AUTOMATED FULLY-STRESSED DESIGN WITH NASTRAN

D. V. Wallerstein and G. W. Haggenmacher
Lockheed-California Company

SUMMARY

An automated strength sizing capability which has been introduced into the Lockheed-California Company's modified version of NASTRAN Level 15.1 is described. The technique determines the distribution of material among the elements of a structural model. Presently, the sizing is based on either a fully-stressed design or a scaled-feasible fully-stressed design. Results obtained from the application of the strength sizing to the structural sizing of a composite material wing box using material strength allowables is presented. These results demonstrate the rapid convergence of the structural sizes to a usable design. Future developments for the generation of incremental stiffness matrices for lay-up studies of composite material structures, and for aeroelastic analyses, are indicated.

INTRODUCTION

Automated strength sizing of a structural finite element model is a very important facet in preliminary structural design. Preliminary design involves many other disciplines, such as aeroeleastic and flutter design. The involvement of many disciplines and their associated data emphasizes the need for a well integrated system which achieves the requirement of rapid response to design changes with a minimum of data communications, time and errors. Since the Lockheed-California Company has adopted a modified NASTRAN Level 15.1 (NASTRAN-LCC) as the primary finite element structural analysis system within its general integrated structural design analysis system, the decision was made to incorporate the strength sizing program within NASTRAN-LCC. This provides the sizing program with access to all facilities of the Company's integrated system, including the data checking of the NASTRAN-LCC system and the Company's integrated data management system. This latter feature, in turn, provides for direct interfacing with the analyses of related disciplines such as aeroelastic loads and flutter. The automated strength sizing capability has been designed to determine efficient structural material distributions which meet strength requirements while reducing the structural mass. This process provides the data for subsequent aeroelastic evaluation.

The automated structural sizing is controlled by DMAP looping within NASTRAN's Rigid Format 1. The actual sizing is accomplished by the introduction into NASTRAN-LCC of two new functional modules and their associated input and output data. The new modules consist of a scan module which identifies the critical load conditions for each element and their associated algebraic maximum and minimum stresses, and a module that performs the actual sizing. The input data required consist of tables for the specification of design element allowables and size constraints. Two kinds of output are provided: the first is print output for use by the designer in evaluating the results of the automated structural sizing processes; the second is in the form of tables, containing the results of the sizing, which are stored into the data base for subsequent analyses.

SYMBOLS

Standard NASTRAN:

DMAP	Direct matrix abstraction approach
CASECC	Case control data table
ECT	Element connection table
EPT	Element property table
EST	Element summary table - a concatenation of ECT and EPT
g-Set	Grid point displacement set
KGGX	Stiffness matrix -g set
MAT2	Anisotropic material property definition input card
MPT	Material property table
OEF1	Output element force table
OES1	Output element stress table
PG	Static load vector -g set
SDR	Stress data recovery modules
SMA1	Structural matrix assembler module
SSG1	Static solution generator module
TA1	Table assembler module
UGV	Displacement vector matrix -g set

Nonstandard NASTRAN:

ECTC	Condensed element connection table
EPTC	Condensed element property table
EPTO	Original element property table

FCEOUT	Table of scanned forces
FSD	Fully-stressed design
FSDI	Structural sizing module
IEPTC	Incremental condensed element property table
KOLD	Stiffness matrix from previous iteration -g set
NASTRAN-LCC	Lockheed-California Company's version of NASTRAN
PIP	Element allowable and size constraint input card.
PIPT	Table formed from PIP cards
SFFSD	Scaled-feasible FSD
SR	Size ratio
STRSCN	Critical load determination module
STSOUT	Table of scaned stresses

INPUT DATA

To provide the input options for element redesign, a new Bulk Data Card (PIP) has been added to NASTRAN-LCC. The Property Input Parameter Table (PIPT) file is formed from these cards. The PIP input permits the following specifications for each element.

- Designation of an element as a design element.
- Two-directional stress allowables ($\bar{\sigma}xt$, $\bar{\sigma}xc$, $\bar{\sigma}yt$, $\bar{\sigma}yc$, $\bar{\tau}xy$).
- Minimum and maximum size constraints.
- Designation of the stress interaction curve to be used.

Only elements included in the PIPT table are design elements, all others are excluded from the sizing process.

In conjunction with two-directional stress allowables on the PIP card, the NASTRAN MAT2 card has been modified to include two-directional stress allowables.

THE SIZING PROCEDURE

A flow diagram of the pertinent features of the DMAP alter necessary for driving the sizing procedure is shown in figure 1. Except as otherwise noted, the main flow is vertical through the path marked A . Initially, path A performs a standard static internal loads solution. The STRSCN module then scans the resulting NASTRAN stress tables by load condition for each design element specified in the PIPT table. For each of these elements, a critical pair of load conditions and their corresponding stresses are determined and output in the STSOUT table. For example, the critical pair for a NASTRAN ROD element consists of the maximum tensile stress and its load condition number, and the maximum compressive stress and its load condition number.

The FSDI module then performs a comparison between the elements listed in the STSOUT table and those in the ECT table; and writes a new condensed connection table (ECTC) which contains only the design elements. Since the sizing logically requires a one-to-one correspondence between design element connection cards and design element property cards, a condensed property table (EPTC) is also formed. Next, the module performs the sizing of all design elements. The EPTC table is updated to reflect the new sizings; and an incremental condensed table (IEPTC) is formed. This latter table reflects the difference between the new and the old sizes.

Return to the top of the sizing loop is executed as shown by path B . Prior to entering TA1, however, the ECT table is equivalenced to the ECTC table and the EPT table is equivalenced to the incremental table IEPTC. As a result, SMA1 forms an incremental stiffness matrix which is added to the stiffness matrix from the previous iteration to form the new stiffness matrix for the current iteration.

When the conditional call to TA1 is entered at TA1-1, the ECT table is equivalenced to the ECTC table and the EPT table is now equivalenced to the condensed table EPTC. The looping continues in this fashion until the FSDI module determines that the solution has diverged or converged.

If convergence has been attained, the FSDI module outputs the last incremental table IEPTC and a full updated EPT table. The full table includes all of the element property information for those elements excluded from redesign, as well as all of the element property information (including final sizes) for the design elements. Path C is then followed.

Path B and Path C differ where they enter TA1-1. At this point, for path C , the ECT table is equivalenced to the original ECT table, which contains all elements; and the EPT table is equivalenced for the full, updated, EPT table, as output from FSDI. Path C then performs final stress recovery for all elements and prepares both standard NASTRAN and special FSD output.

If divergence has occured, the FSDI module outputs a full updated EPT table using the results from the previous iteration. Path D is then followed. Where Path D enters TA1 at TA1-1, the ECT table is equivalenced to the original ECT table and the EPT table is equivalenced to the previous complete and updated EPT table. When SDR2 is entered, the displacement vector matrix UGV from the previous iteration is used. Path D then follows Path C for final stress recovery.

THE SCAN MODULE, STRSCN

The purpose of this module is to scan the NASTRAN OES1 and OEF1 tables, determine pairs of algebraic maximum and minimum stresses (or forces) and their corresponding load conditions, and write the results on the STSOUT (or FECOUT) file. Table I shows the input and output files and parameters required for this module. Figure 2 shows that the module can be used in two different ways depending on whether the PIPT table is purged or not purged.

If, in using the module, the PIPT table is purged, then the CASECC table is searched for the stress output element sets defined by the SET1 through SET5 parameters. The parameter PAIRS determines the number of pairs of algebraic maximum-minimum stresses (or forces) and their corresponding load conditions. These pairs are listed starting with the most critical pair, the next most critical pair, etc. If a PIPT table is an input to STRSCN, only the most critical pair of conditions and stresses is determined for the set of design elements defined in the PIPT table. The particular stress used for the scan procedure is of necessity dependent on the NASTRAN element type.

THE SIZING MODULE, FSDI

The FSDI module performs two major functions. Initially, the module prepares the condensed ECTC and EPTC tables containing only the design variable elements. Secondly, during each resizing iteration each design element is resized based on a selected sizing criterion; and the design parameter values in the EPTC table are updated. The module then checks various convergence criteria and decides whether to enter another sizing DMAP loop or to merge the EPTC and ECTC tables back into the complete tables and end with the final stress recovery. Table II gives the input and output files and parameters for the FSDI module.

The module flow, as demonstrated in figure 3, shows that in the first pass through the module, the condensed tables ECTC and EPTC are defined and the total mass of the structural model is computed. This mass is divided into two parts: the mass of the design elements (WD), and the mass of the remaining elements (WO). A table (WDO) containing the individual masses of the design elements is also written. In the first or any subsequent pass, the design elements are then resized.

The sizing of each of the design elements is currently based on either a Fully-Stressed Design (FSD) or a Scaled-Feasible Fully-Stressed Design (SFFSD) (ref. 1), depending on the value of the input parameter SCALE. Using the results of the STRSCN module and the allowable data and size constraints, a stress ratio of the current active stress to allowable stress is determined along with the ratios of minimum size to current size and current size to maximum size. The maximum of these ratios (designated SR) is determined for each design element and the element sized by its SR ratio to determine its new size.

The SFFSD is similar to the FSD with the exception that all design elements are scaled by the largest of all the SR ratios. This ensures that at each step of redesign an acceptable design is available; that is, a design satisfying continuity and not exceeding any allowable stresses or constraings.

CONVERGENCE CRITERIA

Presently there are three ways to terminate execution (a fourth and unsatisfactory way is time):

- The permissible number of iteration loops specified by the user is reached.
- The total mass decrease between the (i-1) and i-th iteration is less than a user-defined fraction.
- The total mass between the (i-1) and i-th iteration increases.

For the latter case, the mass is currently allowed to increase through the first two (defaulted) iterations without termination.

STRESS ITERATION CRITERIA

The elements currently permitted for use with FSD include the standard NASTRAN BAR, ROD, and SHEAR elements and the Lockheed-California Company developed biaxially stiffened anistropic membrane element (BMEM). The ROD elements use the axial stress for the design stress. The SHEAR elements use the average value of shear for the design stress. Presently, the BAR elements can use only the axial stress for design. For this element type, a section-property smoothing process (to account for moment reversal in frames, for example) is being developed.

The BMEM element has available to it several options for the selection of the design stress or stresses. These options are user selected by a case control parameter. The two options which presently seem most useful are: the principal stresses at the center of the element, or a stress interaction criterion involving the two direct stresses, σx and σy, and the shear stress, τxy, all at the center of the element. This latter criterion also requires the corresponding two directional allowables; hence, the need for the allowables on the NASTRAN MAT2 Card. The BMEM element provides for the input of material property data relative to user-defined coordinate axes; such as the zero degree fiber orientation of a composite material. The element also provides for output of stresses along user-defined coordinate axes.

OUTPUT

At each iterative stage of the redesign, the scan and sizing modules print data which are useful to the designer in determining the acceptability of convergence to a reduced mass structure. Typically, the scan module outputs:

- Element identification.
- Maximum positive stress and the critical load condition identification.
- Maximum negative stress and its critical load condition.

The sizing module outputs:

- The maximum SR for each design element, along with its corresponding load condition number.
- The new size of each design element.

The scan module can also be used independently of the sizing module to scan for any desired number of critical load conditions and their corresponding critical stresses. This feature is of great use in standard stress analyses.

After the final sizing, the full EPT table is stored into the data base. From there, the table can be used by other disciplines, or used as initial input for continued strength sizing iterations or other NASTRAN analyses.

EXAMPLE

Figure 4 shows a plot of a finite element model of a composite material wing box to which the automated strength sizing procedure was applied. The truss structures on the leading edge and the trailing edge are for the purpose of transfering load from control surfaces to the wing box structures. The equations representing this transfer are automatically written by the NASTRAN-LCC rigid element multi-point constraint generator module during execution.

The structure was modeled with NASTRAN BAR elements for the fuselage frames and NASTRAN ROD elements for rib caps and fuselage longerons. The NASTRAN-LCC membrane element was used for all cover, spar, and rib panels. Since this element carries direct stress as well as shear, no rib or spar posts were used.

For the wing, unit panel thicknesses and cap areas were assigned arbitrarily for initial input properties. The fuselage frames had to be presized at this time. The rigid elements, used for external load introduction, and the fuselage frames were excluded from the design elements. Figure 5 demonstrates the total wing mass convergence after three iterations. A study of the resulting stress output for the wing showed that, for the given allowables, the resulting design was acceptable as a starting point for initial flutter evaluations. Figure 6 illustrates the final sizing distributions of the upper surface panels adjacent to the rear beam after three iterations. The tip section is sized to minimum gauge; however, this section will probably require additional stiffening to meet flutter requirements.

CONCLUDING REMARKS

A scheme that permits the gross over-all distribution of material within a structure to be determined using NASTRAN has been presented. The basic sizing algorithm is a fully-stressed design approach which offers the advantage of arbitrary initial size input; however, this method could easily be replaced by any other suitable algorithm. The basic manipulation of the ECT and EPT tables would remain the same. The sizing module concentrates on the modification of the EPT table; and, in particular, on the formulation of condensed ECT and EPT tables and on the formation of incremental stiffness matrices. This approach was selected because it offered several distinct advantages. Firstly, in most large structures, only part of the elements are to be sized. Thus, the formations of the relatively sparse incremental stiffness matrix for the design elements has advantages of operational flexibility. Secondly, the chosen approach lends itself to the formation of incremental stiffness matrices for selected sets of elements to be used as design variables in flutter and other aeroelastic design procedures. Along these same lines, by providing the basic capability to manipulate the EPT table, the material identification (MID) can be treated as a design variable; and the effect of different lay-ups on composite material structures response can be studied using an incremental approach. Finally, during the course of standard point design analyses, the FSDI module has been used to advantage to form condensed ECT and EPT tables representing sub-structure boundary coupling information.

REFERENCES

1. Berke, L.; and Khot, N. S." Use of Optimality Criteria Methods for Large Scale Systems. AGARD Lecture Series No. 70 on Structural Optimization, October 1974, pp 1-29.

TABLE I. - INPUT/OUTPUT DATA FOR STRSCN MODULE

Input Data Blocks

CASECC - Case Control Data Table

OES1 - Output Element Stress Table

OEF1 - Output Element Force Table

PIPT - Property Input Parameter Table

Output Data Blocks

STSOUT - A Table of Scanned Stresses

FCEOUT - A Table of Scanned Forces

Parameters

PAIRS - The number of maximum-minimum pairs to scan for in each element set.

$$\left.\begin{array}{l}\text{SET1}\\ \text{SET2}\\ \text{SET3}\\ \text{SET4}\\ \text{SET5}\end{array}\right\}\quad \text{Sets defining elements to be scanned}$$

TABLE II. - INPUT/OUTPUT DATA FOR FSDI MODULE

Input Data Blocks

EST - Element Summary Table

ECT - Current Element Connection Table

EPT - Current Element Property Table

EPTO - Original Element Property Table

MPT - Material Property Table

STSOUT - Table of Scanned Stresses

WDIN - Table of Current Mass of Designated Design Elements

Output Data Blocks

ECTC - Extracted Element Connection Table

EPTC - Extracted Element Property Table

IEPTC - Extracted Incremental Element Property Table

WDO - Updated Mass of Designated Design Elements

Parameters

LOOP - Number of design iterations to be executed

SCALE - Type of sizing algorithm to be used

PERCENT - A weight convergence criteria

WO - Summed mass of all nondesignated design elements

WD - Summed mass of all designated design elements

FSDP - A DMAP control parameter

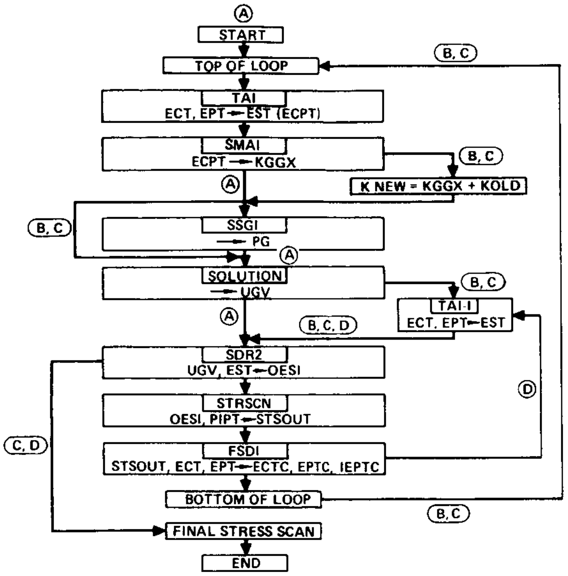

Figure 1. Flow of Sizing Procedure

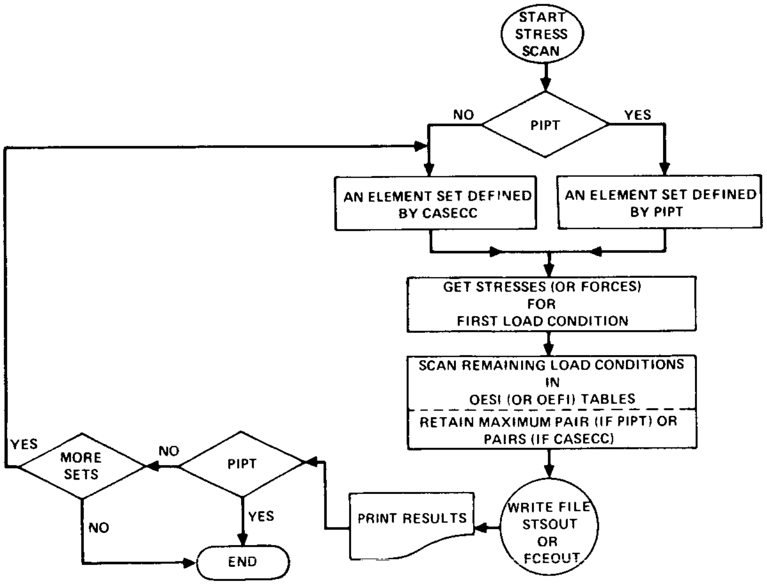

Figure 2. Stress-Scan Module Flow

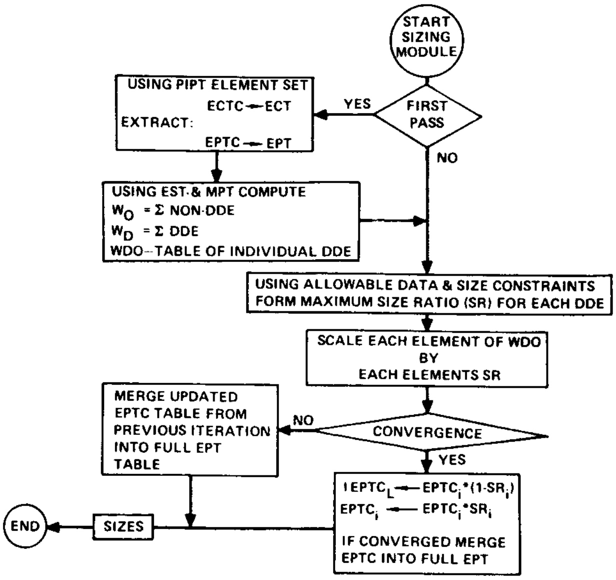

Figure 3. Sizing Module Flow

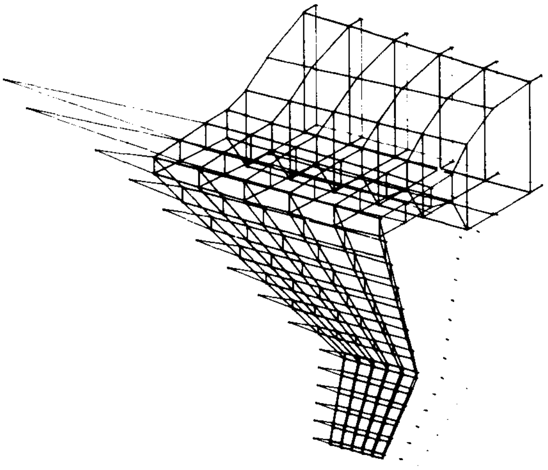

Figure 4. Wing Finite Element Model

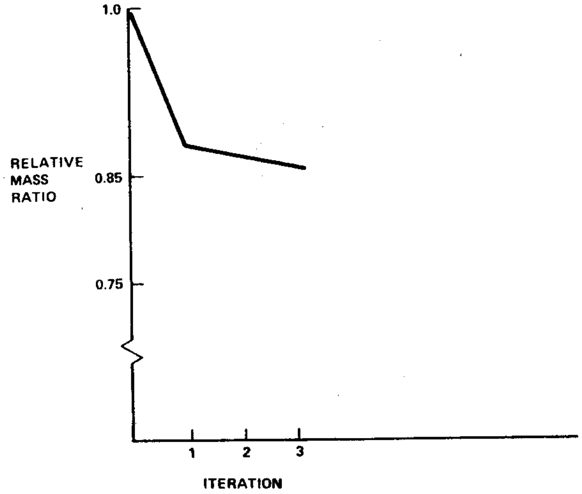

Figure 5. Total Wing Mass Convergence

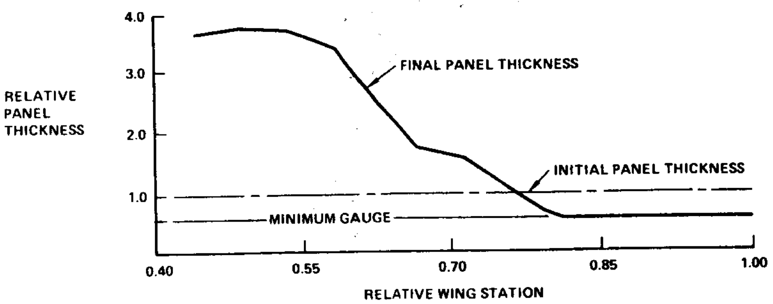

Figure 6. Sizing of Wing Upper Surface Panels Adjacent to Rear Beam