A SURVEY OF NASTRAN IMPROVEMENTS SINCE LEVEL 15.5

John R. McDonough

Computer Sciences Corporation

2

SUMMARY

Since the last public release of NASTRAN (Level 15.5), several improvements and new capabilities have been developed and installed in intermediate levels and are being analyzed and evaluated. This paper presents a survey of current improvements to the program.

INTRODUCTION

Previous discussions of some of the improvements to NASTRAN have already been presented. These are: two axisymmetric solution techniques for identical structural segments called Static Analysis Using Cyclic Symmetry for APP DISP, Rigid Format 14 and Normal Modes Analysis Using Cyclic Symmetry for APP DISP, Rigid Format 15 (Reference 1), the inclusion of a library of linear-, quadratic-, and cubic-isoparametric solid hexahedra for DISP and HEAT APProaches called CIHEX1, CIHEX2, and CIHEX3, respectively (Reference 2), and a subsonic aeroelastic capability called Modal Flutter Analysis for APP AERØ, Rigid Format 10 (Reference 3).

Improvements in operational efficiency, such as matrix handling and processing (Reference 4), are not discussed here since this paper will highlight basic features which the user will implement to gain access to a specific capability.

Other improvements (Reference 5) which have not been previously discussed are: an improved differential stiffness technique (APP DISP, Rigid Format 4), a new normal modes with differential stiffness rigid format (APP DISP, Rigid Format 13), the inclusion of two axisymmetric solids of revolution elements to the NASTRAN library (TRIAAX and TRAPAX), an identical element matrix generator (CNGRNT), an experimental property optimization technique (Fully Stressed Design), two new static analysis output features to print element strain energy (ESE) and grid point force balance (GPFØRCE), and the ability to plot complex modal deformations (CMØDAL).

STATIC ANALYSIS WITH DIFFERENTIAL STIFFNESS RIGID FORMAT

The previous differential stiffness technique (Reference 6) uses the internal element forces obtained from the linear static solution to compute the individual element differential stiffness matrices. This technique is based on the assumption that the internal load is a linear multiple of the applied load and remains fixed in magnitude and direction. Thus the effects of nonlinearity on large displacement responses were approximated by a series of applied load factors.

Another approach (Reference 7) to solve the differential stiffness problem is to iterate the displacements to compute the differential stiffness matrix K^d by

$$[K + K^d (u_i)] \{u_{i+1}\} = \{P\} \tag{1}$$

where u_i and u_{i+1} are the set of displacements at two successive iterations, K is a stiffness matrix, and P is a load vector. To avoid a decomposition of

$$[K + K^d (u_i)]$$

at each iteration, $[K^d (u_i)]$ is removed from the left hand side and is replaced with the term $[K^d (u_e)]$ to give

$$[K + K^d (u_e)] \{u_{i+1}\} = \{P\} + [K^d (u_e) - K^d (u_i)] \{u_i\} \tag{2}$$

or

$$(K + K^d (u_e)] \{u_{i+1}\} = \{P\} + [K^d (u_e - u_i)] \{u_i\} \tag{3}$$

where u_e is an estimate initially equal to the linear elastic solution. With this technique the internal loads may change due to differential stiffness effects so that the solution is not linearly related to the applied load. Thus equation (3) treats the change in differential stiffness as a load correction.

Three PARAMeters are provided to control the iterative process. The first, BETAD, limits the number of load corrections before adjusting the differential stiffness. The second, NT, limits the cummulative number of iterations. Thus load correction iterations can be performed up to the limit BETAD, at which time the differential stiffness is adjusted, and then more load correction iterations are performed and an adjustment is made to a new differential stiffness until NT is exhausted. Smaller values of BETAD allow more frequent adjustments to the differential stiffness. It is more economical to iterate via a larger value for BETA so that fewer DMAP modules in the rigid format are re-executed. The third, ESPI∅, is a convergence criteria which terminates the process when successive iterations of the differential stiffness are sufficiently small. Convergence occurs when ε_i < EPSI∅ where

$$\varepsilon_i = \frac{|\{u_{i+1}\}^T \{P_{i+1} - P_i\}|}{|\{u_{i+1}\}^T \{P_i\}|} \tag{4}$$

A differential stiffness check functional module, DSCHK, performs convergence and timing tests and issues appropriate information each time it is executed. The differential stiffness coefficient functional module, DSMG2, is not used with this technique. The iterated differential stiffness solution does not match exact theoretical results, but shows a significant improvement over the one-step method, particularly where nonlinearity is pronounced.

NORMAL MODES WITH DIFFERENTIAL STIFFNESS RIGID FORMAT

It was possible to obtain the normal modes of a structure including differential stiffness effects by ALTERing the buckling analysis rigid format using NASTRAN Level 12.0 (Reference 8) or Level 15.1 (Reference 9). The obvious difference between the two was that the earlier version required two subcases and the later version required three. The three-subcase version is the one that is incorporated in a new, separate rigid format (SØL 13; APP DISP). Although the static analysis with differential stiffness rigid format solution technique has been substantially changed, as discussed above, the same technique has not been incorporated in this rigid format.

This rigid format requires three subcases to define the three steps of normal modes analysis with differential stiffness effects. The first subcase pertains to the linear solution step in which static loads are defined and output requests are specified. The second subcase is used to prescribe one differential stiffness loading scale factor and to prescribe output for second order effects. The third subcase contains the eigenvalue extraction method and output requests. Overall definitions can be placed above the subcase level and plot requests follow the third subcase in the usual convention. Since the previous differential stiffness technique allowed several loading factors to be prescribed (Reference 10), this approach can only be used with one load factor at a time. If the load factor technique is still preferred for differential stiffness analysis, this rigid format can be used with minor ALTERS to loop back for new loads and JUMP around the normal modes analysis.

This combination rigid format supports requests for static undeformed and deformed structural plots as well as mode shapes.

THE TRIAAX AND TRAPAX ELEMENTS

One of the types of elements included in the NASTRAN library has been the solid of revolution (Reference 11). Two specific cross sections available have been the triangle and the trapezoid which were defined by the CTRIARG and CTRAPRG data cards, respectively. These elements are, however, restricted to axisymmetric loading conditions. Two new axisymmetric ring elements have been developed for the asymmetric loading case. These are the TRIAAX and TRAPAX. Like the earlier elements they may not be used with other types of structural elements and they may be used alone or with each other. Because of the inherrantly different loading capabilities, the -RG type elements cannot be used with the -AX type elements.

In a manner similar to the development of the conical shell element, the radial, tangential and axial displacements of a point $p(r,z,\phi)$, expressed in terms of Fourier series, are

$$u(r,z,\phi) = u_0(r,z) + \sum_{n=1}^{m} u_n(r,z) \cos n\phi + \sum_{n=1}^{m} u_n^*(r,z) \sin n\phi \qquad (5)$$

$$v(r,z,\phi) = v_0^*(r,z) + \sum_{n=1}^{m} v_n(r,z) \sin n\phi - \sum_{n=1}^{m} v_n^*(r,z) \cos n\phi \qquad (6)$$

$$w(r,z,\phi) = w_0(r,z) + \sum_{n=1}^{m} w_n(r,z) \cos n\phi + \sum_{n=1}^{m} w_n^*(r,z) \sin n\phi \qquad (7)$$

respectively.

The generalized displacement amplitudes are obtained for each n harmonic for the TRIAAX and TRAPAX elements at specified azimuth positions, ϕ. When n = 0, the degenerate displacement coefficients are obtained for the TRIARG and TRAPRG elements. Thus, with the use of new property cards, PTRIAAX and PTRAPAX, up to 14 azimuth coordinates can be specified for displacement and stress recovery. Displacements and forces are evaluated at the three (or four) corners. Stresses are evaluated for the triangular element on a circle generated by the centroid while stresses for the trapezoid are computed at the four corners as well as the centroid.

THE CNGRNT FEATURE

When a structural model is made up of elements which are truly identical, i.e., orientation, geometry, etc., an identical matrix generator feature is utilized with a CNGRNT bulk data card. One element is designated as the primary element, i.e., the one for which the stiffness, mass and damping matrices will be calculated and all other identical elements, declared as secondary, have identical matrix elements. This feature will then reduce the amount of time required to generate numerous matrices which are exactly identical. In a limited number of cases, the NASTRAN data generator module, INPUT, can be used in lieu of actual data cards to model bars, plates and scalar elements (Reference 12). The CNGRNT feature is automatically used by this module so it is therefore unnecessary to employ CNGRNT cards with it.

A structure can be modeled in one of three ways: (1) a complete data deck, (2) no connection cards but using the INPUT module or (3) a complete data deck with the CNGRNT feature. Using one of the NASTRAN demonstration problems (static analysis of a simply supported 5 x 50-inch plate subjected to a varying load across its transverse midplane), comparative computer CPU times were compiled to show the improvement using this technique. This problem contains 250 CQUAD1 plate elements, an initial stiffness matrix, $[K_{gg}]$, of order 1836 and a

constrained stiffness matrix, $[K_{\ell\ell}]$, of order 760 (Reference 13). With the complete data deck it took the CDC 6400 a total of 425 CPU seconds to build the individual stiffness matrices, whereas the same computer took about 2.5 CPU seconds to complete one stiffness matrix and duplicate the rest for the same problem set up the other two ways. Compared to Level 15.5, it took the CDC 6400 a total of 730 CPU seconds to build the individual stiffness matrices and about 400 CPU seconds with the original INPUT module. The differences achieved between Level 15.5 and the improved level cannot be attributed to the congruent feature but to an improved matrix generator (Reference 14). Thus these time studies are only valid when modeling techniques are compared within the same level. Table 1 shows the detailed results.

FULLY STRESSED DESIGN

The concept of a "fully stressed" design of a structure is to adjust some parameter so that each member is at a zero margin of safety since a pre-determined stress limit has been achieved. The parameters to be adjusted are cross-sectional area, thickness or moment of inertia. Properties may be scaled for common elements, such as the thickness for all plates, or they may be scaled for individual elements, each having its own design criteria.

The iteration process begins by performing a static analysis (APP DISP; SØL 1) for all loading conditions using the initial values for all element properties, p. A new property, p', will be scaled from

$$p' = p \left[\frac{\alpha}{\alpha + (1-\alpha)\gamma} \right] \tag{8}$$

where p is the current property value, γ is an iteration factor set by the user and α is defined as

$$\alpha = \text{Max} \left| \frac{\sigma}{\sigma_\ell} \right| \tag{9}$$

where σ is a stress and σ_ℓ is the stress limit. The maximum value of α is taken for all loading conditions.

The value of the iteration factor, γ, is by default equal to unity. In this case Equation (8) is

$$p' = p\alpha \tag{10}$$

When $\gamma = 0$, Equation (8) is

$$p' = p \tag{11}$$

Thus the user can regulate the iteration process by choosing an intermediate value for γ.

The maximum change in any property is specified by the user to be

$$KMIN < \frac{p'}{p} < KMAX \qquad (12)$$

The change ratio can be ignored by setting KMAX equal to zero, otherwise KMAX > KMIN.

The iteration process continues until a convergence criteria

$$\epsilon > \frac{|\sigma - \sigma_\ell|}{\sigma_\ell} \qquad (13)$$

is achieved or until the number of iterations is reached, both of which are set by the user.

Optimization can be performed on bars, rods and tubes as well as shear panels and plate and membrane elements. The cross-sectional area or torsional constant of the bars, rods and tubes can be optimized. Additionally the moments of inertia of the bar can be optimized. The moments of inertia or thickness are optimized for the other elements, depending upon whichever is appropriate. The stress limit used is that in tension, compression, shear or torsion, as dictated by the type of element in question.

A PØPT bulk data card contains the number of iterations desided, the values for ϵ and γ, and PRINT and PUNCH options for new property bulk data cards. A PLIMIT bulk data card contains the values for KMIN and KMAX with reference to the element type involved.

Functional Module ØPTPR1, in Link 2, processes the PØPT and PLIMIT bulk data cards and sets up appropriate tables. Functional Module ØPTPR2, in Link 8, performs the new property calculations. These two modules re-execute until insufficient time or user criteria is met.

ELEMENT STRAIN ENERGY AND GRID POINT FORCE BALANCE

A Functional Module, GPFDR, in Link 13 of DISP Rigid Format 1, processes case control requests for grid point force balance and element strain energy output.

The card GPFØRCE $\left[\left(\frac{PRINT}{PUNCH} \right) \right] = \begin{Bmatrix} ALL \\ n \\ NØNE \end{Bmatrix}$ requests a force balance output for

specified grid points. The output is arranged by grid or scalar point number

and lists the forces at the point due to applied loads $\{P_g\}$, forces of single point constraint $\{q_g\}$ and element forces $\{F_e\}$. These forces are summed and the total is ideally identical to zero for true balance; however, in some cases round-off errors cause extremely small force residuals when combining very large and very small quantities.

Element strain energy is output via the card ESE $\left[\left(\dfrac{PRINT}{PUNCH}\right)\right] = \left\{\begin{array}{c} ALL \\ n \\ N\emptyset NE \end{array}\right\}$

in the case control deck. From the equation for element forces,

$$\{F_E\} = -[K_E]\{u_E\} \tag{14}$$

where $[K_E]$ is an element stiffness matrix and $\{u_E\}$ is the displacement vector, the element strain energy is computed from

$$U = 1/2 \mid \{F_E\}^T \{u_E\} \mid \tag{15}$$

The total strain energy for all structural elements is printed followed by a listing of the strain energies of the individual elements, arranged by element type.

COMPLEX MODAL DISPLACEMENT PLOTS

The complex eigenvalue extraction rigid format (APP DISP, S∅L 10) does not contain a deformed plot option, even for real mode shapes (Reference 15). The user would instead employ APP DISP, S∅L 3 to compute real eigenvalues and plot only the real mode shapes. However, included with the subsonic aeroelastic flutter analysis (Reference 3) is the capability to plot complex mode shapes. To do this, the user defines a plot SET of interest in the case control deck, such as a wing surface (Reference 16), which may be modeled with structural or aerodynamic elements. Assuming the preliminary definitions and specifications following ∅UTPUT(PL∅T) are complete, the plot command card PL∅T M∅DAL DEF∅RMA-TI∅N ....etc.... will generate the real part of the mode shapes. The plot command card PL∅T CM∅DAL DEF∅RMATION ....etc.... will generate the imaginary part of the mode shapes. An eigenvalue frequency RANGE of interest may be included in both commands; otherwise all the frequencies at which solutions were obtained will be used for the plots. A MAGNITUDE or PHASE LAG may be specified on the complex plot command card; otherwise a default of zero degrees phase will be used.

DEMONSTRATION PROBLEMS

Levy has suggested a scheme to demonstrate the capabilities offered by NASTRAN with a comprehensive set of demonstration problems (Reference 17). Although total adoption of such a premise would necessitate revisions to many existing demonstration problems (and an extensive writing effort for the NASTRAN Demonstration Problem Manual), new demonstration problems have been devised and are still being devised to adequately exercise new capabilities. Also, a more compact table of options and features illustrated by each demonstration problem is in progress. Where one model is analyzed via more than one technique, such as user data cards versus INPUT data generation, the "duplicate" problem is simply referred to as another version of the first. Thus a proliferation of problem identification does not result and the user need not be familiar with a scattered set of examples in order to study aspects of the program in which he is interested.

The static analysis rigid format (APP DISP, SØL 1) contains the most demonstration problems and has had the most newly devised demonstration problems added to it. A thick walled cylinder, modeled with the TRAPAX and TRIAAX elements and subjected to a pressure load on a portion of its surface, is used to illustrate the use of these elements. A beam, modeled with general (GENEL) elements, described in terms of the flexibility matrix (Reference 18) is used to illustrate that feature introduced in Level 15.5. A reinforced arch, modeled with quadrilateral plates and reinforcing rods, is used to illustrate the property optimization technique in adjusting the plate thicknesses and rod cross sections. The isoparametric solid hexahedra are seperately used to model the same section of a circular cylinder to demonstrate the utility of these elements. The biconvex wing demonstration problem (Reference 19) and the free rectangular plate with thermal loading (Reference 20) are used to separately demonstrate the comparable use of the CQDMEM1 and the CQDMEM2 elements (Reference 21) which were introduced in Level 15.5. The congruent feature is demonstrated as previously discussed. Steady state heat conduction through a washer (Reference 22) is converted to APP HEAT, SØL 1, with minor data changes to illustrate the updated methodology required by the completion of the heat transfer capability for Level 15.5. In addition, the convection and radiation heat transfer additions (Reference 23) lead to the inclusion of appropriate demonstration problems for APP HEAT, SØL 3 and APP HEAT, SØL 9. A beam-column, subjected to a compressive axial load, is used to illustrate the APP DISP, SØL 13 normal modes with differential stiffness effects rigid format. The example presented in Reference 1 is used to show the static cyclic symmetry technique for APP DISP, SØL 14 and a similar version of it demonstrates the normal modes analysis using cyclic symmetry for APP DISP, SØL 15. The example studied in Reference 3 is the basic problem used to model two aeroelastic demonstration problems for APP AERØ, SØL 10 to show the computational and graphic capabilities.

CONCLUDING REMARKS

These are some of the user features, but certainly not all, which are currently in stages of development, testing and evaluation. The choice of an

appropriate demonstration problem illustrates the utility of a particular feature to verify theoretical cases. The real testing rests with the user.

REFERENCES

1. MacNeal, R. H., R. L. Harder and J. B. Mason, NASTRAN Cyclic Symmetry Capability, NASTRAN User's Experiences, NASA TM X-2893, September 1973, pp 395-421.

2. Johnson, Stephen E. and Eric I. Field, Three Isoparametric Solid Elements for NASTRAN, NASTRAN Users' Experiences, NASA TM X-2893, September 1973, pp 423-437.

3. Doggett, Robert V. and Robert L. Harder, Subsonic Flutter Analysis Addition to NASTRAN, NASTRAN Users' Experiences, NASA TM X-2893, September 1973, pp 507-529.

4. McCormick, Caleb W., Review of NASTRAN Development Relative to Efficiency of Execution, NASTRAN Users' Experiences, NASA TM X-2893, September 1973, pp 7-28.

5. NASTRAN NEWSLETTER, September 5, 1973, pp 3-5.

6. The NASTRAN Theoretical Manual, NASA SP-221(01), December 1972, Section 7.1.

7. Zienkiewicz, O. C. and G. C. Nayak, A General Approach to Problems of Plasticity and Large Deformation Using Isoparametric Elements, Proceedings Third Conference on Matrix Methods in Structural Mechanics, AFFDL-TR-71-160, U.S. Air Force, December 1973, pp 881-928.

8. NASTRAN NEWSLETTER, March 13, 1972, pp 7-9.

9. NASTRAN NEWSLETTER, November 7, 1972, pp 22-28.

10. The NASTRAN User's Manual, NASA SP-222(01), May 1973, Section 3.5.

11. The NASTRAN Theoretical Manual, NASA SP-221(01), December 1972, Section 5.11.

12. The NASTRAN User's Manual, NASA SP-222(01), May 1973, Section 2.6.

13. The NASTRAN Demonstration Problem Manual, NASA SP-224(01), June 1972, Section 1.4.

14. The NASTRAN Programmer's Manual, NASA SP-223(01), May 1973, Section 4.124.

15. The NASTRAN User's Manual, NASA SP-222(01), May 1973, Sections 3.11, 4.1 and 4.2.

16. McDonough, John R., NASTRAN Plotting Capabilities, NASTRAN Users' Experiences, NASA TM X-2378, September 1971, pp 709-735.

17. Levy, Alvin, Design of NASTRAN Demonstration Problems, NASTRAN Users' Experiences, TM X-2637, September 1972, pp 497-516.

18. The NASTRAN Theoretical Manual, NASA SP-221(01), December 1972, Section 5.7.

19. The NASTRAN Demonstration Problem Manual, NASA SP-224(01), June 1972, Section 1.1.

20. The NASTRAN Demonstration Problem Manual, NASA SP-224(01), June 1972, Section 1.3.

21. The NASTRAN Theoretical Manual, NASA SP-221(01), December 1972, Section 5.8.

22. The NASTRAN Demonstration Problem Manual, NASA SP-224(01), June 1972, Section 1.12.

23. The NASTRAN Theoretical Manual, NASA SP-221(01), December 1972, Section 8.

TABLE 1 - SAMPLE PROBLEM CUMULATIVE TIMES ON THE CDC 6400

Step	Improved Level (CPU seconds)				Level 15.5 (CPU seconds)	
	Complete Data Deck	Partial Deck with INPUT	Complete Deck with CNGRNT		Complete Data Deck	Partial Deck with INPUT
ΔA	41.198	27.310	43.770		37.228	23.638
ΔB	425.852	2.496	2.788		398.850	398.600
ΔC	332.886	333.792	337.314		296.232	302.070
Total JØB Time	799.936	363.598	383.872		732.310	724.308

ΔA = Total time for initial processing up to beginning of stiffness matrix module

ΔB = Total time in stiffness matrix module

ΔC = Total time from end of stiffness matrix module to EXIT