Integration of **NASTRAN** Into Helicopter Airframe Design/Analyses

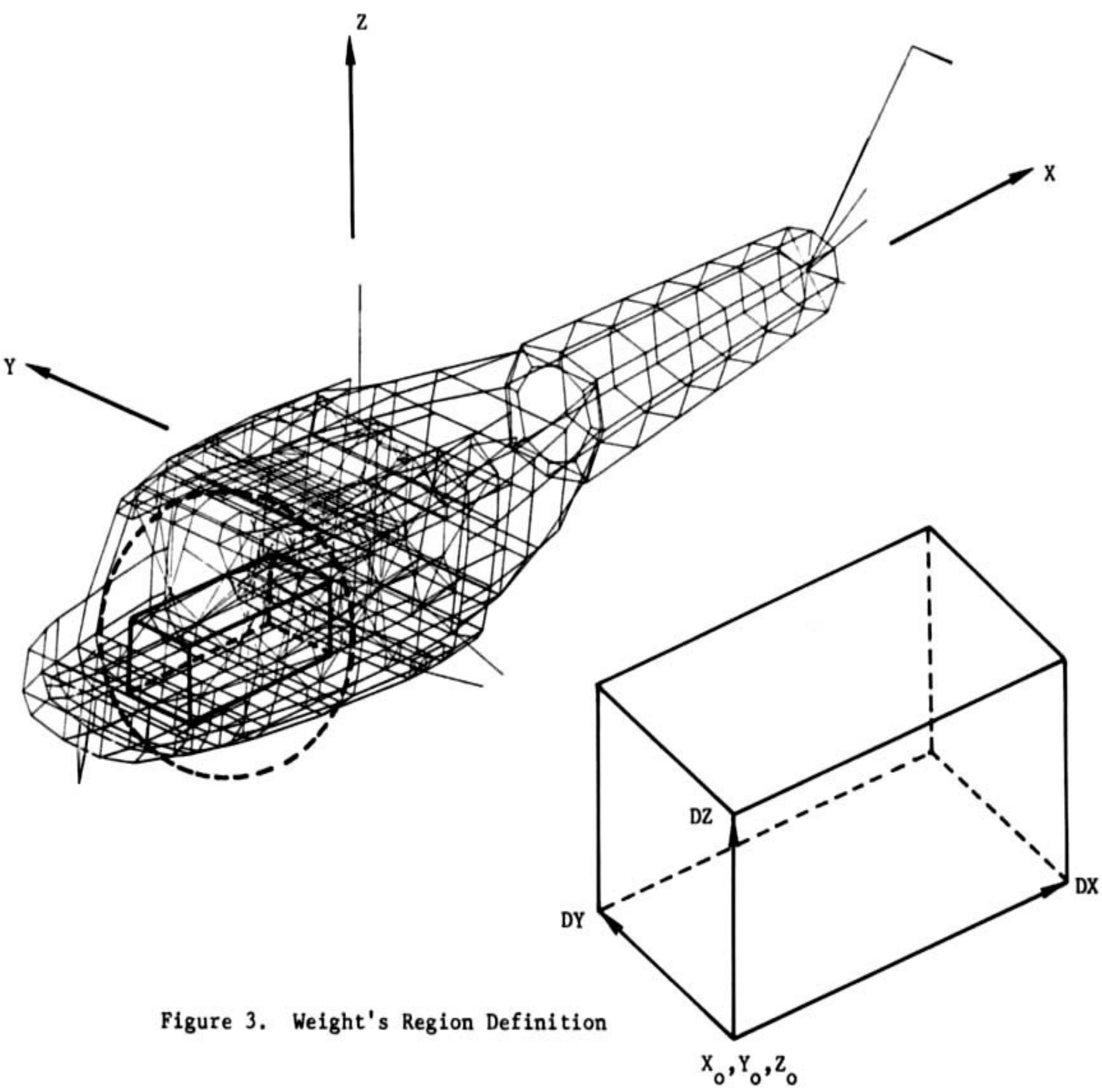


Figure **1:** Helicopter **FEM**

By Anonymous

Preface

The Federal Government's **NASTRAN** (its Finite Element Program) doesn't exist any more and hasn't for many years. Now only commercial programs such as **MSCTRAN**, or **COSMOSM** are employed.

The importance of this paper are the equations for the centroidal mass properties, Eq.'s **1**, **2**, **3**, **4**; and the inertia mass properties, Eq.'s 5, 6, 7, 8, 9, 10; employed in the ***n*-body Problem Closed-form Solution** monograph.

Note also Equation **18** on their page 3 should be I_{xyH} = (not I_{zyH} as published); it has been corrected here. Another (critical) error is $M = I\alpha + \omega^2 I$, where the last term is the effect of the angular velocity. This last term Henry and Dave left out (see last paragraph of their page 3), but does not effect the webpage's work. To correct this see for example the equations in "***Dynamics***," in Vol. **7**, pp. 821-825, and in particular page 825, of the 1968 Edition of the **Encyclopaedia Britannica**.; or later Editions. Or refer to any dynamics textbook, say like Ferdinand L. **SINGER**'s **ENGINEERING MECHANICS, Statics And Dynamics**, 3rd Ed., *Harper & Row*, 1975 (which is one of the best).

For those not versed in Finite Element knowledge, the four short, elementary but excellent articles by Neville F. **RIEGER** and Jefferey M. **STEELE**, as published in ***Machine Design***, are available**:**

- "***The Basics of Finite-Element Modeling***," pp. 185-170, April 9, 1981;
- "***Basic Course in Finite-Element Analysis, Basic Concepts***," pp. 103-107, June 25, 1981;
- "... ***Modeling***," pp. 153-157, July 9, 1981;
- "... ***Advanced Concepts***," pp. 97-100, July 23, 1981.

THE INTEGRATION OF NASTRAN INTO HELICOPTER AIRFRAME DESIGN/ANALYSIS

D. A. GALLIAN

AND

H. E. WILSON

BELL HELICOPTER COMPANY

PRESENTED AT THE 29th ANNUAL NATIONAL FORUM
OF THE
AMERICAN HELICOPTER SOCIETY
WASHINGTON, D.C.
MAY 1973

PREPRINT NO. 780

THE INTEGRATION OF NASTRAN INTO HELICOPTER AIRFRAME DESIGN/ANALYSIS

D. A. Gallian
Structures Engineer
and
H. E. Wilson
Structures Engineer
Bell Helicopter Company
Fort Worth, Texas

Abstract

The integration of the finite element computer program NASTRAN (NASA Structural Analysis) into the helicopter design/analysis loop is described. While the program was successfully used and checked out in its off-the-shelf version, increased usage and familiarity focused attention on several areas which seemed unnecessarily time consuming. To make this powerful analytical tool a cost-effective engineering method, the problem areas were attacked. To reduce engineering modeling time, automated data generation of structures was developed. The problem of nonstructural mass distribution was also solved by the use of preprocessor programs which operate on the already available helicopter weight tabulation in a MIL-STD form. Post-processor programs were developed to allow selective data presentation of the significant load conditions, high stress members, or other areas of particular interest. The methods employed in developing these pre- and post-processor programs and their interface with NASTRAN are discussed in detail.

Notations

F	-	force
I	-	moment of inertia
g	-	acceleration due to gravity
M	-	moment
m	-	mass
N	-	load factors
nreg	-	number of regions
nwt	-	number of weight regions
W	-	weight
$\underset{\sim}{X}$,Y,Z	-	center of gravity
$\ddot{\theta}$	-	pitching acceleration
$\ddot{\phi}$	-	rolling acceleration
$\ddot{\psi}$	-	yawing acceleration

Subscripts

H	-	helicopter
r	-	region
w	-	weight item
x,y,z	-	coordinate axis

Presented at the 29th Annual National Forum of the American Helicopter Society, May 1973.

Background

Before the availability of large finite element programs, internal loads were calculated from a two-dimensional shear and moment diagram. After the development of NASTRAN, MAGIC, and other similar prgorams, the laborious procedures associated with the redundant analysis required to distribute these shears and moments were eliminated. However, before NASTRAN can be executed, the helicopter airframe must be represented with a three-dimensional finite element model, as shown in Figure 1. Many man-hours are required to represent the structure with a mathematical model which could contain as many as 3000 input cards. With such a large amount of input data, the possibility of modeling error is greatly increased.

In addition to the actual modeling, the problem of distributing structural and nonstructural weight to the appropriate areas of finite element model proved to be a time consuming and tedious task requiring many judgmental decisions. If NASTRAN was to be used efficiently for the calculation of internal loads, an automated procedure for model generation and distribution of weight items to the structural model had to be devised.

With a mathematical model of the airframe available, there is a tendency to calculate internal loads for many more loading conditions than was previously feasible. For this influx of data to be efficiently used, an automated procedure for determining critical member loads was needed.

Model Generation

The time required to prepare a three-dimensional finite element model of an entire helicopter airframe is about one man-month. This amount of time may prove to be prohibitive, thus several data generator programs have been developed that can reduce the required modeling time of many structural applications by as much as 90 percent. These computerized procedures also reduce the number of modeling mistakes that can occur, thereby reducing the time needed to check out the finite element model.

One of the data generator computer programs available is a multipurpose program that can create models of rotor blades, tail booms, elevators, or any other shell-type structure. The program calculates the grid location and the element connectivity for any straight or linearly tapered structure. Input data required for the execution of the model generator program includes the coordinates of each end station and location for each interim station. A linear extrapolation is made to obtain the coordinates for these stations. If a sheet-stringer construction is being analyzed, the stringers and the skins are generated along the grid points. Semi- or full-monocoque constructions can be generated also. The data generated in each case consists of NASTRAN grid cards and NASTRAN element-connectivity cards.

An example of a generated finite model of an AH-1G Cobra tail boom, elevator and vertical fin is shown in Figure 2.

Distribution of Weights

The problem of distributing weight items to the three-dimensional model was solved by writing an interface program to NASTRAN that yields an inertial representation of the helicopter under consideration. The mass-generation option in NASTRAN could not be used because much of the helicopter weight is nonstructural. Rather than distinguishing between the structural and nonstructural weight items, they are both handled as one. The required inputs to this computer program consist of the NASTRAN Grid Deck, an AN sequence weights tape,[1] and a set of inertial regions. The AN sequence tape contains the weight, location, and size of each structural and nonstructural weight item, listed by their AN code number. The option is provided to input and operate on weight items which are not currently included on the AN sequence tape. This is valuable when building weight configurations to establish a center-of-gravity envelope.

The regions, as defined in Figure 3, are rectangular parallelepipeds which combine to totally enclose the model's volume. The inertial regions are used to specify the grid points used for the distribution of each item's weight. The average size of an inertial region may vary from helicopter to helicopter, but generally it may take 150 to 200 to correctly represent the inertial distribution. Included in the program is the option to distribute certain weight items individually. This option may be exercised to correctly represent the unique distribution of such weights as the engine, rotor hub, external stores, or transmission. The AN code and the grid points that are associated with each special item are prescribed on a data card. Throughout the remainder of the analysis, this specially handled weight is treated as a regular region. During the execution of the program, these weights are distributed to the model. The center of gravity and mass properties are calculated for each region using the weight items enclosed in that region. Items whose volumes are not completely enclosed within a region boundary have a volumetric proportion of their weight used in this calculation. The center of gravity for region j is calculated as follows:

$$X_{rj} = \left(\sum_{i=1}^{nwt} m_{ji} X_{wi} \right) / m_{rj} \qquad \ldots.(1)$$

$$Y_{rj} = \left(\sum_{i=1}^{nwt} m_{ji} Y_{wi} \right) / m_{rj} \qquad \ldots.(2)$$

$$Z_{rj} = \left(\sum_{i=1}^{nwt} m_{ji} Z_{wi} \right) / m_{rj} \qquad \ldots.(3)$$

where

$$m_{rj} = \sum_{i=1}^{nwt} m_{wi} \qquad \ldots.(4)$$

The mass moments of inertia for region j about its center of gravity are given by

$$I_{xxrj} = \sum_{i=1}^{nwt} m_{ji} \left| (Y_{wi} - Y_{rj})^2 + (Z_{wi} - Z_{rj})^2 \right| \quad ..(5)$$

$$I_{yyrj} = \sum_{i=1}^{nwt} m_{ji} \left| (Z_{wi} - Z_{rj})^2 + (X_{wi} - X_{rj})^2 \right| \quad ..(6)$$

$$I_{zzrj} = \sum_{i=1}^{nwt} m_{ji} \left| (X_{wi} - X_{rj})^2 + (Y_{wi} - Y_{rj})^2 \right| \quad ..(7)$$

$$I_{xyrj} = -\sum_{i=1}^{nwt} m_{ji} \; (X_{wi} - X_{rj})(Y_{wi} - Y_{ri}) \qquad \ldots.(8)$$

$$I_{xzrj} = -\sum_{i=1}^{nwt} m_{ji} \; (X_{wi} - X_{rj})(Z_{wi} - Z_{ri}) \qquad \ldots.(9)$$

$$I_{yzrj} = -\sum_{i=1}^{nwt} m_{ji} \; (Y_{wi} - Y_{rj})(Z_{wi} - Z_{rj}) \qquad \ldots(10)$$

Significant inertias, I_o, for special weight items are input separately and added to the inertia of the region which encloses them.

This process is repeated for each inertial region. The region properties are then used to calculate the center of gravity and mass moments of inertia for the helicopter. The center of gravity used in calculating inertial loads is given by

$$m_H = \sum_{j=1}^{nreg} m_{rj} \qquad \ldots(11)$$

$$X_H = \left(\sum_{j=1}^{nreg} m_{rj} X_{rj}\right)/m_H \qquad \ldots(12)$$

$$Y_H = \left(\sum_{j=1}^{nreg} m_{rj} Y_{rj}\right)/m_H \qquad \ldots(13)$$

$$Z_H = \left(\sum_{j=1}^{nreg} m_{rj} Z_{rj}\right)/m_H \qquad \ldots(14)$$

The helicopter's inertias, used as a check to the final mass distribution, are given by

$$I_{xxH} = \sum_{j=1}^{nreg} m_{rj} \left[(Y_{rj} - Y_H)^2 + (Z_{rj} - Z_H)^2\right] + \sum_{j=1}^{nreg} I_{xxrj} + \sum_{j=1}^{n} I_{xxoj} \qquad \ldots(15)$$

$$I_{yyH} = \sum_{j=1}^{nreg} m_{rj} \left[(Z_{rj} - Z_H)^2 + (X_{rj} - X_H)^2\right] + \sum_{j=1}^{nreg} I_{yyrj} + \sum_{j=1}^{n} I_{yyoj} \qquad \ldots(16)$$

$$I_{zzH} = \sum_{j=1}^{nreg} m_{rj} \left[(X_{rj} - X_H)^2 + (Y_{rj} - Y_H)^2\right] + \sum_{j=1}^{nreg} I_{rzzj} + \sum_{j=1}^{n} I_{ozz} \qquad \ldots(17)$$

$$I_{xyH} = -\sum_{j=1}^{nreg} m_{rj} (X_{rj} - X_H)(Y_{rj} - Y_H) + \sum_{j=1}^{nreg} I_{xyrj} + \sum_{j=1}^{n} I_{xyoj} \qquad \ldots(18)$$

$$I_{xzH} = -\sum_{j=1}^{nreg} m_{rj} (X_{rj} - X_H)(Z_{rj} - Z_H) + \sum_{j=1}^{nreg} I_{xzrj} + \sum_{j=1}^{n} I_{xzoj} \qquad \ldots(19)$$

$$I_{yzH} = -\sum_{j=1}^{nreg} m_{rj} (Y_{rj} - Y_H)(Z_{rj} - Z_H) + \sum_{j=1}^{nreg} I_{yzrj} + \sum_{j=1}^{n} I_{yzoj} \qquad \ldots(20)$$

where n = number of significant inertias of special weight items.

A redundant truss composed of bars having equal section properties is automatically constructed between the region's center of gravity and each grid point in that region. Using kinematics, the translational and rotational accelerations of each region are calculated for unit gravitational and angular accelerations applied to the center of gravity of the helicopter. By applying these accelerations to the mass at the region's centroid, the reactions, which are the desired inertial forces, are calculated at the grid points.

The forces at the region's centroid are given by a 6 by 6 matrix. Columns 1 through 3 represent the forces due to unit load factors in the X, Y, and Z directions respectively. Columns 4 through 6 contain the forces due to a 1 radian-per-second-squared angular acceleration about the helicopter center of gravity. The angular accelerations correspond to roll, pitch, and yaw respectively. Rows 1 through 6 contain Fx, Fy, Fz, Mx, My, and Mz respectively, for each unit load condition. The terms in the matrix for the j^{th} region are:

$$
\begin{array}{c|cccccc}
 & N_x & N_y & N_z & \ddot{\phi} & \ddot{\theta} & \ddot{\psi} \\
\hline
F_x & g\,m_{rj} & 0 & 0 & 0 & m_{rj}(Z_{rj} - Z_H) & m_{rj}(Y_H - Y_{rj}) \\
F_y & 0 & g\,m_{rj} & 0 & m_{rj}(Z_H - Z_{rj}) & 0 & m_{rj}(X_{rj} - X_H) \\
F_z & 0 & 0 & g\,m_{rj} & m_{rj}(Y_{rj} - Y_H) & m_{rj}(X_H - X_{rj}) & 0 \\
M_x & 0 & 0 & 0 & I_{xxrj} & I_{xyrj} & I_{xzrj} \\
M_y & 0 & 0 & 0 & I_{xyrj} & I_{yyrj} & I_{yzrj} \\
M_z & 0 & 0 & 0 & I_{xzrj} & I_{yzrj} & I_{zzrj}
\end{array}
$$

The rotational degrees of freedom associated with the grid points are condensed out using a matrix reduction. The element stiffness matrix is added to the stiffness matrix for the region. The equilibrium equations for a region can be expressed as follows:

$$
\begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix} \begin{Bmatrix} u \\ o \end{Bmatrix} = \begin{Bmatrix} P \\ f \end{Bmatrix} \qquad \ldots(21)
$$

where

$\{u\}$ = the displacement associated with the region center of gravity

$\{P\}$ = the applied loads at the region center of gravity

$\{f\}$ = the inertial loads forces acting at the grid points

$[K]$ = region stiffness matrix

In solving the equation

$$
\{f\} = [K_{21}]\ [K_{11}]^{-1}\ \{P\} \qquad \ldots(22)
$$

since

$$
\{u\} = [K_{11}]^{-1}\ \{P\} \qquad \ldots(23)
$$

the inertial reactions are written on a magnetic tape in a NASTRAN Force Card format, which is then used as an input to NASTRAN.

To ensure a correct inertial representation, several equilibrum checks are made. The forces due to a unit load factor in the x direction are summed. Forces in the y and z directions should sum to zero, while the sum of the forces in the x direction should equal the helicopter's weight. Summations of forces due to unit load factors in the y and z directions are likewise performed. A check of the inertial distribution due to unit angular accelerations about the x (roll) axis is obtained by summing moments about the x axis. The sum should be equal to the rolling inertia of the helicopter. In a similar manner, the inertial forces due to unit angular accelerations when summed about the y (pitch) axis and z (yaw) axis should equal the pitching inertia and yawing inertia, respectively.

Inertial forces due to a unit angular acceleration about the x axis are further checked by summing moments about the y and z axis. The resulting moments should equal to the products of inertia I_{xyH} and I_{xzH} respectively. Likewise, by summing moments for pitching and yawing accelerations all products of inertia can be checked against those previously calculated.

Once the unit distribution has been established, the specific representation for the design load conditions can be determined. The externally applied loads for a given design load condition (rotor thrust, airloads, etc.) are obtained and summed about the helicopter center of gravity. The sum of the forces and moments are then used to calculate the load factors and angular accelerations of the airframe associated with each loading condition.

$$
\begin{Bmatrix} N_x \\ N_y \\ N_z \end{Bmatrix} = \frac{1}{W_H} \begin{Bmatrix} F_{xH} \\ F_{yH} \\ F_{zH} \end{Bmatrix} \qquad \ldots.(24)
$$

$$\begin{Bmatrix} \ddot{\phi} \\ \ddot{\theta} \\ \ddot{\psi} \end{Bmatrix} = \begin{bmatrix} I_{xxH} & I_{xyH} & I_{xzH} \\ I_{xyH} & I_{yyH} & I_{yzH} \\ I_{xzH} & I_{yzH} & I_{zzH} \end{bmatrix}^{-1} \begin{Bmatrix} M_{xH} \\ M_{yH} \\ M_{zH} \end{Bmatrix} \quad ..(25)$$

These load factors and angular accelerations are used as scale factors, which multiply the inertial forces calculated for the six-unit conditions to yield the correct inertial reactions for the different loading conditions to be analyzed. A diagram illustrating the process for determining this inertial representation is given in Figure 4.

An option to the program is to calculate masses at the grip points. The masses are generated strictly from geometric considerations. For a given region, the mass at grid point i is given by

$$m_i = \frac{m_r/L_i}{\sum_{j=1}^{nwt} 1/L_j} . \quad ...(26)$$

where:

m_i = the mass at grid point i

m_r = region mass

L_i = distance from grid point i to the region center of gravity

nwt = number of weight

By punching these masses on cards and including them with the NASTRAN data deck, a dynamic analysis can be performed on the airframe.

NASTRAN

NASTRAN is a large multipurpose finite element computer program capable of performing both a static as well as a dynamic structural analysis. It uses the displacement (stiffness) approach. The structural behavior of the helicopter airframe is represented by various structural elements, including rods, bars, and shear panels. The stiffness matrix for each element is placed in a large matrix, yielding a mathematical model of the airframe. The scaled inertial loads, together with the externally applied loads associated with each loading configuration, are input to NASTRAN. A set of determinant constraints are applied, which allow the stiffness matrix to be decomposed.

NASTRAN output consists of deflections at the grid points, element forces and stresses, plots of the structural model, and constraint forces. The plots are used as an aid in detecting modeling errors as well as providing visualization of deflected shapes. If the determinant boundary conditions are applied properly, the constraint forces will be zero because the applied forces are balanced by the inertial loads. If the constraint forces are not zero, then either the inertial representation is incorrect or the model is overconstrained.

After the initial computer run has been made, the decomposed stiffness matrix is saved on a magnetic tape. This decomposed matrix is used on successive loading configurations to be analyzed, thus yielding reduced run times.

Data Processing

NASTRAN output in its original form is of limited use in performing an airframe stress analysis. For example, axial rod loads output by NASTRAN represent average values and do not reflect the interaction of the rod with an adjacent shear panel. In order to make NASTRAN a more useful tool for analysis, a NASTRAN post-processor program was written. The main function of this program is to reformat NASTRAN output and to perform limited calculations to aid the analyst in the interpretation of the data.

The first step in revising NASTRAN output to a form which may be readily used for analysis is to establish shear flows acting along the edges of shear panel elements. This is accomplished by first resolving the diagonal loads output for a shear panel into components acting along the sides of the panel as shown in Figure 5. To maintain compatibility with the NASTRAN USERS MANUAL,[2] the diagonal forces are assumed to be positive when acting in an outward direction. To assure that the force components acting along the sides of a shear panel add vectorially to yield the diagonal load, the following numerical procedure is established. Considering grid point 1, unit vectors acting in the F1, Fx1, and Fy1 directions are designated as A, B, and C respectively. The force components at grid point 1 can be determined by using vector dot products:

$$Fx1 = (\vec{A}\cdot\vec{B} - \vec{C}\cdot\vec{B}) \quad (|\vec{F1}|) \quad ....(27)$$

$$Fy1 = (\vec{A}\cdot\vec{C} - \vec{B}\cdot\vec{C}) \quad (|\vec{F1}|) \quad ....(28)$$

where $|\vec{F1}|$ = the magnitude of vector $\vec{F1}$. Similar calculations are performed to yield the force components at the remaining shear panel vertices. The signs on the components are then adjusted to conform to the positive sign convention shown in Figure 6. Average shear flows on each side of the shear panel are calculated by dividing the resultant loads acting on a side by the length of that side.

Given the shear flows acting along the sides of the various shear panel elements contained within the finite element model, the axial loads in the rod and bar elements can be adjusted to simulate distributed axial loading by considering the shear flows on adjacent shear panels. Without this adjustment, the NASTRAN output axial loads for rod and bar elements would not give a true picture of the helicopter loading. Referring to Figure 7, the adjusted axial loading on the upper end of member 6 is given by the expression

$$P_{axial} = P6 + (1/2)\,F3 + (1/2)\,F4 \qquad \ldots.(29)$$

Likewise the adjusted axial load on the lower end of member 6 is given by the expression

$$P_{axial} = P6 - (1/2)\,F3 - (1/2)\,F4 \qquad \ldots.(30)$$

Again referring to Figure 7, the NASTRAN post-processor selects only shear panels 3 and 4 for use in these calculations since they are the only panels having two nodes in common with element 6.

After the magnitude of the adjusted axial loads for the rod and bar elements have been calculated, the NASTRAN post-processor then determines if the newly calculated axial load is tensile or compressive. Given that the local X axis for rod and bar elements is positive in the direction from the first node to the second,[2] the program compares the adjusted axial load with the following criteria to determine if it is tensile or compressive.

At end "A"

If $P_{adj} < 0$, then P_{adj} is a tensile force

If $P_{adj} > 0$, then P_{adj} is a compressive force

At end "B"

If $P_{adj} > 0$, then P_{adj} is a tensile force

If $P_{adj} < 0$, then P_{adj} is a compressive force

This process frees the engineer from keeping track of the sign convention used by the modeler and allows him to concentrate on the analysis.

Additional information supplied to the analyst by the NASTRAN post-processor consists of the maximum shear stress acting on a shear panel (calculated by taking the shear flow of largest magnitude acting on that panel and dividing by the panel thickness), the axial stress of each end of a rod element (calculated by dividing the adjusted axial load at each end by the cross-sectional area of the rod), and the element section properties transferred to either the element or helicopter coordinate system. As an example, the output for bar elements consists of the shear loads (SY, SZ) and moments (TORQUE, MY, MZ) as well as the adjusted axial force at each end of the bar. All shears, moments, and section inertias are output in both the element coordinate system or the helicopter coordinate system. This option allows the analyst to work independently of the modeling engineer since the sign convention used in the modeling need not be known before the analysis is performed.

Sample output for selected elements is shown in Figure 8.

Even though the data needed to analyze a particular structural member are at hand, the job of reviewing the output and selecting the critical loads acting on a structural element can be a laborious task, especially when the loads from several loading conditions must be reviewed. To aid in this task, a program has been devised to scan the load data and select, based on a programmed criteria for each type of element, the critical condition for each structural element.

For rod elements, the critical load selector program assumes that rod elements are able to carry axial loads only. Thus, depending upon the geometry and material composition of a particular rod, the critical load will be either the maximum tensile or compressive axial load acting on that rod. The critical load selector program compares the adjusted axial loads computed for each rod element for the loading configurations under investigation and selects the maximum tension and compression loads at each end of each rod. These loads represent the critical loading for that rod.

If so defined in the NASTRAN bulk data, bar elements can carry an axial force, torsion, bending in two perpendicular planes and the associated shears. Since the critical stress for a particular bar will depend upon combination of these loads, the critical loading condition cannot be selected by keying on any one force or moment. Because of this, no loading condition can be labeled as noncritical unless the adjusted axial force, torsional and bending moments, and the shearing forces at each end of the bar are all individually of less magnitude than the corresponding forces and moments produced by another loading condition. For example, forces and moments FX1, FY1, FZ1, MX1, MY1, MZ1, and FX2, FY2, FZ2, MZ2, MY2, MZ2 are produced at end "A" of a bar by loading conditions 1 and 2 respectively. If FX1 has the same sign as FX2, but is of greater magnitude, FX2 is said to be less critical than FX1. Similar comparisons are made for the remaining forces and moments on both ends of the bar. Should loading condition 2 fail to produce a force or moment of greater magnitude than a corresponding force or moment produced by condition 1, loading condition 2 is considered noncritical.

The critical load selector program scans the forces and moments output for each bar element and eliminates any noncritical loading condition. Forces and moments for the remaining loading conditions are listed for each end of each bar.

For plate elements with bending stiffness, the NASTRAN post-processor output consists of the biaxial stresses (SX, SY, SXY) and the principal stresses S_{max}, S_{min}, T_{max}) at the outer fibers. The critical load condition for a material failure can be predicted using a distortion energy criterion. The strain energy due to distortion is calculated for each loading condition to aid in selecting the critical condition. The analyst must also be aware of panel and shear instability and check to make sure that another loading condition is not critical because of an instability. For the sandwich panel, failure due to intercellular buckling and face wrinkling must also be analyzed.

Membrane elements in NASTRAN have a finite in-plane stiffness and zero-bending stiffness. Since these elements cannot carry a bending load, the stresses acting on a membrane element are considered to be constant through its thickness. Using the same critical loading criterion as bending plane elements, the critical loading selector program compares the NASTRAN post-processor output data for the loading conditions under investigation and eliminates any noncritical loading conditions. Both the biaxial and principal stresses are printed for each critical loading condition.

Shear panel elements are able to carry in-plane shearing forces only. The critical loading for a shear panel will be the largest shear flow acting on that panel. The critical loading selector program compares all of the shear flows from all loading conditions acting on a particular panel and selects the shear flow of greatest magnitude as the critical load for that panel. This process is then repeated for each of the remaining shear panels.

A diagram illustrating the process for determining the internal loads is given in Figure 9.

Conclusions

Engineers are rapidly adapting finite element methods to calculate internal loads. In order to fully utilize the structural analysis capability of NASTRAN, the pre- and post-processor programs outlined above have been employed. Through the use of this new approach, the ability to alter the design and observe the effects on the surrounding structure has become a cost-effective analysis process. Along with the ease of analyzing changing configurations, the opportunity to examine and reduce data from many loading conditions, at a small increase in time, ensures a more optimum and safer design.

References

1. MIL-STD-1374, Weight and Balance Data Reporting Forms for Aircraft (Including Rotorcraft), March 31, 1972.

2. NASA SP-222(01), NASTRAN USER'S MANUAL, McCormick, Caleb W., National Aeronautics and Space Administration, Washington, D.C., June 1972.

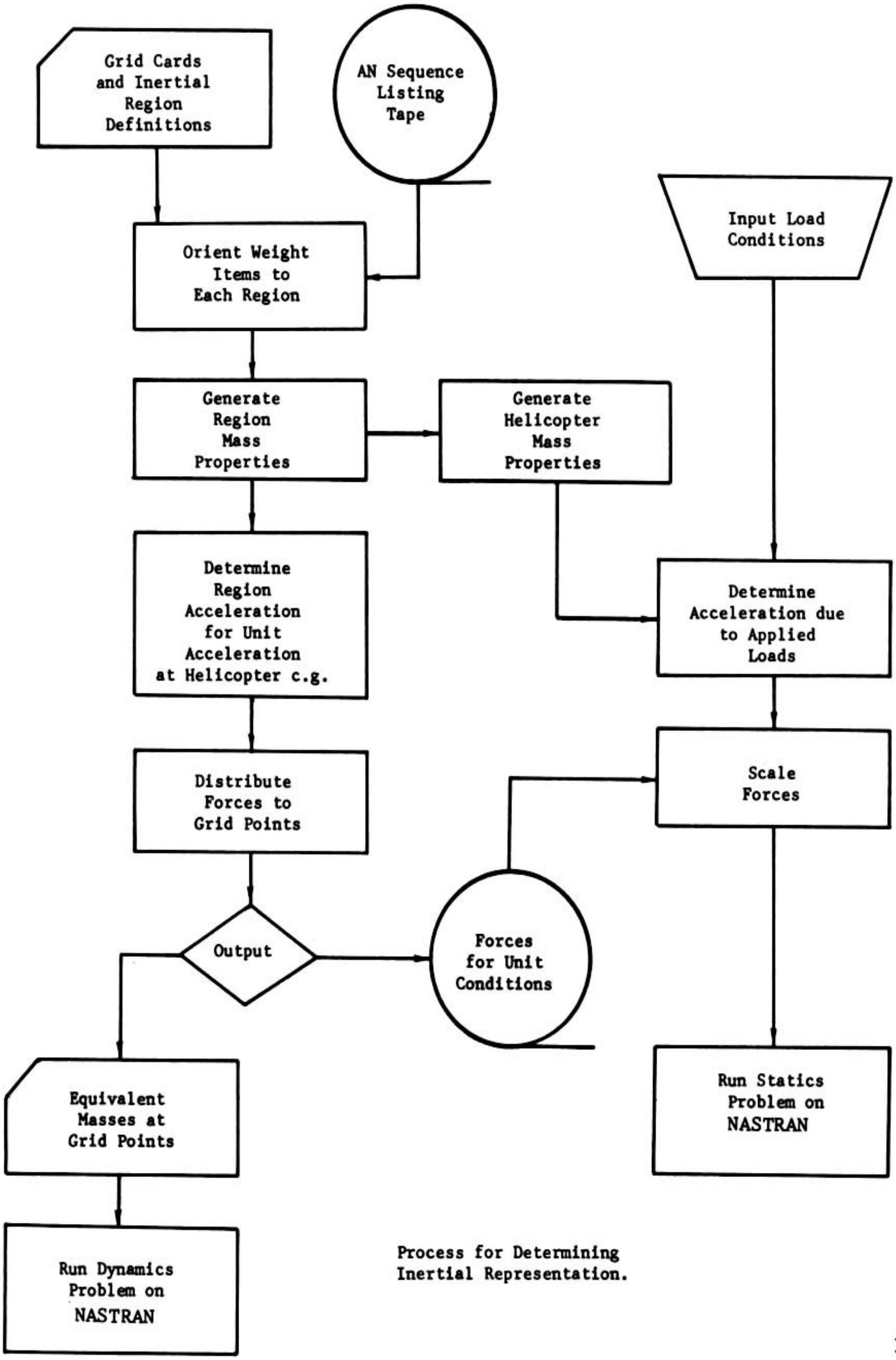
Bibliography

1. Finite Element Model Data Generator for Shells, Bell Helicopter Company Report 599-162-912, July 27, 1972.

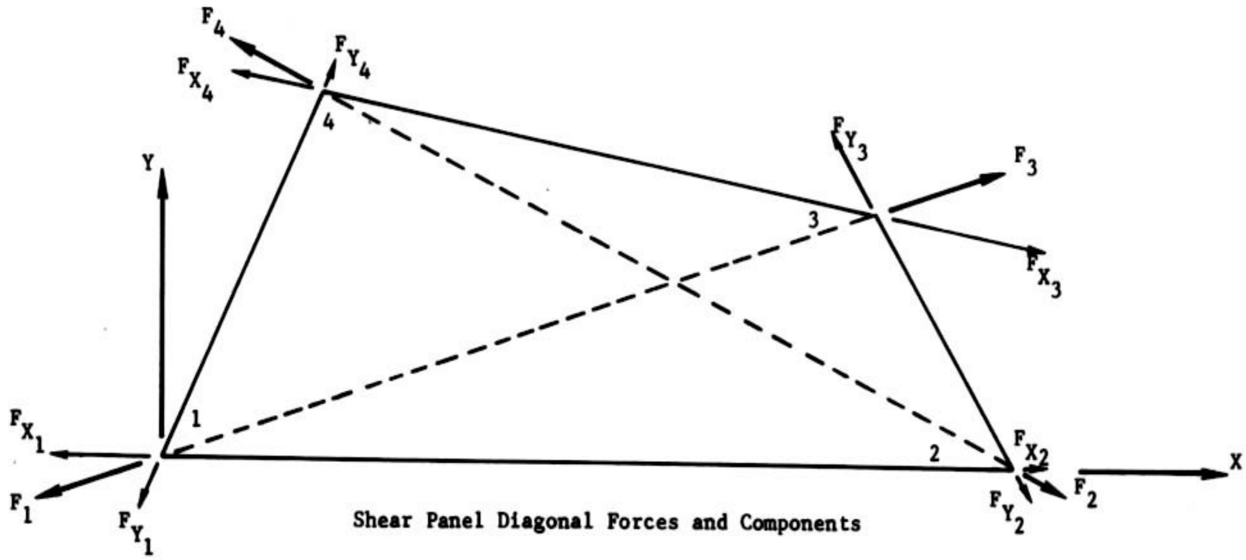
2. Mass and Inertial Distribution Calculated from a Weights Control Tape for a NASTRAN Finite Element Model, Bell Helicopter Company Report 599-162-911, March 30, 1972.

3. Greenwood, D. T., Principals of Dynamics, Prentice-Hall, Inc. 1965.

4. Airframe Output Options for Use with NASTRAN, Bell Helicopter Company Report 599-162-916, October 1, 1972.



Process for Determining Inertial Representation.



Shear Panel Diagonal Forces and Components

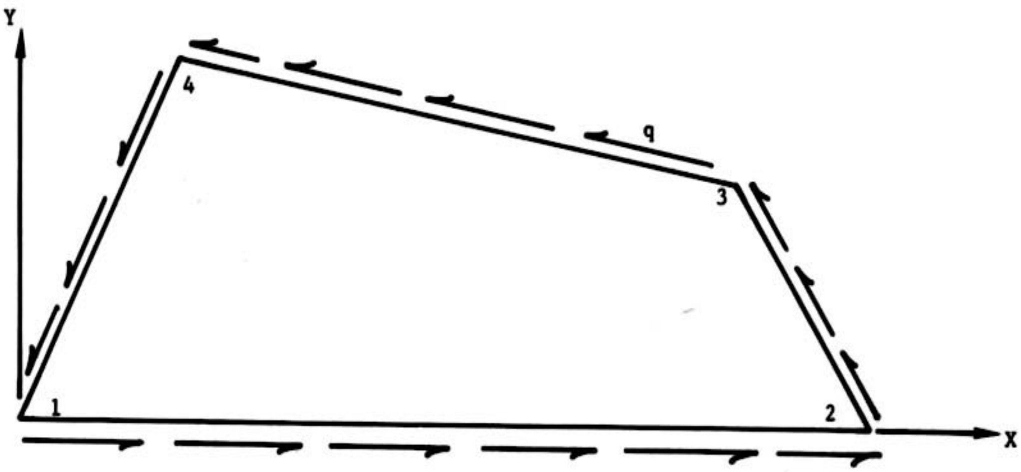


Figure 6. Positive Sign Convention for Shear Flows

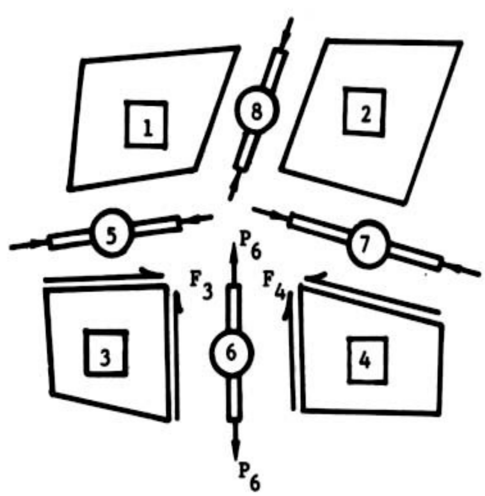


Figure 7. Shear Panel Junction

11

BY: D.A.GALLIAN BELL HELICOPTER MODEL: ALL PAGE:
CHECKED: H.E.WILSON HELI: ALL REPORT:

EXAMPLE OUTPUT FOR STRUCTURAL ELEMENTS
OUTPUT FOR POST PROCESSOR PROGRAM SESN01
CROD ELEMENT

ROD	NODE	AREA	AXIAL FORCE	AXIAL STRESS		ROD	NODE	AREA	AXIAL FORCE	AXIAL STRESS	
12	1	0.250	189.0	756.	COMP.	13	1	0.250	1407.2	5629.	TEN.
	2		300.0	1200.	COMP.		3		1296.1	5184.	TEN.
24	2	0.250	1592.8	6371.	TEN.	34	3	0.250	14.9	60.	COMP.
	4		1703.9	6816.	TEN.		4		200.0	800.	COMP.
35	3	0.250	1296.1	5184.	TEN.	46	4	0.250	1703.9	6816.	TEN.
	5		1000.0	4000.	TEN.		6		2000.0	8000.	TEN.
56	5	0.250	396.1	1584.	COMP.						
	6		100.0	400.	COMP.						

EXAMPLE OUTPUT FOR STRUCTURAL ELEMENTS
OUTPUT FOR POST PROCESSOR PROGRAM SESN01
CBAR ELEMENTS

		PROP. IN LOCAL COORDINATE SYSTEMS				OUTPUT IN LOCAL COORDINATE SYSTEM							
BAR	NODE	AREA	IZ	IY	J	AXIAL FORCE		SY	SZ	MY	MZ	TORQUE
12	1	0.250	0.005	0.005	0.010	1033	COMP.	0	0	-5	0	0
	2					999	COMP.	0	0	4	0	0
14	1	0.250	0.005	0.005	0.010	0	TEN.	0	-1	5	0	0
	4					54	TEN.	0	1	5	0	0
15	1	0.250	0.005	0.005	0.010	25	TEN.	0	0	0	0	0
	5					0	TEN.	0	0	0	0	0
23	2	0.250	0.005	0.005	0.010	50	TEN.	0	1	-4	0	0
	3					50	COMP.	0	-1	-4	0	0

EXAMPLE OUTPUT FOR STRUCTURAL ELEMENTS
OUTPUT FOR POST PROCESSOR PROGRAM SESN01
CSHEAR ELEMENTS

SHEAR PANEL	FROM NODE	TO NODE	PANEL THICK	SHEAR FLOW FORCE/LENGTH	MAX SHEAR STRESS
	1	2		5.64	
1234	2	3	0.032	-5.64	176.3
	3	4		5.64	
	4	1		-5.64	
	1	4		-0.23	
1485	4	8	0.032	0.23	7.1
	8	5		-0.23	
	5	1		0.23	

Figure 8. Internal Loads Sample Output

Figure 9. Process for Determining Internal Loads.