FOURTH EUROPEAN ROTORCRAFT AND POWERED LIFT FORUM

Paper No. 20

FINITE ELEMENT DYNAMIC ANALYSIS OF PRODUCTION AIRCRAFT

A. McLAUGHLIN

WESTLAND HELICOPTERS LIMITED, U.K.

September 13 - 15, 1978

STRESA, ITALY

Associazione Italiana di Aeronautica ed Astronautica
Associazione Industrie Aerospaziale

FINITE ELEMENT ANALYSIS OF PRODUCTION AIRCRAFT

A. McLaughlin
Westland Helicopters Limited, U.K.

ABSTRACT

This paper describes studies to compute the normal modes of vibration and forced response characteristics of production designed helicopters. The MSC/NASTRAN finite element system with the superelement technique - a marked improvement in terms of cost effectiveness and user convenience over previously employed substructure techniques - was used. The overall models contained between 5000 and 6000 degrees of freedom. A split data base was used, one for the internal and one for the external grid points of the superelements which constituted the primary reduction or condensation to the residual structure. A further condensation (Guyan reduction) was performed on the residual structures to leave the models with analysis sets of between 270 and 360 degrees of freedom. 'Free-free' analyses were then performed on these reduced models to extract the normal modes of vibration in the frequency range of 0 to 30 Hz. Using these modes, the responses at several points on the airframe to sinusoidal forcing input at the main rotor head were calculated. Correlation studies were then undertaken to compare the theoretically predicted results with those derived from shake tests performed on the aircraft.

1. INTRODUCTION

Westland Helicopters personnel have amassed some six years experience in the application of the finite element method to the dynamic analysis of structures. This experience has been gained almost exclusively in the application of the MacNeal-Schwendler Corporation version of NASTRAN-MSC/NASTRAN (refs. 1, 2 and 3). In October 1977, the finite element technique was adopted in the Engineering Department of Westland Helicopters as an engineering design tool. To date MSC/NASTRAN has been applied successfully to the dynamic analysis of a company-funded production helicopter and to the dynamic analysis of the production Royal Navy Lynx. In the latter case, MSC/NASTRAN formed a major part of the Lynx long-term vibration improvement programme. The NASTRAN/Multi-point shake tests correlation exercise of the above programme forms the major part of this paper.

Prior to October 1977, the application of MSC/NASTRAN to the dynamic analysis of structures was confined to the Research Department, in particular Structures Research, where the basic Lynx - WX 835 was analysed under an M.O.D. Research Contract (Refs. 4 and 5). This model was compiled from drawings in conjunction with the digitizing of an actual main rotor gearbox casing as a dynamics model. A suite of computer programs was also developed to assist in the data preparation and data checking stages of NASTRAN modelling. Having established a promising correlation between the results of the NASTRAN analysis and shake tests. performed on the aircraft, the technique was adopted in the Engineering Department.

Due to the very heavy workload and the chronic shortage of time available to perform the analyses, and the shortage of personnel trained in the use of NASTRAN for dynamics, an alternative to the production of models from drawings was required. Existing statics models for these aircraft, suitably modified for dynamics purposes, were utilised

The utilisation of existing Stress Office finite element models provided a valuable opportunity to test the feasibility of "common modelling".

2. FINITE ELEMENT IDEALISATION

This section describes the idealisation of the production Navy Lynx using the MSC/NASTRAN finite element system (ref. 1, 2 and 3). The MSC/NASTRAN superelement approach was adopted as it represented a considerable improvement in terms of cost-effectiveness and user convenience over previously employed substructure techniques (refs. 4 and 5).

Substructuring Concepts

The general analysis procedures used in all forms of substructure analysis may, for discussion purposes, be logically divided into three effectively distinct phases:

Phase I Process Individual Superelements

An analysis of each unique substructure or superelement to produce, in matrix form. a description of each superelement in terms of a reduced set of degrees of freedom, that, at a minimum, include the boundary degrees of freedom that connect to adjacent superelements.

Phase II Process Superelements Collectively - Residual Structure.

The combination of the matrices from Phase I with any additional structure the analyst may define and the subsequent analysis of the residual structure.

Phase III Data Recovery

The results from Phase I and Phase II are utilised to obtain response data for the individual superelements.

It must be emphasised that the definition of three distinct phases in a substructure analysis is merely an artifice to aid in the subsequent comparison between the traditional substructure approach and that utilised in the superelement technique. It should not be implied that the concept of phase infers that separate computer runs are required for each phase to perform a substructure analysis with the superelement concept. In fact, no such phase III was employed in this analysis, and Phases I and II were performed as one computer run. (see figure 3).

Structural State of the Helicopter

The aircraft modelled by NASTRAN and used in the experimental shake tests was a production Royal Navy Lynx. An aircraft was taken from the production line and was suitably modified for the experiments. The engines were replaced by time-expired test engines. The main rotor head and servo jacks were represented by an equivalent mass which had a 2.75 metre long aluminium beam attached to its top. This allowed either roll or pitch moments to be input to the head by means of antiphase forces at the ends of the beam. The pilot's, tail rotor, nose radar and all other sensitive instruments were represented by ballast weights. The aircraft was in a fully-fuelled condition and it was further ballasted to represent a mission all-up-weight and centre of gravity condition. The complete helicopter assembly had a mass of 4091 kg.

Derivation of the Model

Traditionally in W.H.L. the statics and dynamics analyses of large-scale finite element structural models have travelled along diverse paths. The personnel responsible for the statics analyses have a predilection for the ASAS system (ref. 6) (an alternative commercially-available structural analysis system) and the dynamics personnel have, on the other hand, a requirement to use NASTRAN - a requirement insofar as there is at present no logical alternative to the power and versatility of MSC/NASTRAN with the superelement capability.

In order to reduce costs and keep the elapsed time required to produce a dynamics model to a minimum, the W.H.L. Stress Office ASAS statics model (ref. 6) formed the basis of the dynamics model. This model though quite detailed, did not have gearboxes or engines, nor did it have a tailcone assembly. All skin elements in the Stress Office model were represented as shear panels. It was also assumed that all panel elements buckled under load unless the particular panels were of sandwich construction. The effective membrane action of the skin elements was then 'lumped' into the beam and stringer elements which bounded the panels.

Using 'in-house' developed ASAS/NASTRAN translator computer programs, which had never been tested on a production standard test model, a NASTRAN version of the Stress Office production Naval Lynx model was produced. The shear panel elements from the statics model were then transcribed into membrane elements. A manual exercise to 'de-lump' the then redundant end-load carrying capability of the shear panels was undertaken. Bending/transverse shear elements were then overlayed, where necessary, on the membrane elements to represent fully the stiffnesses of sandwich construction panels.

From the results of previous work done (refs. 4 and 5) the main rotor gearbox was represented as 'rigid' on four finite stiffness feet. A stick-beam model of the gearbox and engines was produced and overlayed on the fuselage model.

A model of the tailcone assembly was produced directly from engineering drawings and was added to the above.

The element types and complements used in the model is to be found in Figure 2.

In general, the NASTRAN structural mass matrix was generated by using the mass density parameter on the material property cards. The masses were allocated to the grid points by simple lumping. All non-structural mass items were allocated to the grid points manually and were represented by CONM2 elements.

The model was then divided implicitly into four superelements, the forward fuselage up to the forward main lifting frame, the rear fuselage up to the transport joint, the engines and gearbox assembly and the tailcone assembly. Full details on the superelement technique are to be found in the relevant NASTRAN manuals (refs. 1, 2 and 3).

Superelement Philosophy

As the NASTRAN analysis of the production naval Lynx was a part of the Lynx long-term vibration improvement programme, it was considered that some form of substructuring technique was required. In this way key areas of the Lynx aircraft could be set aside for detailed study without the financial or time consuming constraint of regeneration and re-assembly of the complete mass and stiffness matrices.

The superelement technique was adopted because of the powerful data storage and data control features of the data base system. The model was read from cards once and the bulk data was stored on a data tape. All stiffness and mass information was then stored on two data bases, one for all external grid point information and one for all internal grid point information. Each of these two data bases required two magnetic tapes. One for the data base from which information was retrieved and one for the newly created data bases. A total of five tapes was required. To perform the same analysis using the automatic substructuring technique (Refs. 1, 2, 3, 4 and 5) at least double this number of magnetic tapes would have been required.

3. THE ANALYSIS

Because of the insufficiency of in-house computer power for technical and scientific work, the commercially attractive computing "rates" and turnround available, the services of the London based computer bureau SIA (LONDON) LTD. are used. There a CDC 6000 series computer can be made available for large-scale finite element dynamics work.

Normal Mode Analysis

A schematic representation of analysis is to be found in Figure 3. Single-level superelements were used on MSC/NASTRAN version 40, DMAP 3 (ref.3).

The stiffness matrices for each individual element of each superelement was generated in turn. The stiffness matrices for each superelement was then assembled and reduced. The mass matrices were assembled and reduced using the same transformation matrices as were used for the stiffness matrices. The residual structure mass and stiffness matrices were then subjected to a further static condensation (Guyan Reduction) to the final analysis set of equations of order 335. The Givens Method was then applied to extract the first twenty-five normal modes. The elemental strain energy for the elements of the residual structure for all twenty-five modes was also computed.

The eigenvalues and eigenvectors, the mass and stiffness matrices for the residual structure was dumped onto magnetic tape. This was required in order to produce an accurate mass matrix for the shake test personnel so that the orthogonality of the experimentally derived modes might be checked. This also provided an opportunity to calculate the forced response 'in house' plus further post-processing of the modes.

A budget of £13,000 was allocated for this task and five iterations to a solution were required to produce Figure 4.

Forced Response Analysis

The acceleration responses to pitch and roll moments, and lateral and vertical shears input, in turn, at the main rotor head was calculated at various points on the airframe.

The modulus of the response was calculated for comparison with the experimental results.

These calculations were performed 'in-house' on the W.H.L. IBM 370-155 machine by utilising the output from the NASTRAN Normal Modes Analysis which was read onto an IBM compatable 9-track magnetic tape.

4. EXPERIMENTAL WORK

Shake tests were conducted using the MAMA system - Manual/ Automatic Multipoint Apparatus. The apparatus controls up to six electro-magnetic vibrators which are suspended from mobile supports and which can transmit independent forces of up to 62N at any point on the structure under test. The excitation frequency is automatically controlled to maintain the acceleration response in quadrature with the exciting free at one point on the structure, whilst the distribution of the forces is controlled manually.

The aircraft was suspended from an overhead gantry by a heavy-duty braided rubber cord attached to the rotor head. This gave an approximately free-free condition as the suspension modes were of low frequency when compared to the elastic modes of the fuselage.

Prior to setting the normal modes some overall "feel" for the dynamic characteristics of the helicopter was required. This was achieved by recording the complex frequency response (using single point excitation) of the structure at a number of points, for a number of excitor positions, to produce a "panorama" of sensitive frequencies and areas of maximum response.

Isolation of Normal Modes

Each mode shape was recorded at 122 points in the vertical, lateral and fore/aft directions giving a response vector of order 366. These monitoring points were the same as the grid point set chosen for the NASTRAN analysis. These points were uniformly distributed throughout the structure and all large items of mass were included to ensure an accurate representation of the kinetic and strain energies.

In general, a phase-error of less than $\pm 5^{\circ}$ was obtained in total quadrature acceleration response at all points.

Response to Sinusoidal Forcing

An 890N vibrator was attached to each end of the aliminium 'I' beam on top of the mock-up rotor head. Moments were input to the structure by driving these vibrators in antiphase. Shear inputs were obtained by driving the vibrators in phase.

The modulus of the acceleration response was obtained in the frequency range 0 to 30 Hz at various points in the structure for force inputs of 542.3 Nm pitch moment, 542.3 Nm roll moment, 355.9 N fore and aft shear, and 355.9 N lateral shear. All forces were input at the main rotor head.

5. RESULTS

The NASTRAN analysis extracted 335 eigenvalues which ranged from 7.365154E-05 to 5.537332+09. The lowest 26 of these lay in the frequency range of 0 to 50 Hz.

Figure 4 is a chart of the NASTRAN and experimental modal comparisons. The 'Mode Shape Description' refers to the NASTRAN mode shape and is for identification purposes only. The modal damping was assumed to be viscous in nature and it is presented in the form of 'Q' factors (the dynamic magnifier) for each mode. These values were obtained from measurements made in the shake tests.

Unexpectedly, excellent correlation has been obtained in mode shape comparisons, but less than perfect agreement has been obtained for the frequencies. An orthogonality test parameter of 1.0E-07 was specified, that is, provided that no off-diagonal term on the generalised mass matrix was greater than 1.0E-07 the modes would be considered as orthogonal. The largest off-diagonal term was found to be 8.84E-13 and this occurred between modes 4 and 7.

Eleven NASTRAN elastic modes were found in the range of interest compared with twelve experimental modes. The discrepancy arises from the response of the port engine moving rather unexpectedly in isolation. The six rigid body modes should have had zero frequency but, because of the nature of the algorithm used (the Givens Extraction), this is not possible.

Figure 5 shows the theoretical and experimental mode shapes for the first elastic mode. This is the fundamental vertical bending mode shape of the fuselage. Lack of space precludes the inclusion of more mode shape comparisons.

Initial forced response results derived from the above modes have proved disappointing. It is considered that this is in the main due to the discrepancies in the frequencies and the inability of NASTRAN to produce a mode equivalent to that from the shake tests whereby the port engine was vibrating virtually in isolation. The non-linearity of the engines assembly as displayed in the isolation of the lateral engine normal modes is also a major factor. Work is continuing, however, to improve the model and hence the response predictions (see Figure 6).

6. CONCLUSIONS

The ultimate aim of this work was to provide, in parallel with experimental work carried out in the aircraft, a mathematical model which was to be representative of the airframe in terms of both normal modes and forced response. Comparison between the shake tests and the MSC/NASTRAN finite element results have shown excellent correlation for the mode shapes, but less than perfect agreement between frequencies and the forced responses. It is considered that these results could have been bettered.

There are obvious discrepancies in the frequency comparison. The tailplane frequencies are obviously in error. These can and will be corrected. A possible error source is the fitting at the root of the tailplane which attaches the tailplane to the fin. This fitting was assumed to be 'rigid', but there could be flexibility there which had not been considered. It is not expected that the tailplane modes would have any influence on the forced response. Another major discrepancy is in the engine/gearbox region. The engine lateral modes in NASTRAN are two low in comparison with the shake test counterparts. The representation of the engine/gearbox torsion couplings is understiff. A high degree of non-linearity for the gimbal torsional stiffness was evident from the shake tests performed to isolate the engine lateral modes. It is considered that, if the normal modes for the engines were set at or around 25 Hz. then there would be sufficient modal interaction to increase the frequency of the fuselage torsion mode and amplify the torsional content of that mode (mode 15).

In general, the normal modes from NASTRAN are unexpectedly low. One would have expected the theoretical modes to converge from above to the experimental modes since we are using the displacement approach. The displacement method determines a minimum potential energy solution from a set of assumed displacement patterns and yields a lower band in displacements. Hence the frequencies should converge from above as a model is refined. This convergence from above would also be expected because of the extensive use of the Guyan Reduction facility.

In order to explain this discrepancy in frequency a series of exhaustive tests was undertaken. The mass representation for the aircraft was completely re-checked and declared to be correct to an acceptable tolerance. The stiffness additions to the Stress Office statics model, namely, the gearboxes, engines and tailcone assembly were also checked and cleared. At the present time, though our investigations are incomplete, it is considered that the stiffness deficiencies are to be found in the Stress Office stiffness representation of the main fuselage.

The Stress Office modelling philosophy is extremely convervative in nature and it is also the case that they do not employ a one-to-one correspondence between finite elements and the constituent parts of the airframe. It is considered that, if "common modelling" for statics and dynamics was to have any measure of success in W.H.L., the stress analysts would require a much more refined stiffness representation in their models.

Though the forced response results continue to be disappointing it is considered that with an improvement in the frequencies and mode shapes a corresponding improvement in the forced response will accrue.

The superelement technique as employed in NASTRAN is most efficient and cost-effective approach to large-scale modelling as used to date in W.H.L. Studies are now underway to improve the speed and accuracy of the generation of finite element models (Ref.7). It is considered that the use of such systems will improve dynamic analysis turn-round and reduce costs.

7. DISCUSSION

It is a common complaint among designers that the results from a dynamic analysis of large-scale structures are made available to them far too late to have a significant input at the design stage. This is a perfectly valid criticism. Because of the shear bulk of the data required and the degree of manual involvement, the production of large-scale finite element models is both slow and error-prone. It is also the case that dynamics analysis tends to lag the corresponding statics analysis.

To date, at W.H.L. the dynamics analysis of large-scale structures has commenced well after the commencement of the corresponding statics analysis, during the initial design stages.

In order to make best use of the finite element technique in its application to structural analysis for dynamics, the method should first be applied at the project definition stage. The dynamic analysis must also keep page with the statics analysis.

Figure 7 shows the block diagram representation for what is considered to be the optimum approach. The salient differences between the optimum approach and that which has been applied to date at W.H.L., is the involvement at the outset of the structural dynamicist allowing them to make recommendations, at the earliest possible time, on stiffness characteristics of the structure. This in turn leads to more positive recommendations concerning vibration attentuation methods.

The following is the recommended procedure:

1. A crude statics model should be produced and an analysis whereby the model is subjected to normal flight loads at the normal operation speed, undertaken. The results of this analysis should be made known to the dynamicist. The areas of high stress or strain energy should then be scrutinised and, if required, modelled in greater detail.

2. A crude dynamics analysis should then be performed to provide the normal modes for a datum case. This is possible at this stage because the non-structural mass distribution can readily be determined at this stage.

3. Model generation routines should then be employed to minimise the elapsed time required to produce models and reduce the possible number of errors in the model due to human factors.

4. An iterative approach to the analysis is then to be recommended such that, as the design becomes more advanced and detailed, then the model complexity is increased in unison.

Although it would be desirable and convenient at W.H.L. for a common idealisation to be representative for a dynamic and statics analysis, because the basic philosophies employed are different, this is not possible. It is agreed, that all aspects of commonality should be exploited to reduce costs and provide the best use of manpower resources. The stress and dynamics analysts should work together to produce the co-ordinate geometry, the connectivity and then they should go their separate ways at the choice of elements and element sizing stage.

It is of great benefit to the designers concerned, that the statics and dynamics models be at least common in co-ordinate geometry, as communication between the stress and dynamics analysts and the designers is simplified.

At W.H.L. serious consideration is being given to the use of the FEMALE (ref. 7) system which is commercially available at SIA (LONDON) LTD. In this system at the input stage the connectivity, grid points, element and material properties and constraints are specified and an MSC/NASTRAN data deck is produced. It is considered that the use of such a system would decrease the elapsed time required and minimise the possible errors in the production of large-scale finite element models.

8. ACKNOWLEDGEMENTS

The author would like to acknowledge the contributions made by Mr. D. Sapp and Dr. S. King of W.H.L. He would also like to acknowledge Mr. C. Barber and his co-workers who under took the MAMA shake tests.

Acknowledgement is also due to Mr. P. Jeanes and Mr. G. Shoppee of S.I.A. (Ltd.).

This work was performed with the support of the Ministry of Defence (P.E.).

9. REFERENCES

1. Caleb W. McCormick. MSC/NASTRAN Users' Manual. The MacNeal-Schwendler Corporation, 7442 North Figueroa St., Los Angeles.

2. Richard H. MacNeal. The NASTRAN Theoretical Manual. The MacNeal-Schwendler Corporation, 7442 North Figueroa St. Los Angeles.

3. Jerrand A. Joseph. MSC/NASTRAN Application Manual for CDC 6600 Series. The MacNeal-Schwendler Corporation, 7442 North Figueroa St., Los Angeles.

4. A. McLaughlin, G. M. Venn and A. J. Barnard. An Investigation of Helicopter Structural Dynamics using MSC/NASTRAN. W.H.L. Research Paper R.P.556, July 1977.

5. A. McLaughlin, G. M. Venn and A. J. Barnard. A Vibration Study of the Lynx Airframe Using the Finite Element Method. Proceedings of the Third Helicopter and Powered Lift Forum. Paper No. 21, Sept. 1977.

6. ASAS - Atkins Structural Analysis System Users Manual. Atkins Computing Services Ltd., Woodcote Grove, Ashley Road, Epsom, Surrey, England.

7 FEMALE - Finite Element Modelling and Language for Engineers S.I.A. (London) Ltd., 23 Lower Belgrave St., London SW1W ONW, England.

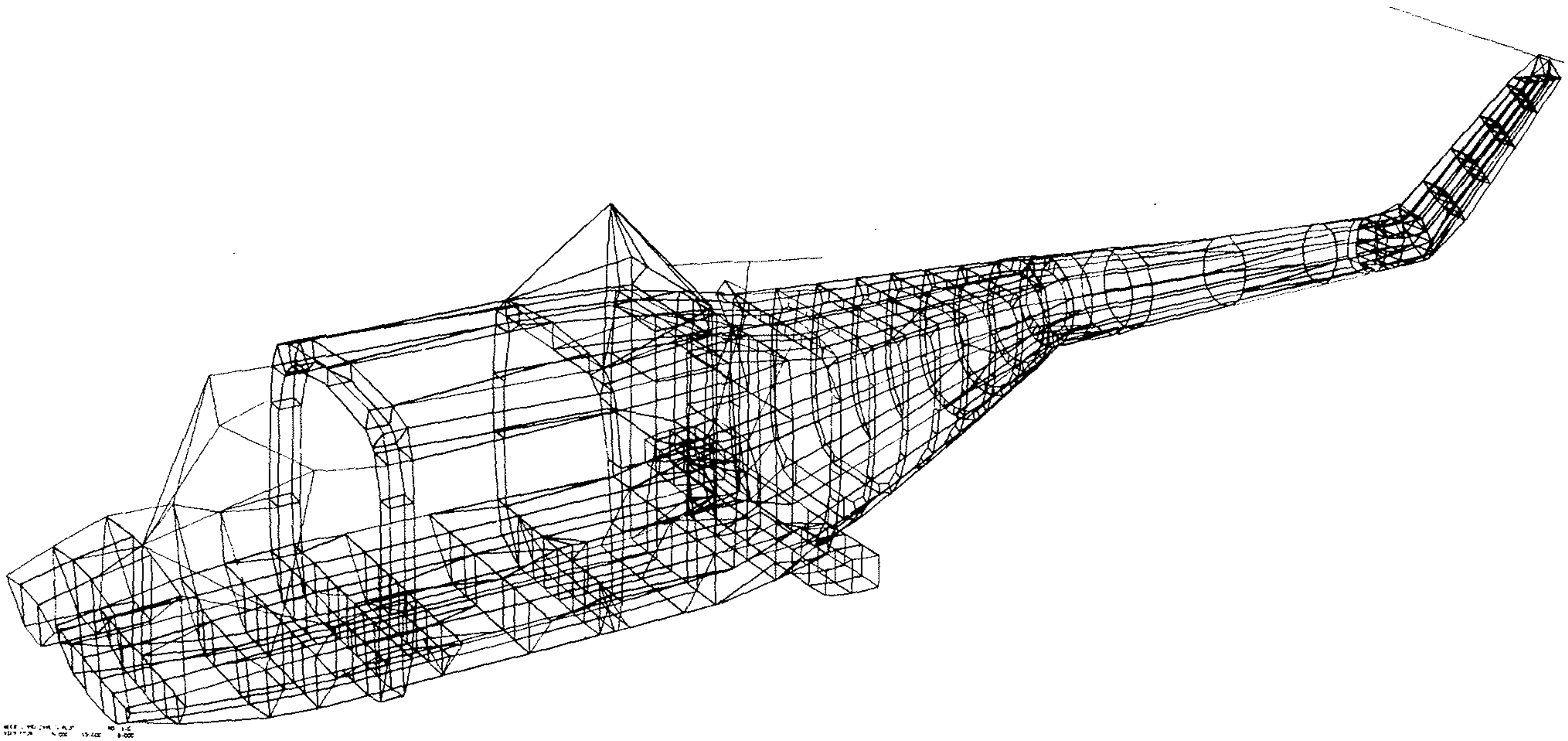


FIGURE 1
PRODUCTION NAVY LYNX
NASTRAN MODEL

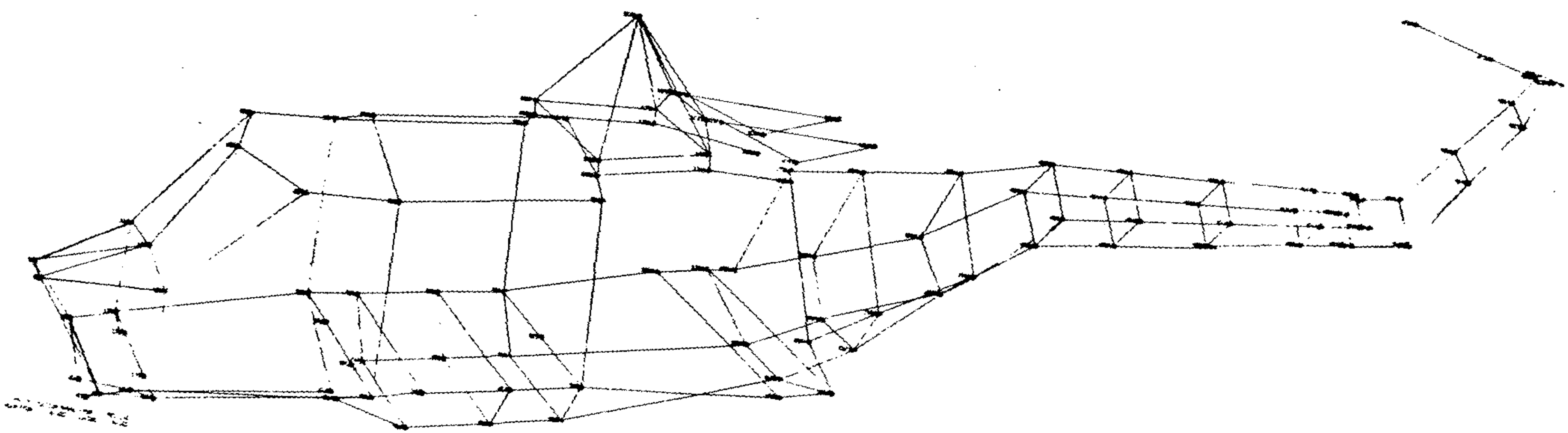


FIGURE 1A
PLOTEL REPRESENTATION
OF RESIDUAL STRUCTURE

FIGURE 2 FINITE ELEMENT MODEL

The principal element types used in the model were as follows:

BAR	Beam element with extensional, torsional and bending properties together with associated shears. This element was used to represent beams and formers.
ROD	Simplified BAR element with extensional and torsional properties only. In general, the torsional degrees of freedom of the elements were not used. This element was used to represent stringers, longerons and intercostals.
QDMEM2	Two-dimensional quadrilateral element which resists in-plane shear and membrane action. This element was used in preference to the shear panel which was used in the Stress Office model to represent the skin elements and the sandwich construction panels.
TRMEM	Triangular two-dimensional membrane element with finite in-plane stiffness and zero bending stiffness. This element was used where the geometry was better suited to triangular panels.
QUAD4	Two-dimensional quadrilateral element with in-plane membrane stiffness and bending and transverse shear stiffness. This element was superimposed on the SHEAR elements which constituted the sandwich panels in order to represent the bending and transverse shear properties. The membrane action was set to zero and anisotropic material properties were input to represent the different bending and transverse shear action.
TRIA3	Two dimensional triangular element. The triangular complement of the QUAD4 element.
ELAS2	Scalar spring elements used to represent the engine/gearbox couplings.

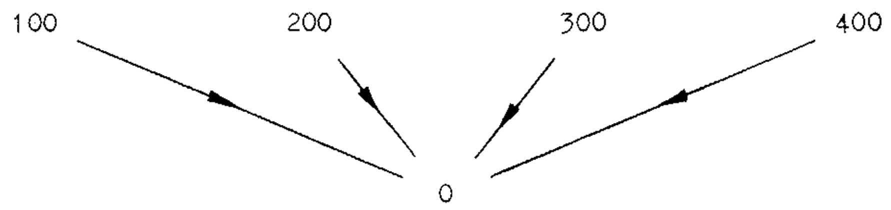
The model had in the order of 5,000 degrees of freedom and 1040 grid points, and was composed of the following elements:

BAR	196
ELAS2	8
QUAD4	273
ROD	1414
QDMEM2	804
TRIA3	108
TRMEM	146

A total of 2949 structural elements were used and 2111 CONM2 elements were used for the non-structural mass items.

FIGURE 3 THE ANALYSIS OF PRODUCTION NAVAL LYNX

Superelement Tree for Single-Level Analysis



100 - the superelement representation for the forward fuselage area.

200 - the superelement representation for the rear fuselage area.

300 - the superelement representation for the gearbox and engines.

400 - the superelement representation for the tailcone area.

0 - the residual structure.

Stages of Analysis

1 - Data check and production of superelement map

2 - Split data base for the generation of superelements 100, 200, 300 and 400.

SEMG = 100, 200, 300, 400	Internal Points DB02 External Points BD01

3 - Split data base for the assembly of the stiffness and mass matrices for superelements 100, 200, 300 and 400.

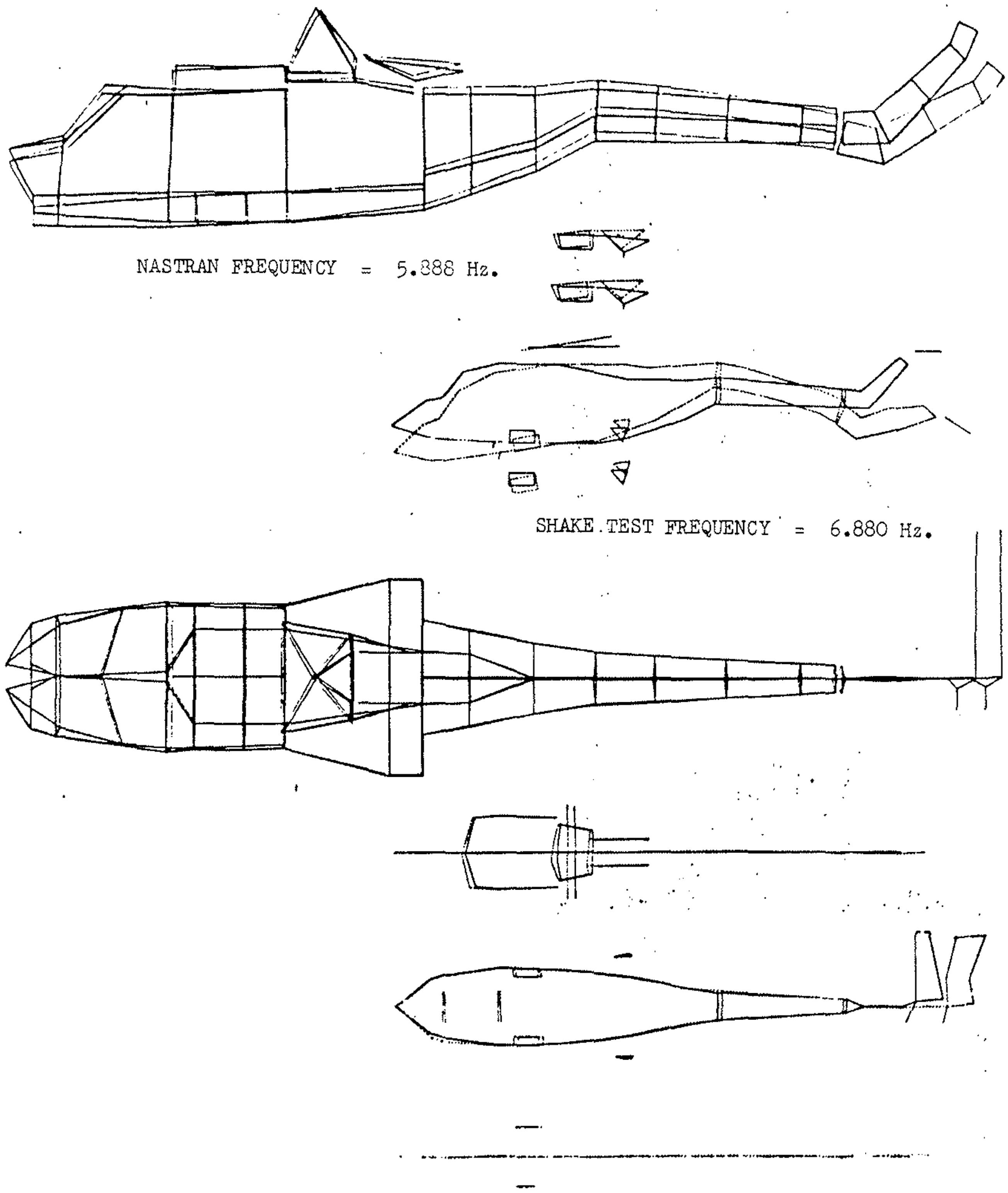
SEMA = 100, 200, 300, 400	Internal Points DB02
SELA = 100, 200, 300, 400	External Points DB01

4 - Assembly of the stiffness and mass matrices for the residual structure, the second reduction (Guyan Reduction) of the residual structure to the final dynamic analysis degrees of freedom and the eigenvalue extraction in Data Base One. DB01

FIGURE 4 NASTRAN AND EXPERIMENTAL MODE SHAPE COMPARISONS

NASTRAN MODE NO.	FREQUENCY HZ. NASTRAN	FREQUENCY HZ. EXPERIMENT	MODAL DAMPING FACTOR Q	MODE SHAPE DESCRIPTION
1	0.001			'RIGID BODY' LATERAL TRANSLATION
2	0.022			'RIGID BODY' YAW ROTATION
3	0.030			'RIGID BODY' FORE AND AFT TRANSLATION
4	0.052			'RIGID BODY' PITCH ROTATION
5	0.106			'RIGID BODY' VERTICAL TRANSLATION
6	0.341			'RIGID BODY' ROLL ROTATION
7	5.888	6.880	22.6	FUNDAMENTAL VERTICAL BENDING OF FUSELAGE
8	6.380	6.715	31.4	FUNDAMENTAL LATERAL BENDING OF FUSELAGE
9	14.856	16.050	10.4	SECOND VERTICAL BENDING OF FUSELAGE
10	16.696	14.510	55.7	VERTICAL BENDING OF TAILPLANE
11	19.482	20.390	36	THIRD VERTICAL BENDING OF FUSELAGE
12	20.554	18.200	40	FORE AND AFT BENDING OF TAILPLANE
		23.350	9.4	PORT ENGINE LATERAL BENDING
13	21.813	25.650	12.6	LATERAL ENGINE MOTION : ANTISYMMETRIC
14	22.139	25.970	9.9	LATERAL ENGINE MOTION : SYMMETRIC
15	24.334	28.920	55	FRONT FUSELAGE TORSION
16	30.375	23.470	22.5	TAILCONE/FIN BENDING
17	31.200	28.730	10.7	FOURTH VERTICAL BENDING OF FUSELAGE

FIGURE 5 COMPARISON OF NASTRAN WITH SHAKE TEST
FIRST ELASTIC MODE



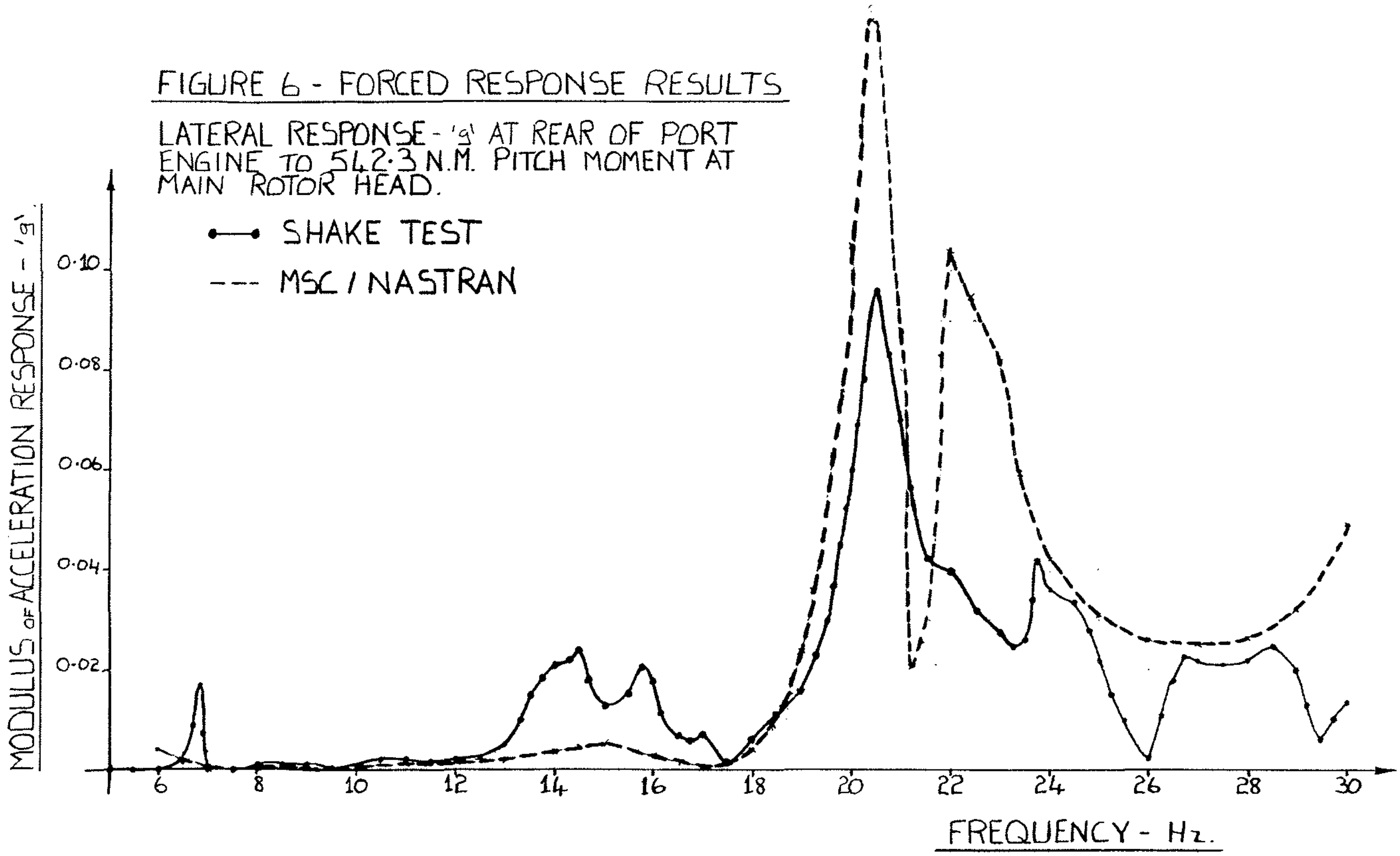


FIGURE 6 - FORCED RESPONSE RESULTS
LATERAL RESPONSE - 'g' AT REAR OF PORT ENGINE TO 542·3 N.M. PITCH MOMENT AT MAIN ROTOR HEAD.
SHAKE TEST
MSC / NASTRAN
MODULUS OF ACCELERATION RESPONSE - 'g'.
0·10
0·08
0·06
0·04
0·02
6
8
10
12
14
16
18
20
22
24
26
28
30
FREQUENCY - Hz.

FIGURE 7 THE RECOMMENDED FUTURE APPLICATION OF FINITE ELEMENTS FOR DYNAMICS