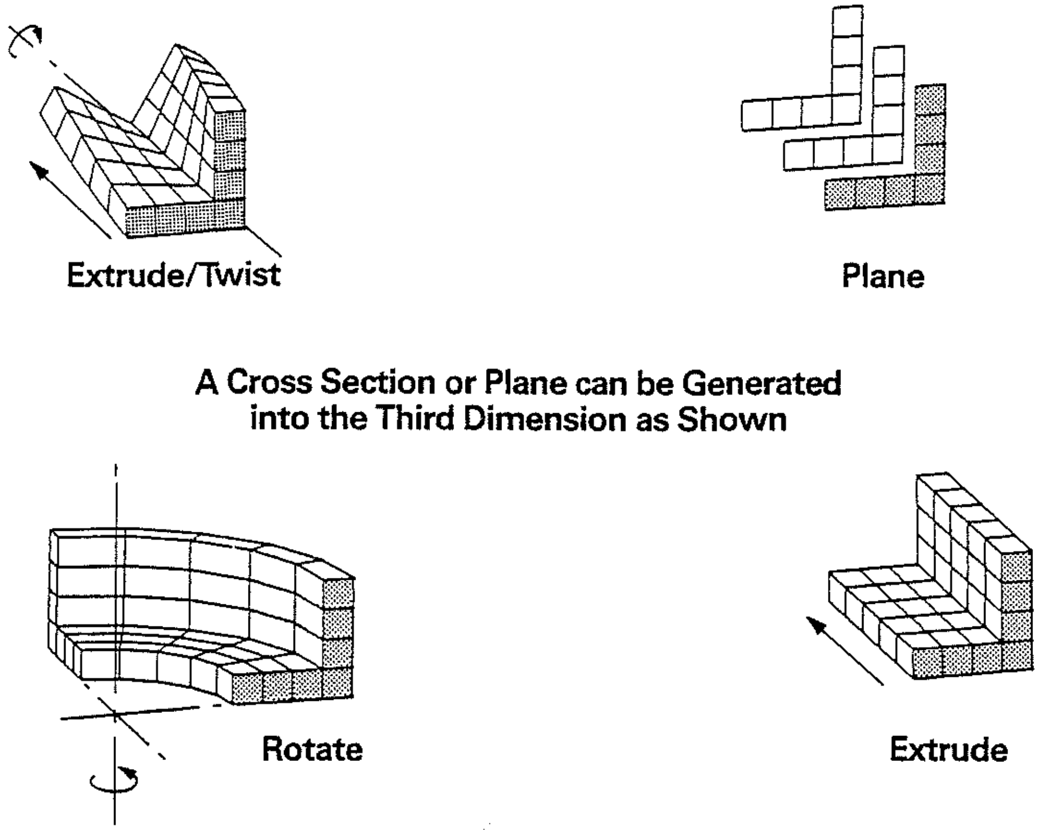


Figure 2 Data Generation from 2D Confirmation

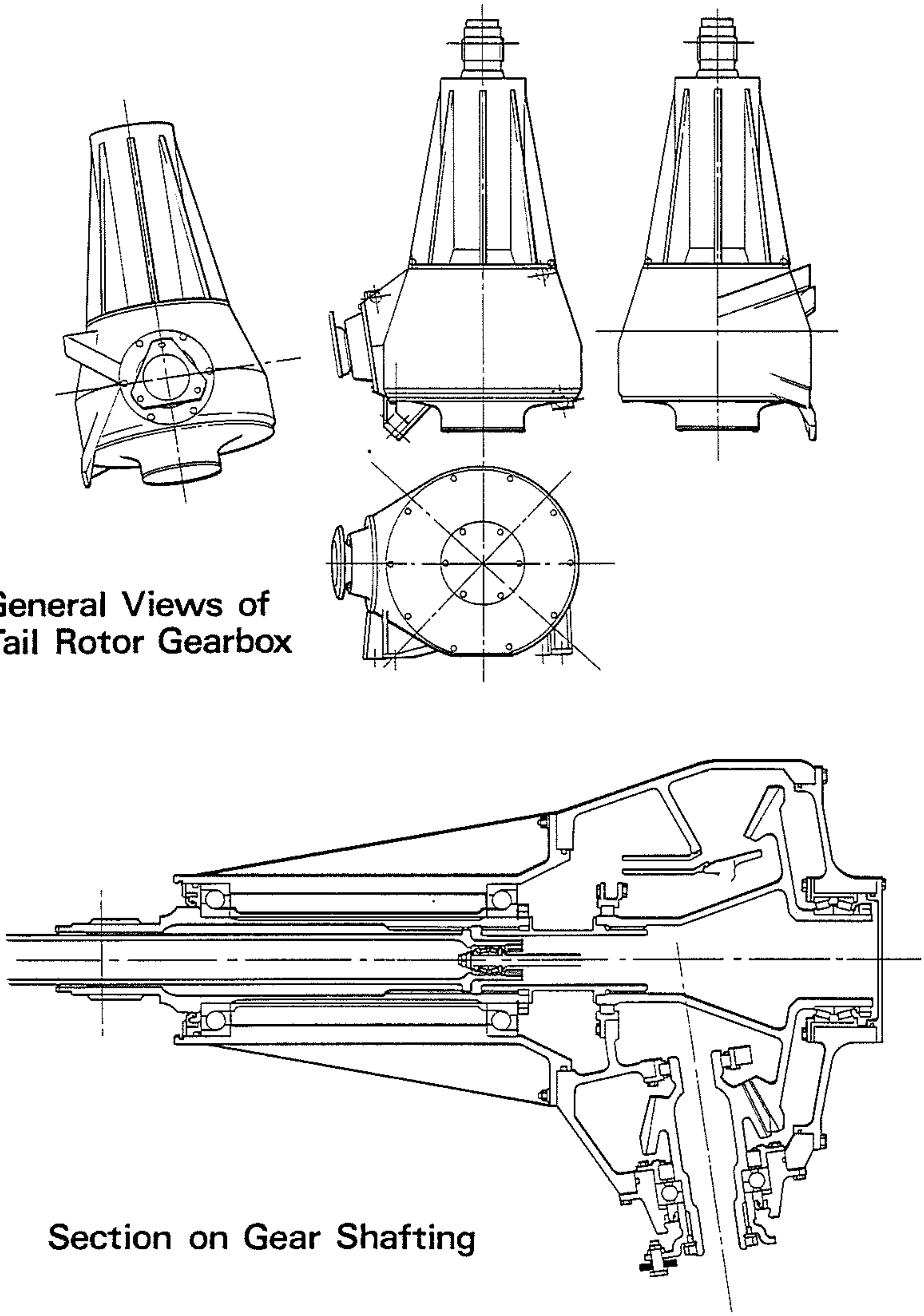
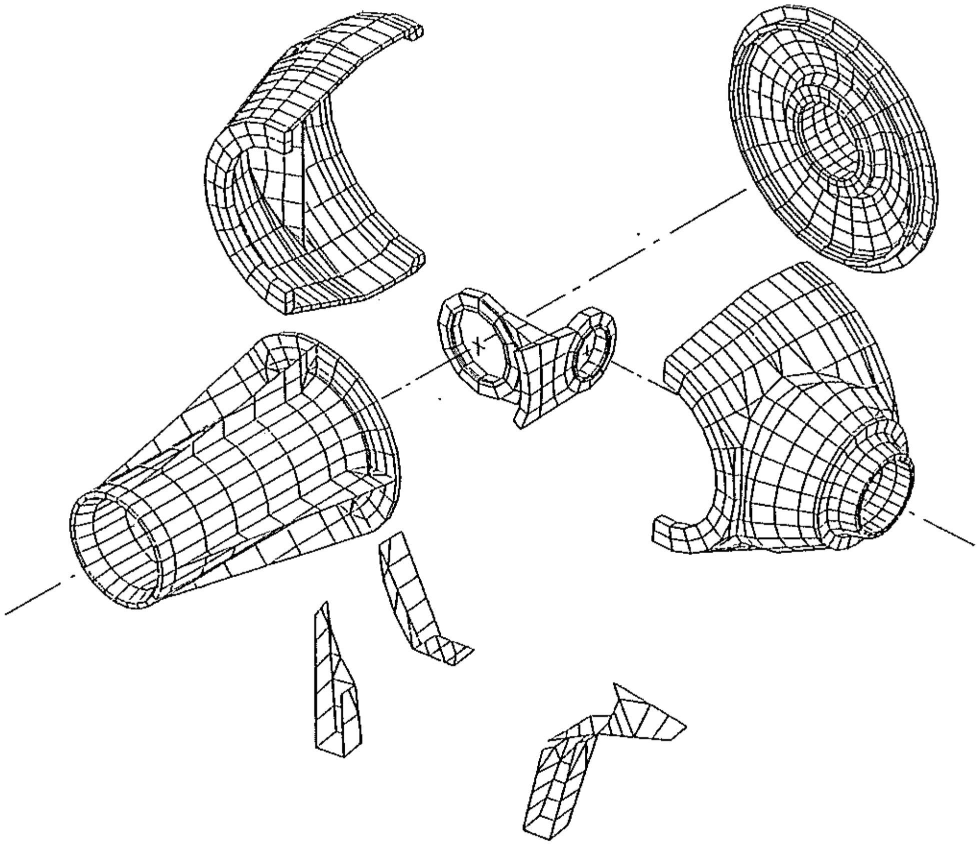
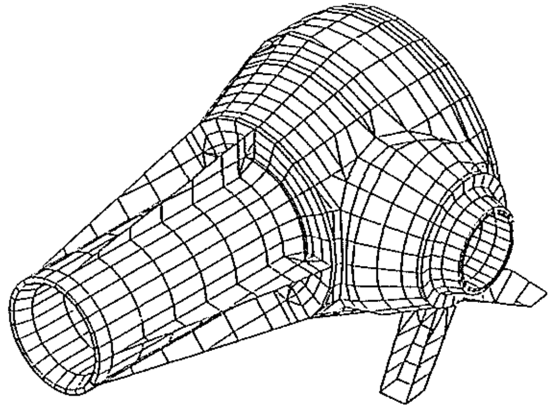


Figure 3 Detailed Drawing of Tail Rotor Gearbox

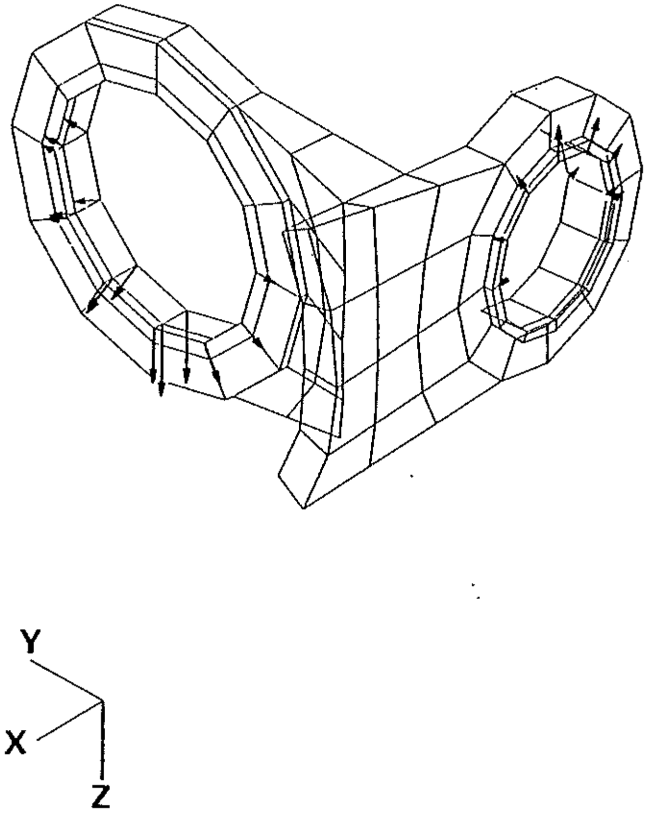


Exploded View of Gearbox Casing

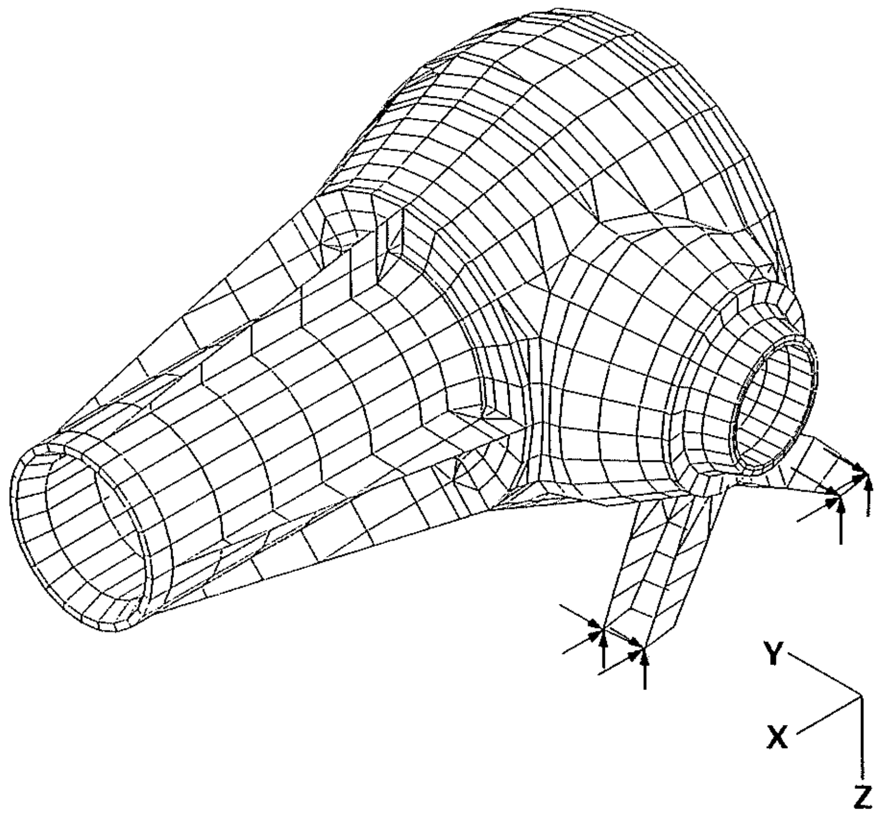


FE Model of Complete
Tail Rotor Gearbox

Figure 4 Segments of Gearbox and Complete Box



Typical Bearing Load Distribution



Gearbox Showing Constraints

Figure 5 Constraints and Loading

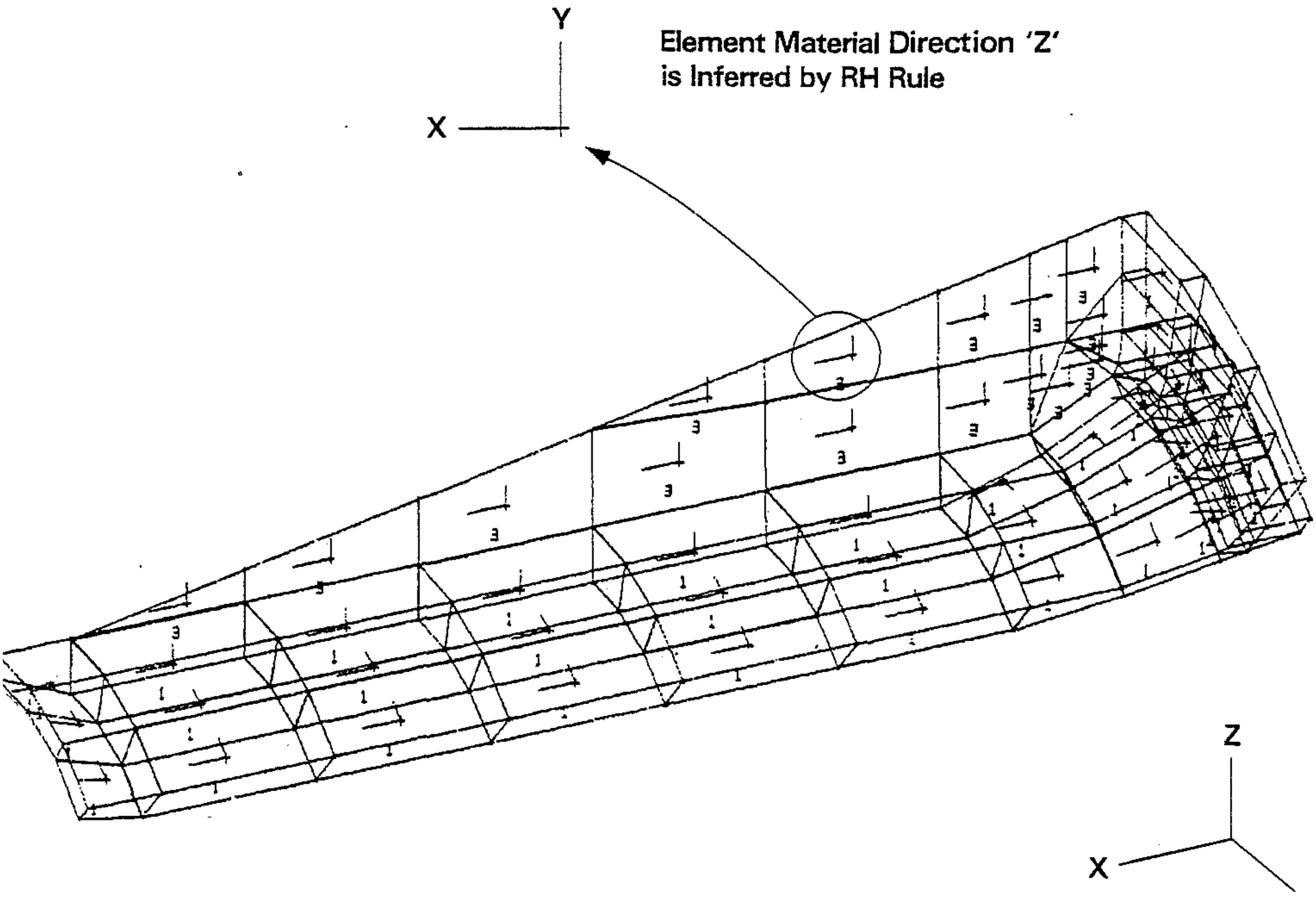


Figure 6 Local Element Directions

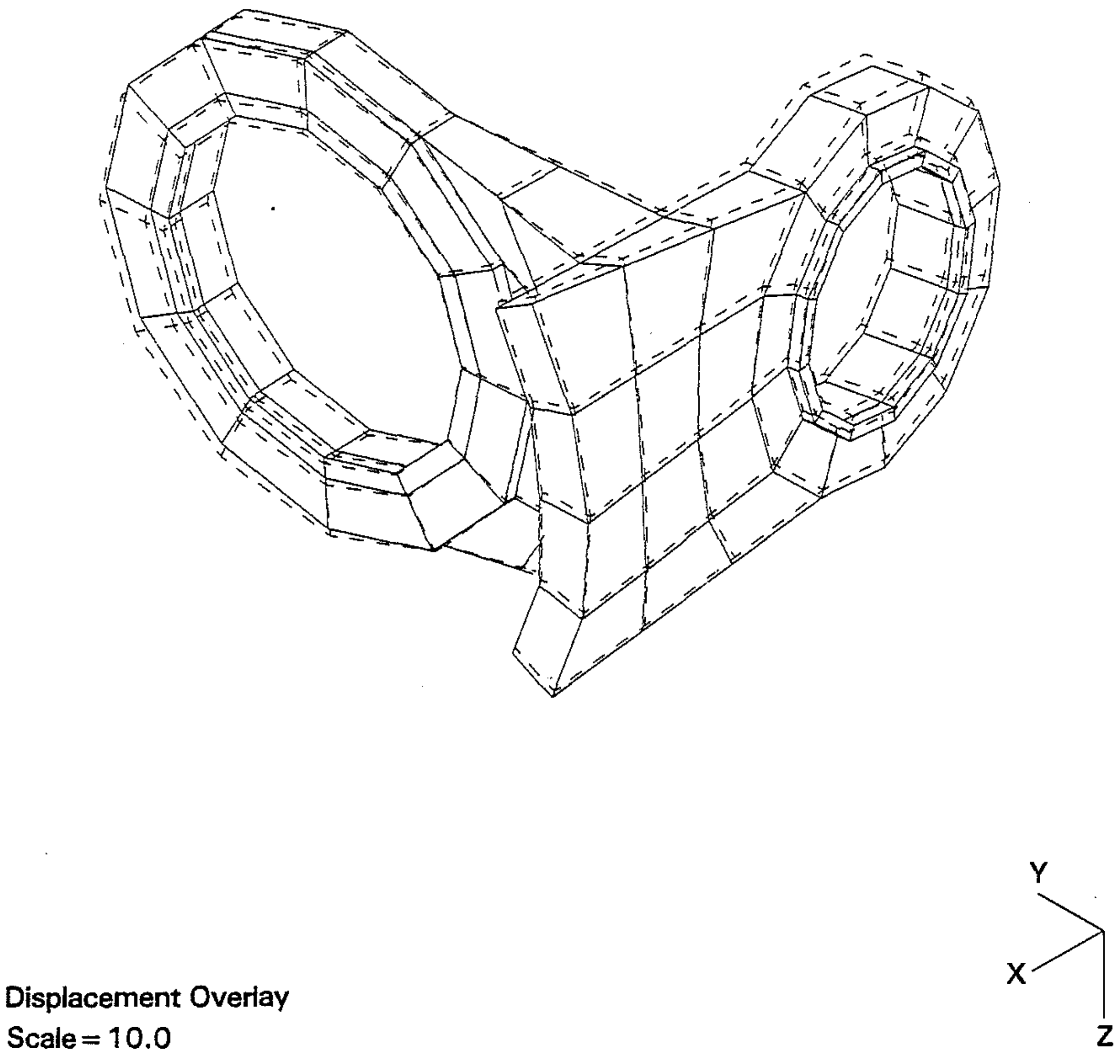


Figure 7 Internal Bearings Local Bending at Wall Junction

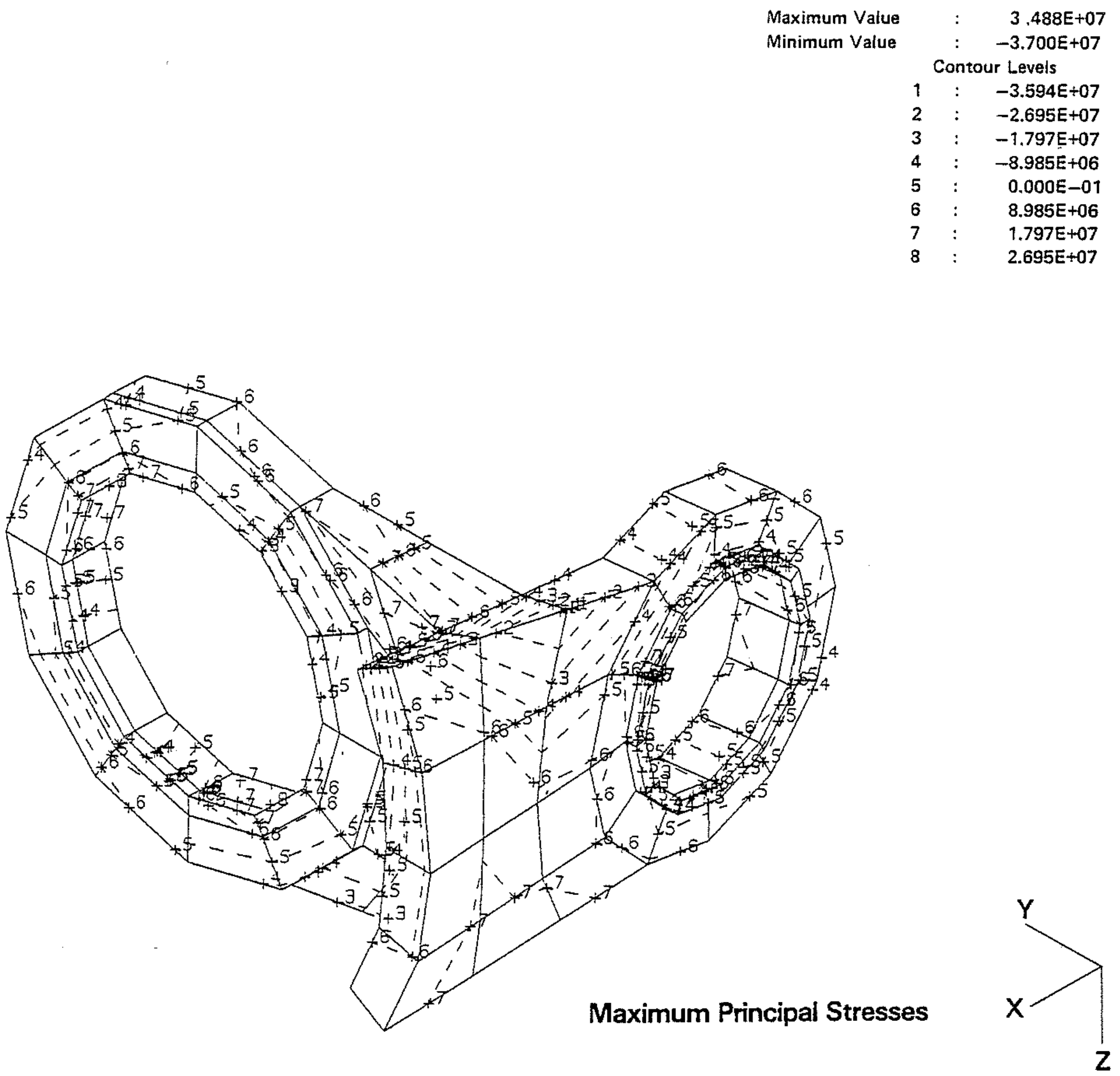


Figure 8 Internal Bearings Stress Contours

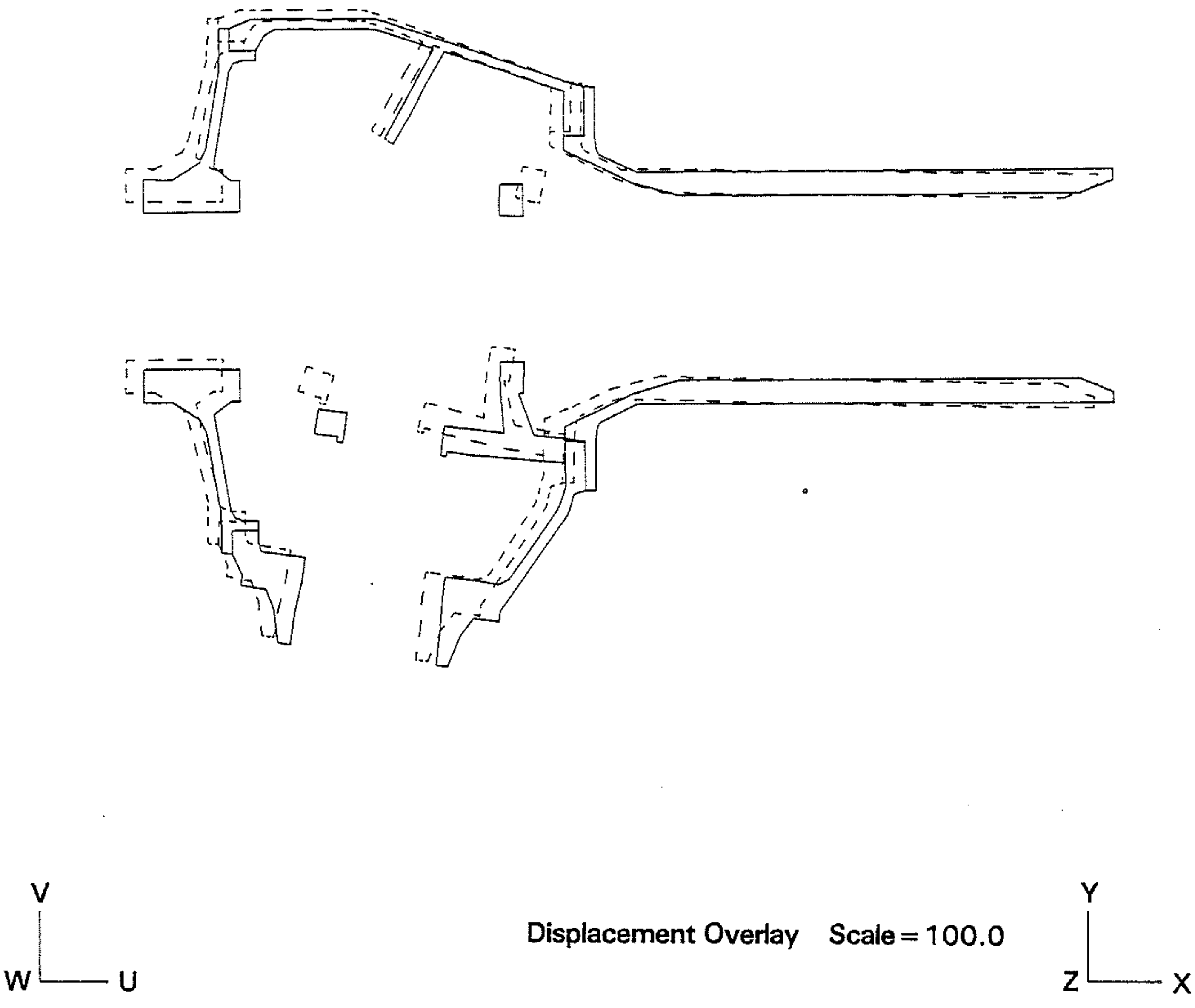


Figure 9 Section through Shafts Showing Displacement Overlay

Tail Rotor Gearbox
Inboard Bearing Output Shaft
Gear Loads And Axial Tail Loads
1. Normal X Stress
2. Normal Y Stress
3. Normal Z Stress
4. Max. Principal

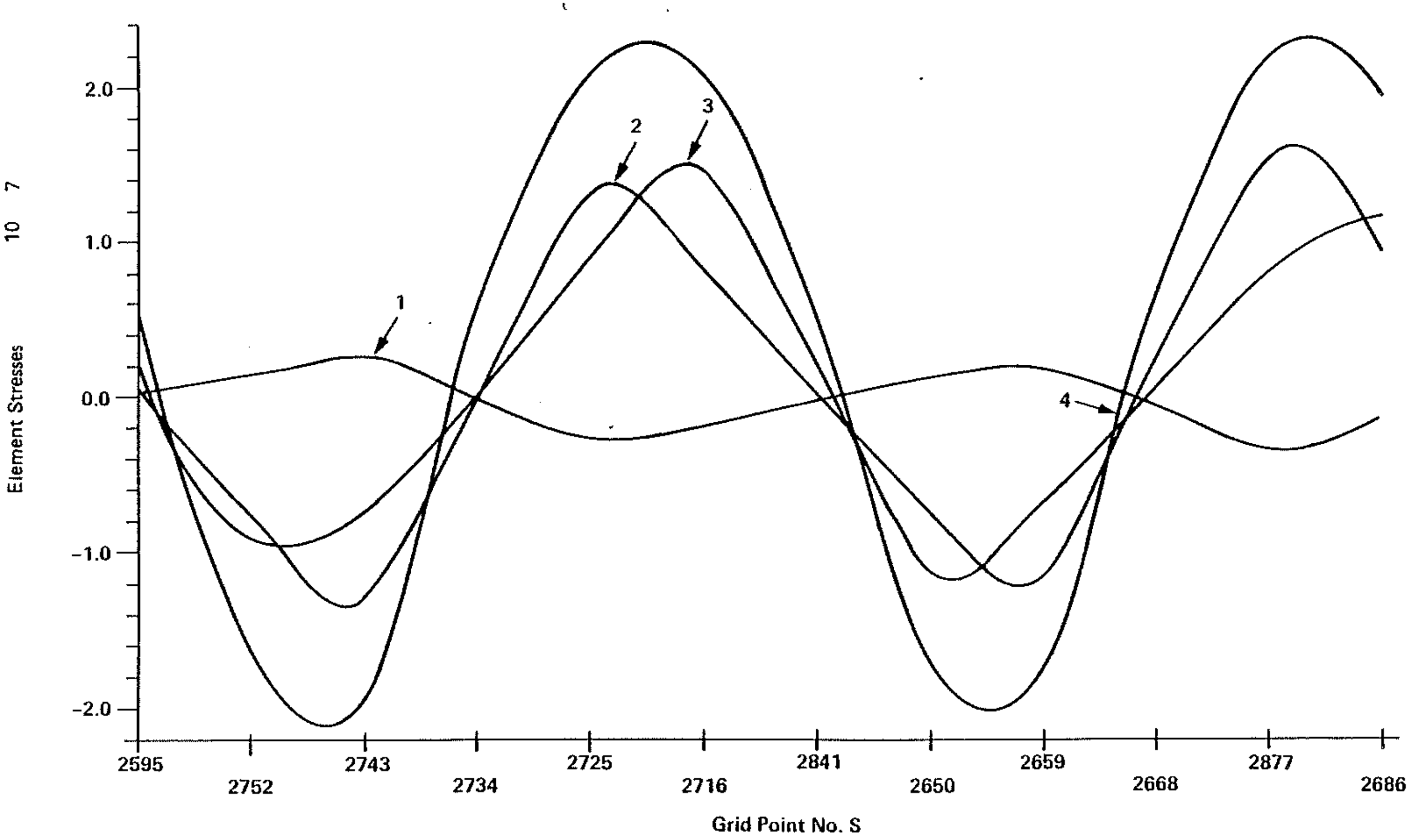


Figure 10 Stress Distribution Graph Presentation

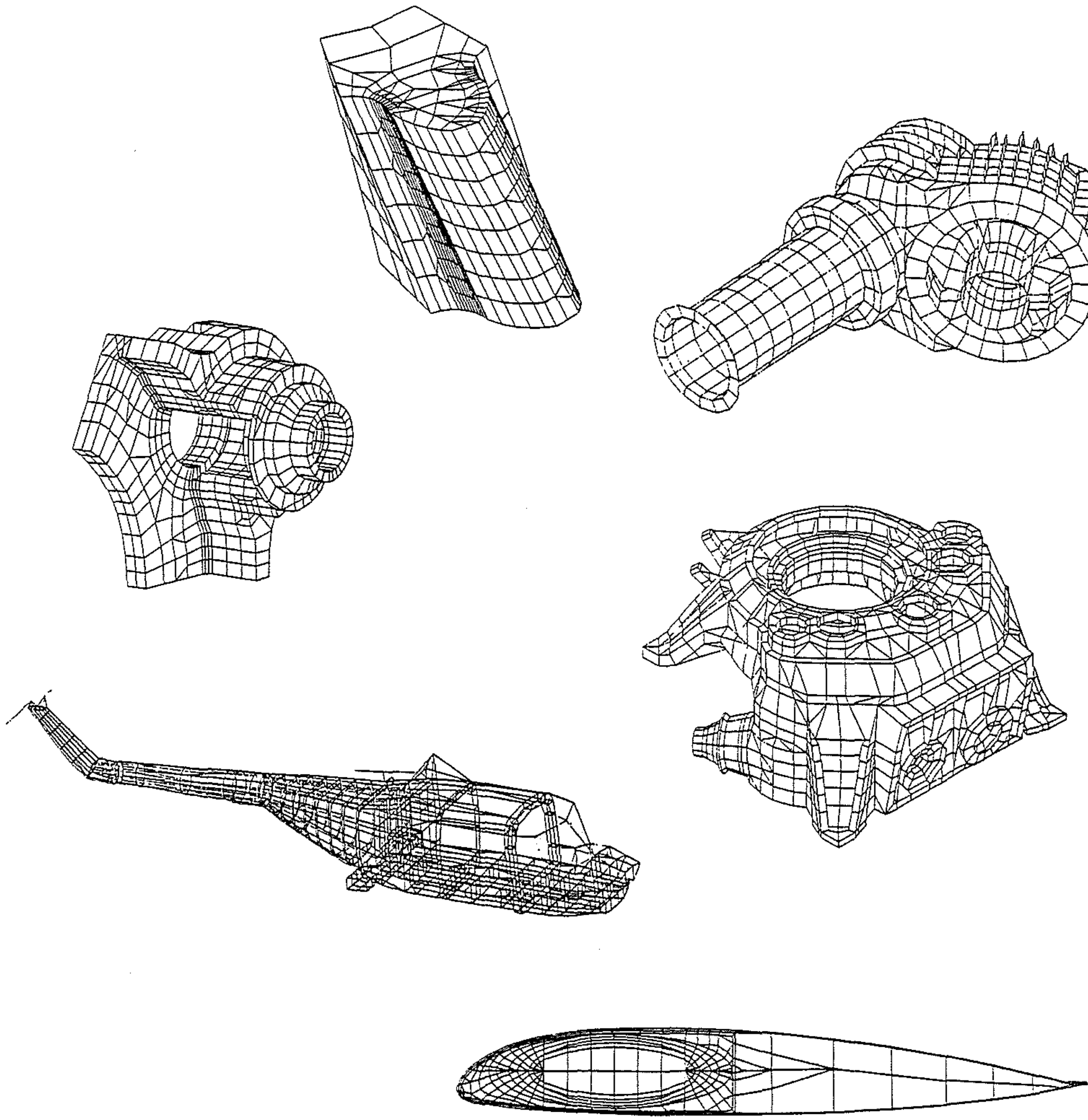


Figure 11 Examples