NASA SP-260

NASTRAN

A SUMMARY OF THE FUNCTIONS AND CAPABILITIES OF THE NASA STRUCTURAL ANALYSIS COMPUTER SYSTEM

Butler • Michel

NATIONAL AERONAUTICS AND SPACE ADMINISTRATION

NASA SP-260

NASTRAN

A SUMMARY OF THE FUNCTIONS AND CAPABILITIES OF THE NASA STRUCTURAL ANALYSIS COMPUTER SYSTEM

by

Thomas G. Butler and Douglas Michel

Scientific and Technical Information Office 1971
NATIONAL AERONAUTICS AND SPACE ADMINISTRATION
Washington, D.C.

For sale by the Superintendent of Documents, U.S. Government Printing Office
Washington, D.C. 20402 - Price 35 cents

FOREWORD

The NASTRAN computer system was developed for NASA by a team comprised of Computer Sciences Corporation, the MacNeal-Schwendler Corporation, and the Martin Company under contract to the NASTRAN Project Office at Goddard Space Flight Center. After a period of extensive checkout and correction of errors, NASA believes that the current version of NASTRAN designated "level 12.0" is suitable for general release. A system the size of NASTRAN can never be completely debugged; however, with proper maintenance and cooperation from the user community, inconveniences can be minimized. It is NASA's intention to provide continuing support for maintaining and improving the NASTRAN system.

With the completion of level 12.0 and first public release of NASTRAN, responsibility for management of the system has been placed with the NASTRAN System Management Office (NSMO) at Langley Research Center. Future inquiries regarding NASTRAN should be directed to:

NASTRAN Systems Management Office (NSMO)
Mail Stop 188C
Structures Division
NASA Langley Research Center
Hampton, Virginia 23365
Telephones: (703) 827–2388 or (703) 827–2414

It is planned that NASA will provide the NASTRAN user community with significant support through the operation of an in-house management office and through contractor maintenance of a standard level of the system. Users of NASTRAN should report to NSMO any errors they find in a standard version of the system and, also, suggestions for improvements and additions of new capability. With user cooperation and NASA support, NASTRAN can become a widely used industry tool for structural analysis.

NASTRAN is being distributed through the COSMIC facility at cost. The address and telephone number are

Computer Software Management and Information Center (COSMIC)
Barrow Hall
University of Georgia
Athens, Georgia 30601
Telephone: (404) 542–3265

MELVIN G. ROSCHE
Office of Advanced Research and Technology
National Aeronautics and Space Administration

PREFACE

This publication summarizes the NASTRAN program for prospective users. NASTRAN is described as a general-purpose digital computer program designed to analyze the behavior of elastic structures under a range of loading conditions using a finite-element displacement method approach. The current capability of the program is described. It includes a broad capability in solving linear static, dynamic, and eigenvalue problems, and a limited capability for solving some nonlinear problems.

A detailed description is given of the current library of elastic elements in NASTRAN. Included are bars, rods, tubes, membranes, shear and twist panels, triangular and quadrilateral plates, conical and toroidal shells, solids of revolution, scalar elements, general elements, and constraint elements. The NASTRAN geometrical systems and material properties are also discussed.

Although most of the report is directed to the user, there is a section for the user and the programmer regarding computer system implementation. The computers on which NASTRAN is currently designed to operate are the IBM 7094/7040 Direct Coupled System operating on IBSYS/DCOS; the Univac 1108 operating on EXEC 8; the IBM 360 models 50 and above; and the CDC 6400, 6500, and 6600 operating on scope 3.2. A description of plotting capability and plotting equipment for which NASTRAN has currently been designed to operate is also given.

A description of the NASTRAN documentation consisting of a theoretical manual, a user's manual, a programmer's manual, and a demonstration problem manual is also presented.

T.G.B
D.M.

CONTENTS

1. INTRODUCTION

The purpose of this report is to provide the reader with a brief description of the NASTRAN Computer Program in sufficient depth for him to decide whether or not to avail himself of it. For additional information, he should consult the manuals listed in section 10.

NASTRAN is a general-purpose digital computer program designed to analyze the behavior of elastic structures under a range of loading conditions using a finite-element displacement method approach. The program is applicable to most linear and some nonlinear structures that can be represented by combinations of finite elements contained in the NASTRAN library of elements described in section 5. Section 5 also discusses the material properties for elements, and section 4 discusses geometry such as the coordinate systems and grid points.

A wide range of analytical capability has been built into NASTRAN, including (1) static response to concentrated and distributed loads, thermal expansion, and enforced deformation; (2) dynamic response to transient loads, steady-state harmonic loads, and random excitation; and (3) determination of real and complex eigenvalues for use in vibration analysis, dynamic stability analysis, and elastic stability analysis. Details of these and other capabilities are discussed in section 2 and expanded upon in sections 6, 7, and 8 which cover the subjects of static analysis, eigenvalue problems, and dynamic analysis, respectively.

NASTRAN is highly user-oriented. The user may present data to the computer with ease, since the program is systematically organized to do much of the work automatically and minimize the burden on the analyst. Error messages are written in the vocabulary of the structural analyst rather than in the vocabulary of the programmer. Printed output is liberally annotated and presented in separate, easily distinguishable units. NASTRAN has the capability of restarting only between modules (see section 3). This means that if there is an interruption during execution of a subroutine within a module, a restart will occur at the beginning of the module; but it will not recover the completed computations of the interrupted subroutine nor its predecessor calculations within that module.

NASTRAN is aimed primarily at the large problem involving many degrees-of-freedom. Generally, economic considerations will limit the problem size long before reaching the limitations of the NASTRAN algorithms; therefore, the system has the capability of treating problems of virtually unlimited size. Computational procedures have been selected to provide efficiency and accuracy for large problems. Also, to have an accuracy consistent with large matrix operations, double precision is currently carried throughout all calculations.

To the programmer NASTRAN is more akin to a computer operating system than to an applications program. Because the NASTRAN system requires a significant effort on the part of the computer analyst to make it operational on any particular computer, section 3, on computer system implementation, has been included. Other subjects discussed in this section are the various computers on which NASTRAN is currently designed to operate and the modular design of the program.

NASTRAN has a broad plotting capability, both in structure plotting and curve plotting (described in section 9). The plotting equipment of various manufacturers—including table, microfilm, and incremental plotters—on which NASTRAN is currently designed to operate is also listed in this section.

The last section, section 11, discusses the user experience of the program as it is currently viewed. A program the size of NASTRAN can never be completely debugged. However, much of the program has seen use to varying degrees, and it is the purpose of this discussion to convey some measure of reliability regarding the use of various parts of the program to the potential user.

Modifications, corrections, and improvements to the NASTRAN system are worked on in stages.

When a stage is ready, a major revision is made to the system and it is called a level: the current NASTRAN level is 12.0. Changes between levels, called patches, are accumulated in increments and are designated after the decimal, e.g., 12.1.

2. SUMMARY OF PROGRAM CAPABILITY

A wide range of analysis capability has been built into NASTRAN as indicated in table 1.

TABLE 1.—*Summary of Program Capability*

RIGID FORMATS

- Static analyses
  - Basic static analysis
  - Static analysis with inertia relief
  - Static analysis with differential stiffness
  - Piecewise linear analysis of nonlinear static response
- Elastic stability analysis
  - Buckling
- Dynamic analyses
  - Normal mode analysis
  - Transient response, modal method
  - Transient response, direct integration method
  - Frequency and random response, modal method
  - Frequency and random response, direct integration method
  - Complex eigenvalue analysis, modal method
  - Complex eigenvalue analysis, direct integration method

DIRECT MATRIX ABSTRACTION PROGRAM (DMAP)

- Input
- Matrix operations
- Restart provisions
- Output

It contains programmed sequences of ordered matrix operations termed "rigid formats" for solving the cases indicated in Table 1 and described below:

2.1 Rigid Formats

STATIC ANALYSES

Basic static analysis solves for the response of complex structures to static loads. It yields grid point displacements, constraint forces, element forces, and stresses, as well as yielding weight and balance information and generating structural plots of load deformations.

Static analysis with inertia relief solves for responses to static loads and to inertia loads resulting from steady accelerations. It generates plots of all static load deformations and also yields weight and balance information.

Static analysis with differential stiffness solves for the response to a single loading condition and then determines the differential stiffness effect caused by large nonlinear motion. The simultaneous application of load and differential stiffness effects are applied in increments. This format yields the usual static analysis results for a single loading and then yields displacement, force, and stress information for each increment of the differential stiffness factor. It also yields weight and balance information and generates plots of combined linear and differential stiffness responses.

Piecewise linear analysis of nonlinear static response solves for the response of complex nonlinear elastic or plastic structures to any one static loading. The stiffness matrix is modified incrementally as the loads reach piecewise linear threshold values. It yields accumulated displacements, forces, and stresses at the end of each increment. This analysis also gives weight and balance information and generates plots of deformations accumulated after each linear increment.

ELASTIC STABILITY ANALYSIS

Buckling analysis performs a differential stiffness analysis of a complex structure and then performs an eigenvalue analysis on the pair of matrices consisting of the linear stiffness matrix and the differential stiffness matrix, to determine the value of the load which would cause buckling and the resulting deformation mode. It is equipped to handle only first-order buckling and not post-buckling displacements. It yields displacement, force, and stress information at the threshold load for buckling and also gives weight and balance information and generates plots of buckling modes.

DYNAMIC ANALYSES

Normal mode analysis solves for frequencies and shapes of the natural modes of complex structures. It yields normalized modal, grid-point displacements and constraint forces and normalized modal element forces and stresses. It also gives weight and balance information and generates plots of normalized mode shapes.

Transient response, modal method solves for the response of complex structures to time varying loads in which damping can be either viscous or structural. A real eigenvalue analysis operates on structural mass and stiffness matrices to define generalized modal coordinates. The differential equations are reformulated in terms of modal coordinates in an uncoupled set. Integration is performed by finite differences on the uncoupled modal differential equations. This format yields the time-varying displacements, velocities, accelerations, and constraint forces at grid points and the time-varying forces and stresses in elements. It also yields weight and balance information and generates curve plots of any output quantity against either time or frequency and generates plots of structural deformation at specified instants of time.

Transient response, direct integration method solves for the response of complex structures to time-varying loads in which damping can be either viscous or structural. Integration is performed by finite differences on the coupled differential equations as formulated directly in terms of grid point degrees-of-freedom. It yields the same information described under "transient response, modal method" above.

Frequency and random response, modal method solves two problems in frequency space, performing real eigenvalue analysis on the matrices with grid-point degrees-of-freedom to set up smaller matrices in modal coordinates before doing the following two analyses:

(a) It solves the response of a complex structure, having either viscous and/or structural damping, to a spectrum of steady sinusoidal forcing. It yields the real and imaginary parts of displacements, velocities, accelerations, and constraint forces at grid points and the real and imaginary parts of forces and stresses in elements. These output quantities can be normalized to unity forcing amplitude to form transfer functions. It plots output quantities against frequency.

(b) It solves the response of a complex structure, characterized by the above transfer functions, to stationary random forcing applied in the form of a cross spectral density. It yields the auto-spectral density in that quantity used in the transfer functions, i.e., displacement, velocity, acceleration, constraint force, element force, or stress. It also transforms auto-spectra to auto-correlation functions and plots density functions against frequency and correlation functions against time.

This format also gives weight and balance information and generates curve plots (not structural plots).

Frequency and random response, direct integration method solves two problems in frequency space, but operates on the matrices as formulated directly in terms of grid point degrees-of-freedom. It performs the same two analyses described above under "frequency and random response, modal method" and provides the same information.

Complex eigenvalue analysis, modal method solves for frequencies and eigenvectors of the complex vibration modes of complex structures in which the damping can be either viscous or structural. A real eigenvalue analysis operates on matrices with grid point degrees-of-freedom to set up smaller matrices in modal coordinates before extracting complex eigenvalues. It yields normalized complex eigenvectors of grid point displacements and grid point constraint forces, plus complex element forces and stresses, but does not generate complex modal plots. Weight and balance information are also given.

Complex eigenvalue analysis, direct integration method solves for frequencies and eigenvectors of the complex vibration modes of complex structures in which the damping can be either viscous or structural. The complex eigenvalue extraction module operates on the matrices as formulated directly from the grid point degrees-of-freedom. It yields the same information as described above under "complex eigenvalue analysis, modal method."

2.2 Direct Matrix Abstraction Program (DMAP)

If an analyst wants to do a case not provided for in the rigid formats, he can organize his own problem steps by using a language, called DMAP, contained within NASTRAN. The executive control, the input file processor, and the output file processor are made to operate automatically with the analyst's DMAP program, so that the user need not concern himself with core assignments and secondary storage assignments. The user need only specify the sequence of matrix operations and the module selections needed to solve the mathematical formulation of his particular case. He arranges for the forms of his input and output and makes provision for problem recovery and restart. In fact, DMAP need not necessarily pertain to structures; it can handle problems in any discipline so long as it is formulated in matrices. The other principle option in using DMAP is in the instance of a user being essentially satisfied with a rigid format but wanting to modify or augment its operation. It is possible to make a DMAP alteration to a given rigid format.

3. COMPUTER SYSTEM IMPLEMENTATION

The computers and operating systems on which NASTRAN is designed to operate are those used by the NASA Centers, which probably includes over half of the large computers and systems used by the aerospace industry. Although the IBM 7094/7040 Direct Coupled System was included in this list at the outset, it has been replaced by more modern equipment. As a result, NASTRAN has been brought up to only level 8.0 for this system, whereas the others have been brought up to level 12.0. Also, because of NASTRAN's size, it currently operates only on those computers equal to or larger than the model 50 of the IBM 360 series, the UNIVAC 1108, or the CDC 6400. A summary of computers, operating systems, and other requirements necessary for operating NASTRAN is given in table 2.

The ability to operate on many computers was accomplished by dividing the structure of the program into two distinct parts: the executive system and the solution modules. The executive system, of necessity, heavily interfaces with the computer operating system, and consequently, is machine- and operating system-dependent. The solution modules communicate only with the NASTRAN executive system and are therefore machine-independent. The executive amounts to less than 10 percent of the total code, so, for the most part, NASTRAN is machine-independent. The bulk of the code consists of the modules, i.e., functional modules and mathematical modules, which are written in a common subset of FORTRAN. Thus their coding language and their communication path make the solution modules machine-independent.

The restriction that all solution modules communicate only with the executive provides not only a measure of machine and operating system independence but also provides the modularity for updating. No module communicates with any other module directly; communication between modules is only through the executive. Therefore, a change in any one module affects no other module as long as the changed module preserves the interface with the executive. Each module is constructed in terms of data blocks, subroutines, parameters, and drivers. Subroutines within a module can communicate with one another. Data blocks and parameters are assigned to the module when the executive delivers it into core. The drivers are the communication stations at the various levels in the overlay structure of the module in core. These are established in the initial building of the program from the object decks.

4. GEOMETRY

The organization of the structure to be analyzed is established by the selection of a reference system. The reference system can be chosen as a geometric coordinate frame or a collection of scalar connections.

4.1 Coordinate Systems

A rectangular Cartesian coordinate system is the basic system used to define position and is the primary reference. The basic Cartesian coordinates can be transformed to the following coordinate systems:

(1) Basic cylindrical
(2) Basic spherical
(3) Local Cartesian
(4) Local cylindrical
(5) Local spherical

TABLE 2.—*Current Computer Configurations*

Computer	Operating system	Minimum main core assignment	Peripheral storage*
UNIVAC 1108	EXEC 8	64K words	At least one FASTRAND drum
IBM 360 model 50	OS 360/PCP	235K bytes of a 256K-byte core	At least one 2314 disc
IBM 360 models 50, 65, 67	OS 360/PCP or MFT HASP permitted	300K bytes of a 512K-byte core	At least one 2314 disc
IBM 360 models 50, 65, 67, 75, 85, 91, 95	OS 360/PCP or MFT or MVT	400K bytes of a 1000K-byte core	At least one 2314 disc
CDC 6400, 6500, 6600	SCOPE 3.2 RUN compiler with nonstandard returns	130K octal	At least one 6603 or 6638 disc

* For maximum operating capability, up to five tape drives can be called for restarting and plotting.

Points and vectors can be described in the basic system or in additional coordinate systems that may be defined with respect to the basic system. Local coordinate systems can also be defined as a matter of convenience either in terms of the basic system or in terms of a previously defined supplementary system. There are three types of systems currently available: rectangular Cartesian, cylindrical, and spherical. Output can be called for in a system different from that which was used to define the input. All plotting is done in the primary reference system.

4.2 Grid, Scalar, and Extra Points

Each geometric *grid point* is supplied with six degrees-of-freedom unless constrained to less. (See table 3.) Constraints can be supplied at the time of point definition without variation, or as part of constraint sets to test for variations in behavior, or as combinations of both. Single point constraints impose fixed values to grid point degrees-of-freedom, while multi-point constraints set up relations between grid points that allow all such involved freedoms to vary. However, their variations maintain certain relative conditions, such as points constrained to lie on a surface or to remain in line with each other. After a structure is modeled completely with mass and stiffness properties assembled, equations at grid points can be omitted from the solution set by a Guyan reduction. Sliding joints at grid points can be represented by grid point options called pin flags.

TABLE 3.—*Grid, Scalar, and Extra Points*

GRID POINTS—Six degrees-of-freedom constrained by:
Single point constraints
Multipoint constraints
Omitted equations
Sliding joint
SCALAR POINTS
EXTRA POINTS

Scalar points have only one degree-of-freedom that can have vector connotation or not, as problem requirements demand. Scalar systems can be joined with grid point systems, e.g., when a scalar hydraulic system causes a vector force to appear in a structure, or the scalar model can be used to solve for a scalar magnitude without regard to orientation, such as voltage, hydraulic pressure, or speed.

Extra points are non-geometric points used to represent generalized coordinates such as modal coordinates. These can also join with grid point systems.

5. ELASTIC ELEMENTS AND MATERIAL PROPERTIES

The structure to be analyzed is represented as an array of elastic finite elements each of which is provided with an appropriate set of sectional properties and material properties such as elastic moduli, thermal moduli, and mass properties.

5.1 Elastic Elements

The NASTRAN element library currently contains one-, two-, and three-dimensional elements, as shown in table 4.

The *bar* is the most general one-dimensional element and deforms in bending, torsion, and extension but not in shear. It is prismatic (no taper) and has 6 degrees-of-freedom at each end, but the transverse shear loads produce only couples resulting in longitudinal stresses. The connections to grid points may be offset by differing amounts at either end.

The *rod* is a special simplified case of the bar and deforms in extension and torsion. It is introduced for the convenience of not having to constrain a bar to behave like a rod. Trusses can be quickly modeled with the rod element.

The *tube* is another special element introduced for convenience and is similar to a rod element except for the additional provision of wall thickness.

All two-dimensional elements appear both as triangles and as quadrilaterals except the shear and twist panels which are quadrilaterals only.

The *shear panel* supports only in-plane shear forces.

The *twist panel* supports only moments about in-plane axes directed out of the sides.

The *membrane panel* supports both in-plane direct forces and in-plane shear.

All *plate* elements support various uncoupled loads. All of them support bending and transverse shear loads, but none will carry moments about an axis normal to the plane. In addition to the elements with their basic properties, there are more elaborate elements. One allows for the basic plate to be *nonhomogeneous* with respect to bending and shear. Two more are appended with membrane carrying ability, one being *homogeneous* and the other having distinct properties in all three actions suitable for representing *sandwich plates.*

Each three-dimensional element is axisymmetric and cannot be connected to any other type of element. Only the *conical shell* element can take unsymmetrical loads and differential stiffness. The two other three-dimensional elements can only deform axisymmetrically.

The *conical shell* is made of straight line generators but covers the range from cylinders to cones to discs. This element supports membrane, bending, and transverse shear loads.

The *toroidal shell* elements can have curved or straight line generators. These elements support membrane and bending loads.

Solid-of-revolution elements are rings with either triangular or trapezoidal cross-sections. They are used in the analysis of axisymmetric solids with axisymmetric loads.

If data for a structure are known partly or entirely in terms of stiffness data without reference to any element classification, it can be input in the form in which it exists. For example, if the influence coefficient matrix for a complicated structural component were obtained experimentally at a fixed number of connection points, then the *general* element would be used for the input. If a stiffness matrix were given for the entire structure, the *direct matrix input* format would be used.

Differential stiffness includes the effect of prestress on the linear deformations of the structure. It arises from a simultaneous consideration of large nonlinear motion and the applied load. Its implementation assumes that the applied loads remain fixed in magnitude and move with their points of application. Differential stiffness terms are developed for rods in extension; bars under the influence of moments, shear forces, and axial forces; plates under membrane loading without bending or transverse forces; and conical shells under axisymmetric axial loading without bending moments or transverse forces.

Scalar elements are the simplest forms of the elementary ingredients of reaction producing systems. They consist of (1) scalar *springs* that produce reactions when the relative distance between connections changes; (2) scalar *dampers* that produce reactions when there is a relative speed between connections; and (3) scalar *masses* that produce reactions when their associated joints undergo acceleration. Scalar models can represent structures in the abstract without any geometric significance; or a scalar system, such as a hydraulic mechanism, could be given geometric significance by connecting it to a degree-of-freedom at a grid point of a structural model to cause a force vector to develop there.

There are four *nonlinear* elements in NASTRAN. They are fundamental in nature and are capable of being used in conjunction with one another. A fairly wide variety of nonlinear behavior can consequently be modeled. The function generator element generates a nonlinear force according to data tabulated against computed displacements. The multiplier element generates a nonlinear force resulting from the product of two computed displacements. The power elements generate nonlinear forces by raising computed displacements to either positive or negative powers.

5.2 Material Properties

The material properties are represented in as general a form as possible and are listed in table 5.

Elastic moduli can be as simple as the usual Young's modulus, Poisson's ratio, and shear modulus, or the analyst can have the practical enhancement of detail afforded by orthotropic constants; or, when a three-dimensional element permits it, the analyst can employ a complete anisotropic set of 21 moduli. Moduli can be made to be temperature-dependent and stress-dependent. Nonlinear elastic relationships can be represented in tabular form.

Thermal moduli can also be isotropic, orthotropic, or completely anisotropic. Temperature dependence of thermal moduli can also be represented.

Mass is considered to be associated with grid points. It is commonly expressed in lumped form by treating the mass in the immediate vicinity of a point as being condensed at that point. If further detail of the mass is required, the first and second moments of the associated mass can be calculated with respect to a given point. Mass is subdivided into two categories: structural mass determined from the density of the load-carrying members and nonstructural mass determined from a distribution factor to represent such items as ablative coatings or acoustic blankets. In dynamics problems it is quite often necessary to represent the forces at one point due to the influence of mass at other points. This is called coupled mass, which is available for most elements. Weight is not considered as a primary parameter, but it can be determined from whatever gravitational acceleration is supplied. An associated computation can be carried out in the program for center-of-gravity or balance.

6. STATIC ANALYSIS

The program provides for a broad variety of static loadings which, through application of the rigid formats or the direct matrix abstraction program, will yield output data in various forms. The program is designed to handle a large number of degrees-of-freedom. However, the number of degrees-of-freedom and the attendant stiffness and mass data can be condensed by means of a Guyan reduction before applying loads. If nonlinear materials are involved, a NASTRAN rigid format using piecewise linear analysis can be invoked to iterate over each piecewise linear segment until the full magnitude of load has been reached.

6.1 Loads

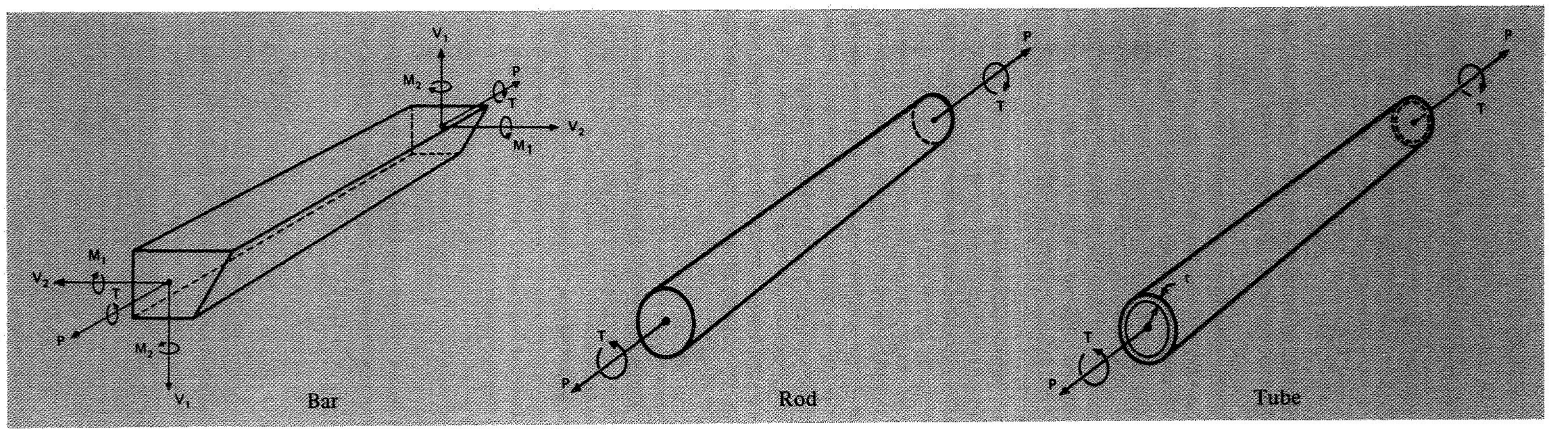
Static loads arise from four sources: surface loads, body forces, forces induced by scalar systems, and forces arising from enforced displacements (table 6).

Surface loads can be either point force loads applied at grid points or pressure loads which are converted to equivalent point force loads.

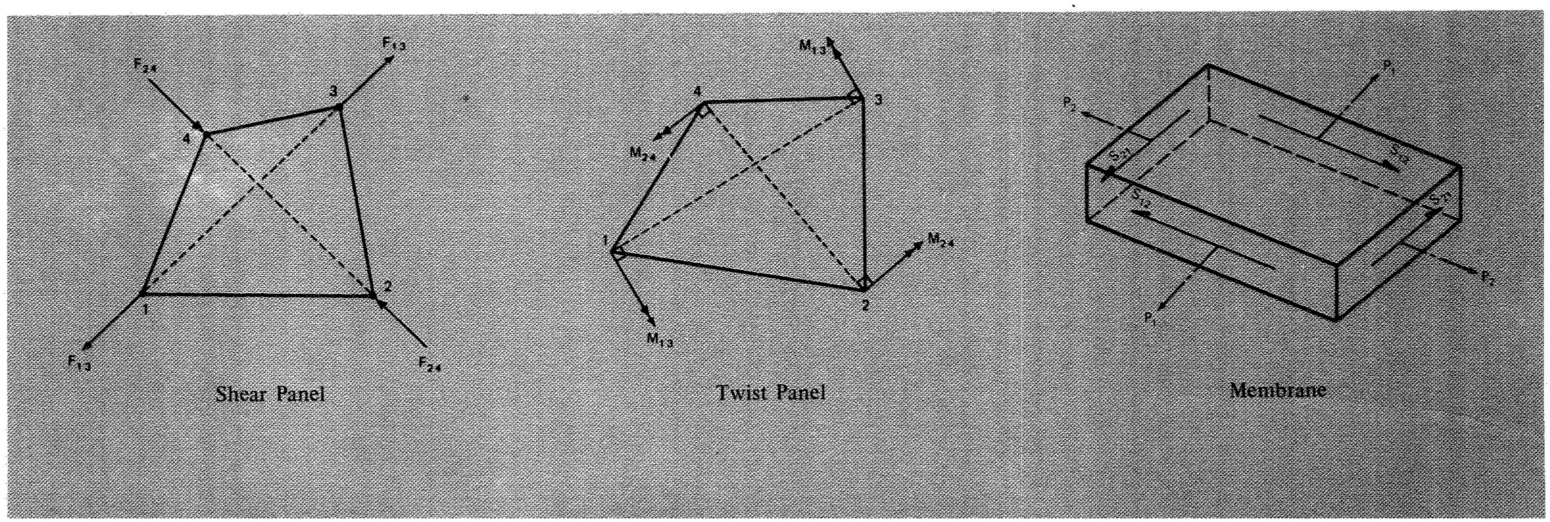
Body force loads come about from an acceleration field and are divided into three types: gravity loads, inertia relief loads, and rotational forces. Gravity loads are calculated internally for equivalent grid point loads according to the magnitude and direction of the specified gravity vector.

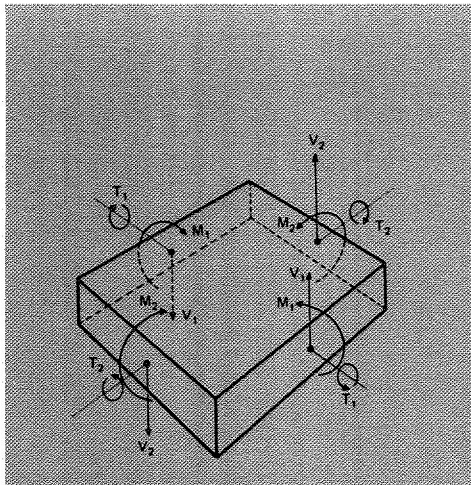
TABLE 4.—*Elastic Elements*

ONE-DIMENSIONAL

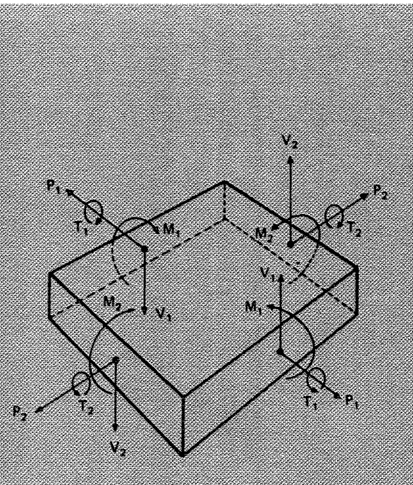


TWO-DIMENSIONAL

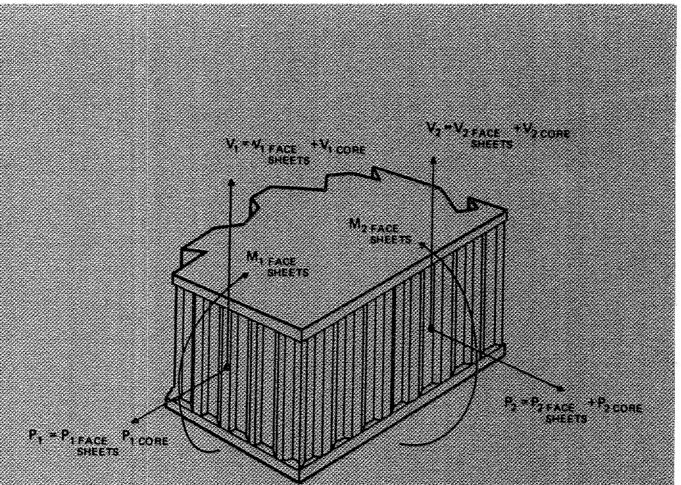




Nonhomogeneous Plate

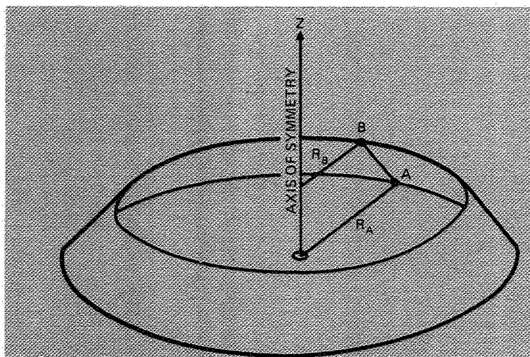


Homogeneous Plate

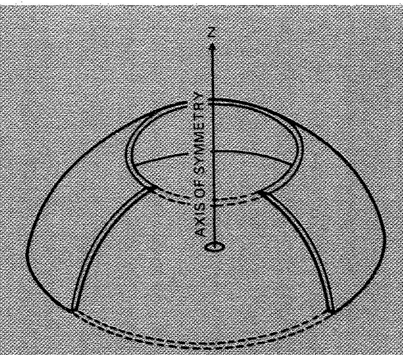


Sandwich Plate

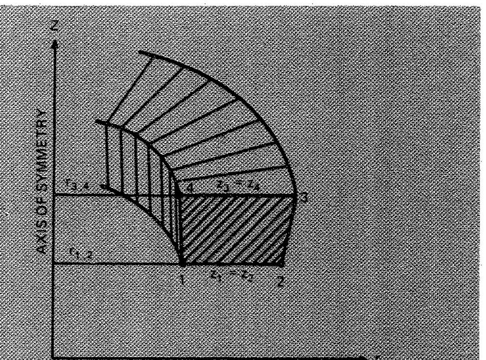
THREE-DIMENSIONAL



Conical Shell



Toroidal Shell



Solid of Revolution

General	Differential Stiffness	Scalar
General Element	Rod	Spring
Direct Matrix Input	Bar	Mass
	Plate	Damper
	Conical Shell	Nonlinear

TABLE 5.—*Material Properties*

Elastic moduli	Thermal moduli	Mass
Isotropic	Isotropic	Structural density
Orthotropic	Orthotropic	Nonstructural mass
Anisotropic	Anisotropic	Lumped properties
Temperature dependent	Temperature dependent	Coupled properties
Stress dependent		Weight
		Center of gravity

TABLE 6.—*Static Loads*

Surface	Body	Induced scalars	Enforced
Point force	Gravity	Voltage	Thermal
Pressure	Rotational	Hydraulic	Element deformation
	Steady accelerations		Grid point position

Similarly, inertia relief loads are calculated after static external equilibrating forces are replaced by D'Alembert forces and after steady acceleration is applied. Centrifugal loads resulting from steady rotation are determined from the specified point of rotation using rotational acceleration formulas and are then converted to equivalent grid point loads.

Forces induced from *scalar* systems are connected into structural systems as a vector. Two typical systems from which such forces are transduced are hydraulic and electric.

The final class of static forces are those that arise from *enforced* displacements such as a key link being too long in a bridge. Constrained systems subjected to a temperature profile produce enforced deformation loads, and, of course, any point of an elastic system constrained to retain a fixed position, regardless of the loading, is also enforced.

6.2 Output

Results of a problem can be controlled as to the type of item to be reported and the quantity of such data to be assembled. Static output includes:

(1) Grid point displacements
(2) Applied loads at grid points
(3) Forces at constrained grid points
(4) Element forces
(5) Element stresses
(6) Deformation plot

At a grid point the analyst can call for: (1) a recapitulation of input loads or, in the case of enforced deformation, the amount of load that was computed and actually applied; and (2) the elastic deformation and the amount of force that develops at constrained points. In the elements, the analyst can call for the average internal forces that develop or the average stresses at various points in the cross section. The most useful output of all, however, is a structural plot to an exaggerated scale to give a concise summary of the structural behavior under load.

7. EIGENVALUE PROBLEMS

Since NASTRAN is designed primarily for solving large-order matrices, several different situations in eigenvalue analysis arise for the analyst to obtain:

(1) A few roots of a large-order matrix
(2) All the roots of a large-order matrix
(3) A few roots of a complex matrix

Procedures are included in NASTRAN for the above situations to be used in buckling and vibration problems.

Various techniques are used to obtain eigenvalues as shown in table 7. *Root tracking* techniques are used to obtain a few roots of large

TABLE 7.—*Eigenvalues*

ROOT TRACKING	ROTATIONAL
Determinant method	Givens/Q–R method
Inverse power with shifts method	
Buckling loads and modes	

OUTPUT
Modal frequencies
Generalized mass
Mode shape normalization
  Generalized mass
  Maximum deflection
  Arbitrary component
Modal plots

matrices—both real and complex. Two root tracking schemes are available. One is the well-established determinant method; the other is a method formerly used for refinement but now organized as a self-contained technique called the inverse power method with shifts. One advantage of this method is that a root can be obtained with high accuracy regardless of its order of appearance in frequency. It is well suited for the determination of bifurcation buckling loads and modes.

To obtain a large number of roots, a *rotational* method called the Givens/Q–R method is available wherein all roots are obtained at once. The Givens method is used to tri-diagonalize real matrices only, and the Q–R method is used to extract the eigenvectors.

The *output* from an eigenvalue analysis lists the roots in order of ascending frequency. For each root a tabulation is made of the extraction order, eigenvalue, frequency, and generalized mass. After obtaining the eigenvectors from any of the methods, the analyst has three options for normalizing. They are with respect to the generalized mass, the maximum deflection in the mode, and a particular point of interest. The normalized modes can be plotted to any exaggerated scale either by themselves or superimposed on the undeformed structure.

8. DYNAMIC ANALYSIS

There are several options available to the analyst when solving dynamics problems, as indicated in table 8. Because dynamics problems usually require longer running times on the computer, the analyst will quite often want to *reduce the number of degrees-of-freedom* of his problem from that which was formulated for a statics solution. The Guyan reduction method is one way of accomplishing this, or the analyst may choose to remodel his structure using fewer degrees-of-freedom.

If the characteristics of the structure parts are obtained experimentally or if they come from an independent analysis, they can be *attached* to an existing analytical model in the form of modal degrees-of-freedom. To complete the modeling of a dynamic system, *damping* must be introduced. Three damping options are available: viscous damping, structural damping, and damping whose value is frequency-dependent. Solution of the dynamic differential equations can proceed by the Lagrangian *modal* approach if a sufficient number of modes were previously obtained from eigenvalue analysis, or *direct integration* can be carried out using finite differences in time.

There are essentially four different types of dynamics problems that can be solved—transient response, frequency response, random response, and elastic structure-control system stability. Each is described below.

8.1 Transient Response

The time-dependent forcing function or *dynamic load* is handled in a routine manner. Every point can be independently forced or, at the other

TABLE 8.—*Types of Dynamic Problems*

Dynamics			
Option to reduce degrees-of-freedom Option to augment with attached systems Specification of damping Option for modal method or direct integration			
Transient response	**Frequency response**	**Random response**	**Stability**
DYNAMIC LOAD Grid point force amplitude Grid point force phase Time delay	SINUSOIDAL LOAD Amplitude Phase Frequency range	AUTO-SPECTRUM OF GRID POINT LOAD Frequency range	CONTROL SYSTEM Sensors Signal conditioners Actuators
		TRANSFER FUNCTION	STRUCTURAL TRANSFER FUNCTION
Initial conditions Integration time step Nonlinear scalar	OUTPUT Displacement, velocity, acceleration, force, stress Transfer function General frequency response Curve plot	OUTPUT Frequency range Auto-spectrum Autocorrelation Curve plot	DYNAMIC LOAD Grid point force amplitude Grid point force phase Time delay Initial conditions Integration time step Nonlinear scalar
OUTPUT Grid point displacements, velocities, accelerations Grid point applied loads Element forces, stresses Deformation plot Curve plot			OUTPUT Grid point displacements, velocities, accelerations Grid point applied loads Element forces, stresses Deformation plot Curve plot

extreme, all points can be forced by loads equal in amplitude and in phase. Any variation between the two extremes can be handled as well. The time of application of a dynamic forcing function can be controlled as in the passage of a stress wave. Provision is made for setting up initial conditions in general. The coupling of nonlinear functions into the dynamic equations is available from the library of scalar elements. Finally the solution strategy is under the control of the analyst by his selection of integration time-step increments.

With regard to *output*, grid point data can be selected to include displacements, velocities, accelerations, and applied loads at the time and space intervals desired. Element output data can relate to forces or stresses at chosen time intervals. The most easily understandable form of output is not a printed listing but curve plots of the major areas of concern. A series of structural plots portraying the transient response at successive time intervals can be photographed and displayed as a motion picture.

8.2 Frequency Response

The generality that exists elsewhere in the program is maintained in the *sinusoidal loading* of the structure. The amplitude and phase can vary independently of the frequency and of each other in going from point to point on the structure. The frequency range over which the function is forced is selected by the analyst. The special case of damped, steady state, unit-synchronous sinusoidal loading will produce the control system transfer function or the Green's function in random response problems.

The *output* can be in terms of steady state grid point displacements, velocities, and accelerations. When the steady state amplitude and phase vary with frequency and position, the response function is general. A synchronous unity-forcing will give a special transfer function between the point or points of application and the points of interest. Curve plots of steady state response versus frequency are usually favored for transfer functions.

8.3 Random Response

The random analysis module in NASTRAN operates only in the frequency domain. Random input or forcing data is required in terms of its *auto-spectral density*. The analyst can regulate the frequency bandwidth for each increment of the spectrum as well as fix upper and lower frequency limits to the spectrum. The spectral signature of the structure comes from the frequency response module.

The response or *output* is obtained by applying the forcing spectral density function to any of the transfer functions for displacement, velocity, acceleration, force, or stress to give the autospectrum of response for each function respectively. A Fourier transform of the response autospectral density will yield the autocorrelation function if desired. All these results may be curve-plotted.

8.4 Stability

The program has the capability to analyze feedback problems such as the stability of an elastic structure–*control system* combination. This is accomplished by including control systems and actuators with the elastic structure in a transient analysis. The number of sensors that can be modelled, such as position or rate gyros, is not limited. Signal conditioners can also be manifold. Finally the number and types of control actuators can be as many as practically needed.

Structural transfer functions can be obtained from the frequency response module by calculating the response of the structure per unit amplitude of sinusoidal force at the locations of sensors and actuators. These transfer functions are used if the elastic structure–control system stability problem is solved in the frequency domain. Because of the ability to solve coupled differential equations, it is simpler to model the control system components and actuators in terms of their differential equations.

The *dynamic load* contains all of the characteristics described for the dynamic load under transient response above. Also, every feature of the *output* from transient response applies equally well to the elastic structure–control system stability.

9. PLOTTING

The NASTRAN program makes use of a broad variety of plotting capability on various types of plotters.

9.1 Plotters

The equipment for which NASTRAN was designed to interface are those routinely employed by the NASA Centers. Three types of plotters are included: table, microfilm, and incremental plotters. Two models of each type that have been used to date are listed in table 9. There is also a software package included in NASTRAN called the General-Purpose Plotter, which has reduced the commands to generic types. It needs only a small additional plotter-dependent translator of these generic commands to make it applicable to a plotter of any manufacture.

TABLE 9.—*Plotters*

Type	Model
Table	Electronics Associated EAI 3500 Benson-Lehner LTE or STE
Microfilm	Stromberg Carlson SC 4020 Display Data DD 80
Incremental	California Computer, CALCOMP 500 Series California Computer, CALCOMP 700 Series

9.2 Types of Plots

Two broad types of plots can be obtained as indicated in table 10—structures and curve plots.

Structure plotting.—The pictorial representation of the structural analytical model can involve as large or as small a portion of the specimen as the analyst decides. He may call for any of orthographic, perspective, or stereoscopic projections. The orientation from which the essentials can best be viewed are selected and then

TABLE 10.—*Types of Plots*

Structures plotting	Curve plotting
PORTION	ANY OUTPUT FUNCTION VS: *Time*
PROJECTION: *Orthographic*	*Frequency*
Perspective	COORDINATES: *Positions*
Stereoscopic	*Graduations*
VIEW	*Labels*
SCALE	QUANTITY: *No. of curves*
POSITION	*per plot*
LINES: *Density*	*No. of plots*
Color	*per sheet*
PAPER: *Transparency*	LINES: *Density*
Vellum	*Color*
Bond	PAPER: *Transparency*
FILM: *Negative*	*Vellum*
Positive	*Bond*
LABELS: *G.P. numbers*	FILM: *Negative*
G.P. symbols	*Positive*
Element numbers	TITLING: *Individual*
TITLING	*curve traces*
UNDEFORMED	*Individual*
DEFORMED: *Connected*	*coordinates*
Vectors	*Individual*
Undeformed underlay	*plots*

automatically calculated. With deformed structures the scaling of the grid point displacements can be as exaggerated as the analyst prefers in order to translate the actual displacements, which are ordinarily in mils, to a size that will make the behavior quite evident. Ordinarily the plot algorithm will automatically center a plot within the margins, but if for reasons of text or multiple plots per sheet it is more desirable to position the plot elsewhere, the analyst may make the selection. Contrast can be obtained on the microfilm plotters by modifying the black line density (i.e., line width); on the other plotters this can be done by changing not only the width but also by varying color. Details can be added by calling for joint numbering, element numbering, and joint symbols. Presentation of deformation behavior can be enhanced by underlaying the deformed plot with an undeformed plot, or by indicating deformations by vectors at individual grid points with respect to an underlay of a structure plot. Two examples of structure plotting are shown below.

Figure 1 is a plot of the structural model of the Rosman I satellite tracking antenna in the undeformed case. The structure was analyzed for

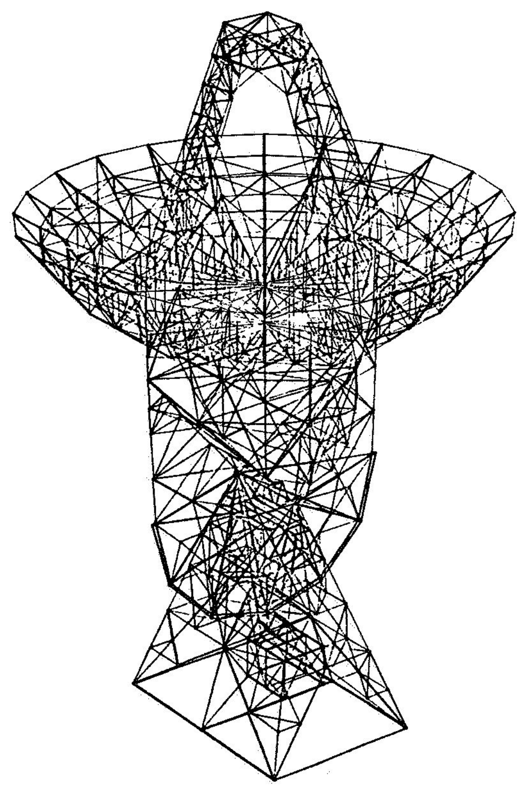


FIGURE 1.—Stereoscopic view of Rosman antenna.

vibration loads and transfer functions to be used for control system stability studies. The antenna was modeled with 3000 degrees-of-freedom and 80 modes were found in the 0- to 30-Hz range. Another example is shown in figures 2 and 3. Figure 2 is an artist's concept of the VISSR (visible infrared spin scan radiometer) that is to be flown in the SMS (Synchronous Meteorological Satellite). Figure 3 shows the 800 degrees-of-freedom structural model (with portions of the substructure omitted) of the VISSR that was analyzed for strength and vibration characteristics. The use of labels is demonstrated on the structural model.

Curve Plotting.—Instead of looking at total deformation of the structure, it may be more meaningful to explore time histories or frequency histories of element behavior and/or grid point behavior, such as stress, velocity, or autocorrelation. Coordinate lines can be arranged in almost any manner with regard to position, graduations, and labels. More than one curve can be drawn on a given plot, and more than one plot can appear on a page. Line density or color can be called out for contrast. Considerable latitude is offered in titling the curve traces, coordinate lines, and the complete display. Two examples of curve plots are shown below in figures 4 and 5. Figure 4 is a plot of the transient response of a free beam, weighted at one end and forced sinusoidally at 60 Hz on the unweighted end for a tenth of a second. The plot shows the displacement time history of the unweighted end. Figure 5 results from a random analysis of a simply supported beam exposed to a flat spectrum in the 0- to 100-Hz range cross-correlated among three central points. The plot shows the auto-spectral density of the center of the beam in the 0- to 90-Hz range.

10. DOCUMENTATION

During the development of NASTRAN a deliberate attempt was made to assure that the program is well documented. As a result the documentation that is provided should contain the answers to most of the questions that could be raised by the theoretician, programmer, and user, since each has a manual designed specifically for his needs. To provide for updating, the

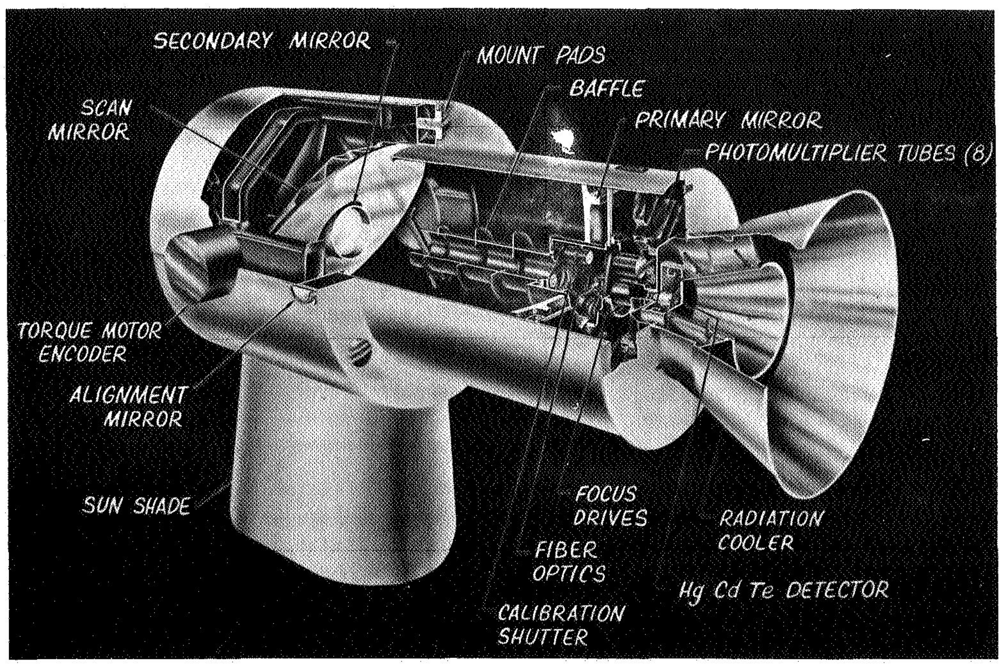


FIGURE 2.—Artist's concept of the visible infrared spin scan radiometer.

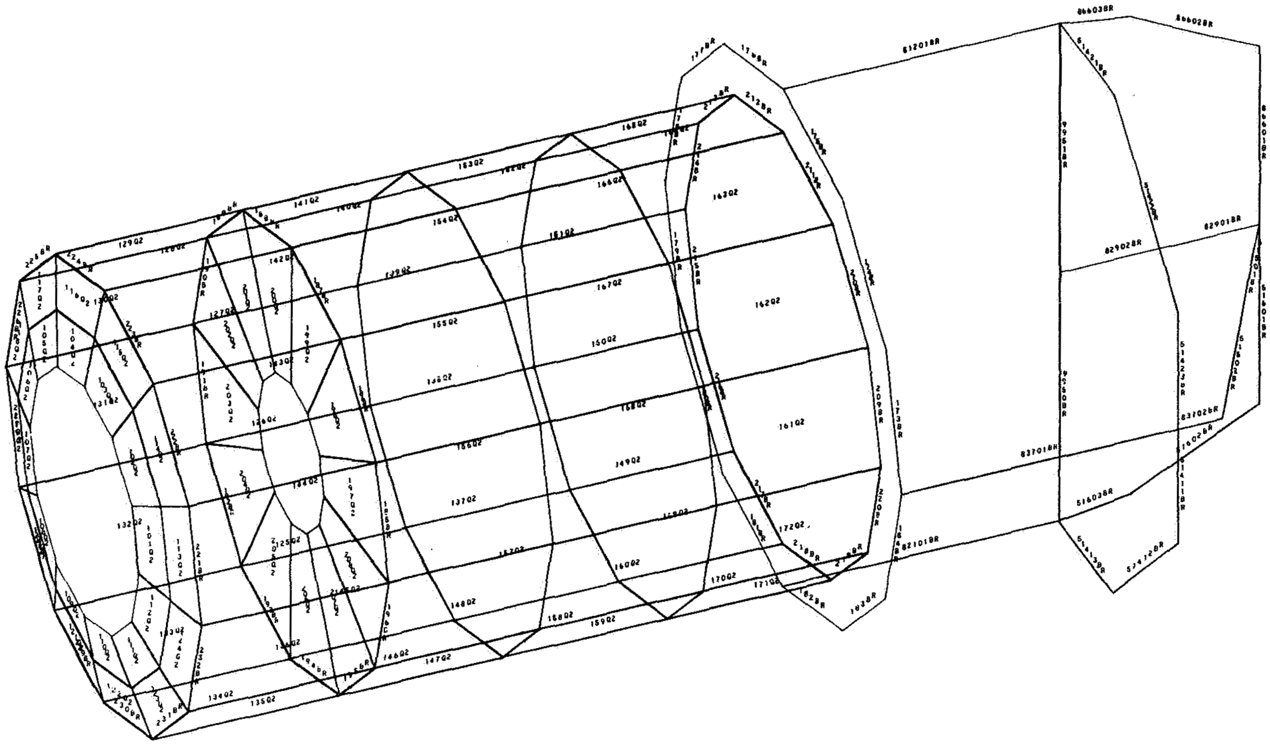


FIGURE 3.—VISSR 800 degrees-of-freedom structural model.

manuals are not bound but are punched for insertion in a three-ring binder. The separate manuals that are available are

(1) NASA SP-221, *NASTRAN Theoretical Manual*
(2) NASA SP-222, *NASTRAN User's Manual*
(3) NASA SP-223, *NASTRAN Programmer's Manual*
(4) NASA SP-224, *NASTRAN Demonstration Problem Manual*

10.1 Theoretical Manual

The *Theoretical Manual,* written for the structural analyst, explains the analytical and numerical procedures that underlie the program by presenting derivations and explanations to help the analyst assess NASTRAN's suitability for the needs of his particular problem. Where it is germane, the theoretical topics are embellished with comments on program organization and data processing. The *manual* is divided into the following 12 sections:

Executive system.—An overview of the entire program is given, which contains a discussion of the requirements imposed on the executive by the needs of the general-purpose functions, design considerations, and operation sequence. The section ends with a discussion on how the analyst exercises control of his problem.

Matrix operations.—The solution strategies of matrix add, multiply, transpose, partition, and merge are interpreted according to the character of the matrices involved in structural analysis.

Static analysis.—Solutions of statics problems are considered from the standpoints of input processing, problem solution, and output processing. Essentials are defined and then the five static solution cases are walked-through with reference to flow diagrams and the gross mathematical operations.

Structural elements.—Theoretical considerations of the structural elements and the means of implementing them in the program are presented. In addition to the usual one- and two-dimensional elements and the axisymmetric three-dimensional elements, the topics of constraint elements, scalar elements, and general elements are included. The formulations for stiffness, mass properties, and stress recovery are outlined only briefly with references to mathematical detail in the *Programmer's Manual.*

Differential stiffness and buckling.—A theoretically logical basis is laid for the development of differential stiffness in general, and the application of this theory to each of the elements having differential stiffness follows. Buckling is handled as an eigenvalue problem.

Dynamic analysis organization.—The case types dealing with dynamics problems are managed in a unified manner. The assembly of the equations is centralized, and then the section branches off for particular treatment of each of seven cases. Starting with real eigenvalue analysis, the dynamic management subsequently splits operations between the modal approach and the direct approach for complex eigenvalue, transient, and frequency response cases. It closes with the organization for dynamic data recovery.

Eigenvalue extraction.—The three eigenvalue routines of Determinant, Inverse Power, and Givens are introduced in terms of their general characteristics. The details of the theory for each method, aided by flow diagrams, follow in succession.

Transient analysis.—Subsections deal with (1) the methods of formulating the transient load vector, (2) the complications arising from nonlinear terms, (3) integration of coupled differential

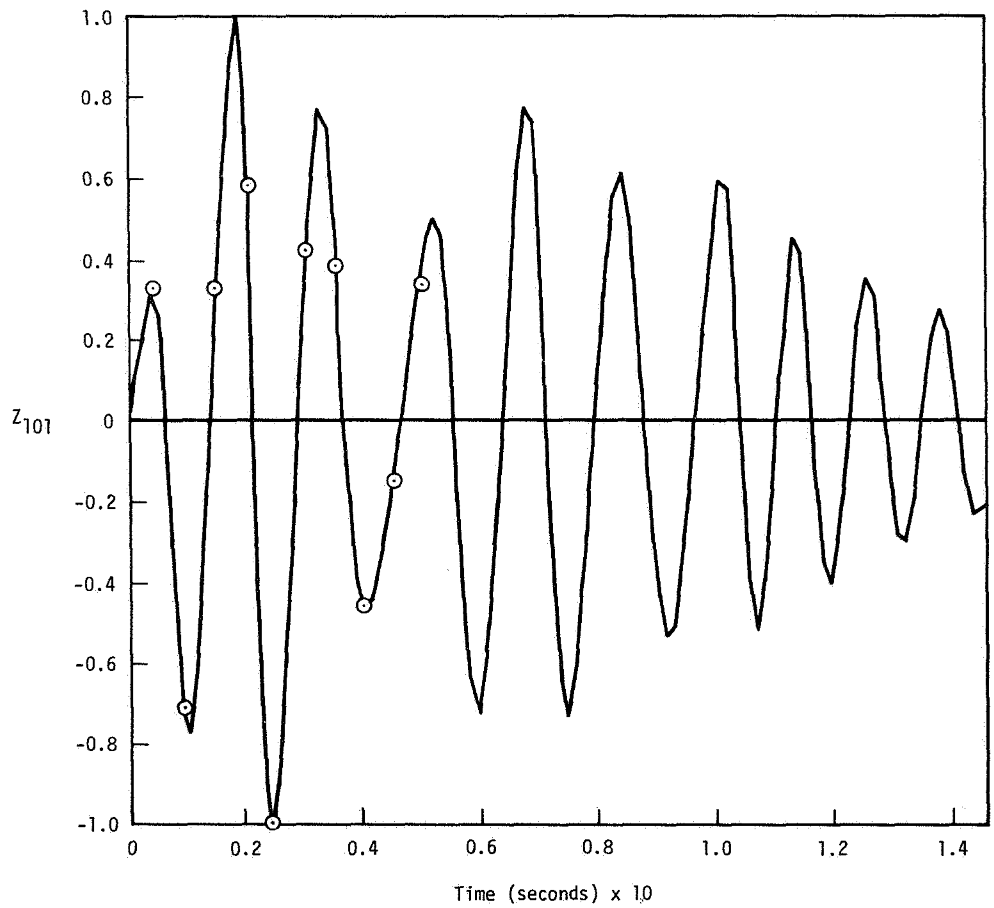


FIGURE 4.—Comparison of NASTRAN and analytic displacements versus time.

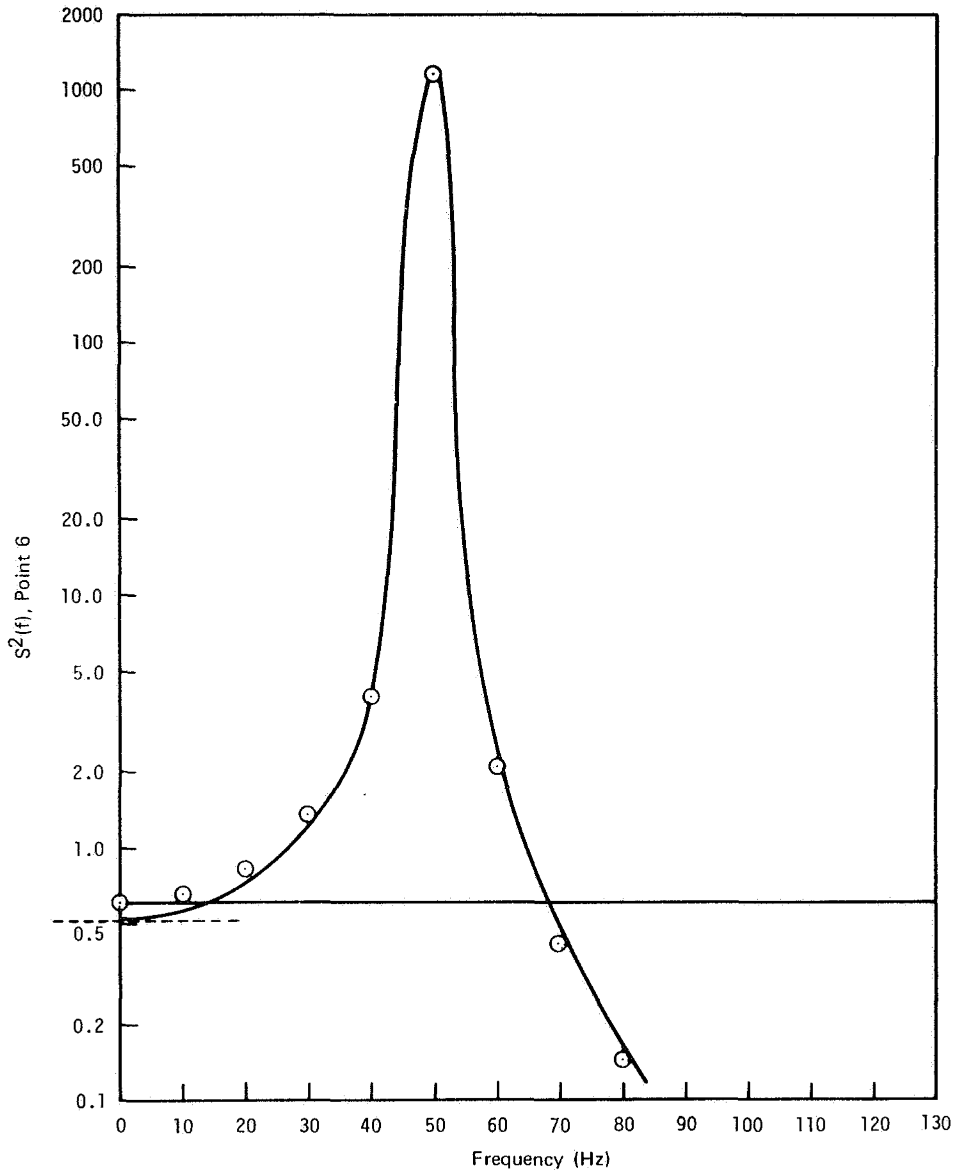


FIGURE 5.—Power spectral density of center point displacement.

equations, and (4) the integration of the uncoupled differential equations. Special emphasis is given to the influence of convergence requirements on the coefficients of the direct integration formulas.

Frequency response and random analysis.—This section briefly tells that NASTRAN implements these two features in a straightforward way.

Computer graphics.—Only those graphical topics needing theoretical attention are treated. Transformation relationships between viewer and object are defined. Discussions are provided to help the user establish the values of scale, vantage point, projection plane separation, and ocular separation for perspective and stereoscopic plots.

Special modeling techniques.—The use of NASTRAN for special applications from the formulations already contained in existing rigid formats is contained in this special section. Representation of part of a structure by its vibration modes (sometimes called dynamic partitioning or component mode synthesis) is in the first part, and the representation of control systems is in the second part.

Error analysis.—The basis for specifying double precision for all matrix calculations is explored from a "worst case" standpoint. In order to guide the analyst in setting up his structural model when plate elements are used, the error magnitude versus mesh size is depicted for the NASTRAN plate element in contrast to other well accepted elements.

10.2 User's Manual

The *User's Manual* describes how to operate NASTRAN and is the primary guide to preparing input. It introduces the user to modeling considerations with references to the other two manuals for further details. Instructions are given on the entries to be made for problem definition, problem control, computer usage, plotter usage, and diagnostic interpretation. Its contents are divided into the following seven sections:

Structural modeling.—Structural data submission depends on how the program is organized to handle it. This organization and the dependent relationships that must be observed are explained with respect to how these factors will influence a user's way of modeling a structure. Topic by topic the user is introduced to the name and nature of the input data cards contained in the next section. References to explanations taken up in the *Theoretical Manual* are liberally supplied.

NASTRAN data deck.—Functions of the three parts of the data deck are explained. Every possible card is individually presented. The requirements and implications of each field are given in detail. There is a detailed description of over 200 possible input data cards.

Rigid formats.—The overall organization of every rigid format is discussed first, giving the function of each major part. This is followed by a listing of the Direct Matrix Abstraction Program (DMAP) statements which constitute each of the 12 rigid formats (see section 2). After each such listing is an explanation of every major step with supporting mathematics where appropriate; then come the tables for problem continuation, and, finally, associated problem case control entries and sundry parameters employable by the user are discussed.

Plotting.—A general discussion of types of plots and plotters available in NASTRAN is followed by a detailed description of every type of plot control card. A number of examples are included to amplify the text.

Direct matrix abstract.—This section provides the user with the rules that will allow him to understand the rigid format DMAP sequences, write alteration packages, and construct his own DMAP sequences using the many modules contained in the NASTRAN DMAP repertoire. Individual descriptions are given for the many nonstructurally oriented modules in the DMAP library. The section concludes with a number of examples of correct DMAP usage.

Diagnostic messages.—NASTRAN messages are catalogued under various headings and listed serially under each heading. Messages are differentiated between fatal, warning, and information.

NASTRAN dictionary.—Various NASTRAN terms are defined, codified as to area to which they pertain, and referenced to a manual section for further explanation.

10.3 Programmer's Manual

The *Programmer's Manual* is a narrative description of the coding design quite distinct from the listing of the coding source statements. It describes the executive system and the coding practices used in it, explains the function of each module and subroutine (including the mathematical equations implemented in each instance), and contains information necessary for maintenance and modification of the program. Its contents are divided into the following six sections:

NASTRAN programming fundamentals.—An overview of the program is written as background to all subsequent material. It outlines the program objectives, program organization, executive system, program design features such as programming language versus computers, input/output, matrix routines, module executions, and restarting.

Data block descriptions.—This section contains descriptions of data blocks, which are the principal means of data communication between the program's functional modules and the executive system. In addition, it contains descriptions of executive tables and miscellaneous tables that are accessed by a class of modules.

Subroutine descriptions.—Those subroutines that are not an integral part of a module are described. They are classified into executive, utility, and matrix subroutines. First they are indexed and then individually described in a uniform format of title, entry point, purpose, calling sequence, method, and design requirements.

Module functional descriptions.—Modules have been classified into seven categories:

(1) Executive preface modules
(2) Executive modules
(3) Executive DMAP instructions
(4) Executive DMAP modules
(5) Functional modules
(6) Output modules
(7) Matrix modules

They are indexed alphabetically and individually described according to a uniform format of name, entry point, purpose, calling sequence, input data, output data blocks, method, subroutines, design requirements, and diagnostic messages. The mathematical steps are given in the sequence in which they are executed.

NASTRAN—operating system interfaces.—Interfaces are explained for the following:

(1) IBM 7094/7040 (44) direct coupled computer under the IBSYS operating system
(2) IBM 360 computer models 50 through 95 under OS360 operating system
(3) UNIVAC 1108 computer under the EXEC8 operating system
(4) CDC 6600 computer under the SCOPE 3.0 operating system

Interfaces for all computers are discussed with respect to:

(1) Input/output
(2) Link switching
(3) Overlay considerations and implementation of the open core concept
(4) Setup of the operating system control cards associated with the NASTRAN data decks
(5) Generation of an executable NASTRAN system
(6) Machine-dependent routines

Modifications and additions to NASTRAN.—Restrictions on the FORTRAN IV languages are specified in order to produce equivalent object code across the several computers on which NASTRAN operates. Individual phases are examined for modification impact, ending with the addition of a new link.

10.4 Demonstration Problem Manual

The *Demonstration Problem Manual* states the nature of each problem, the features of the program that it is intended to exercise, the references for the solution basis, and the comparison between the NASTRAN solution and the basis solution. The problems are as follows:

(1) Delta-wing analysis—static solution under point loads
(2) Problem 1 restarted with load relocated
(3) Problem 1 restarted for continuation into eigenvalue analysis
(4) Pressure load on a spherical cap resting on a smooth plane
(5) Problem 4 restarted with clamped boundary
(6) Thermally loaded plate having temperature dependent material
(7) Transverse line load on a long orthotropic plate
(8) Problem 7 restarted with modified output request
(9) Cylinder modeled with conical elements, under nonsymmetric bending
(10) Solid disc under axisymmetric thermal load, modeled with solid axisymmetric element
(11) Shallow spherical shell under pressure load, modeled with toroidal elements
(12) Inertia relief analysis of a circular ring under concentrated and centrifugal loads
(13) Vibration of a rectangular plate
(14) Differential stiffness of a 100-cell beam under axial and bending loads
(15) Symmetric buckling of a cylinder
(16) Piecewise linear analysis of an axially loaded cracked plate

(17) Complex eigenvalue analysis of a 500-cell string using scalar elements
(18) Frequency response analysis of a rectangular plate by the direct method
(19) Transient analysis with direct matrix input
(20) Traveling wave on a 1000-cell string by transient analysis
(21) Complex eigenvalue analysis of a rocket guidance and control system by the modal method
(22) Frequency response and random analysis of a 10-cell beam by the modal method
(23) Transient analysis of a free 100-cell beam by the modal method

11. USER EXPERIENCE

Although the released version of NASTRAN has been intensively field tested, its magnitude and complexity are such that undetected errors are bound to exist. With the number of users increasing steadily, more combinations are being exercised with attendant discovery of errors; however, the remarks that follow are based upon reports from users in about 30 different organizations. This section gives the prospective user a gross picture of the extent to which certain parts of the system have been used, a measure of remedial activity, and a qualitative appraisal of reliability. A complete status report is beyond the scope of this publication, but it is intended that the subsequent paragraphs provide a flavor of user experience.

Error corrections are being made on a continuing basis through the NASA system. Experience with statics rigid formats began in May 1967. All of the one- and two-dimensional elements with isotropic materials have been used sufficiently to declare all but one of them reliably checked out. There is a discrepancy that still exists in the shear panel when the sides are almost parallel. The only known outstanding error remaining in one- and two-dimensional orthotropic elements is in the stress recovery of the membrane. Experience is less, and therefore the reporting has been less on the three-dimensional elements; however, the only known outstanding errors are minor. Static loadings and constraints have also been well exercised, and all reported errors have been cleared up.

Root tracking routines for real eigenvalues were put into service in August 1967. Many correction cycles have since transpired. The current activity centers about improvements in execution times. The Givens routine was activated in June 1969. Sufficient usage has since occurred to uncover impediments that are now fixed, but less than 20 percent of its possible permutations have been tried. The run-time for matrices of order up to 300 are competitive timewise with root-tracking methods. Complex eigenvalue analysis has been limited, but it has produced worthwhile corrective activity. The system has been applied to a number of classical structural problems and correlates well with known results, but it has had limited application to complicated problems in complex eigenvalue analysis. Separation of eigenvectors from roots of high multiplicity needs much more extensive testing in all routines, both real and complex.

Buckling, differential stiffness, and piecewise linear analysis have been in service since April 1969, but have been given routine checking with only sparse applications. Very little field reporting has been received. Dynamic transient, random response, and frequency response analyses have been in service since April 1969, and have been used substantially less than basic static analysis. Several serious logical deficiencies in the integration algorithm have been corrected. Applications of nonlinear elements in dynamics have revealed that for the present, if the coefficient matrices for any term of the differential equation is null, it is advisable to insert a minute dummy quantity in order to enhance convergence.

A limitation on restart that definitely needs correction is the inability to obviate a repetition of an eigenvalue analysis when a modal vibration analysis is followed by a modal transient analysis. Other restarts have been found to be reasonably successful.

The highest corrective activity has occurred in plotting and diagnostics. This is not surprising, since it is in these two areas that the analyst insists on high reliability. Operationally, plotting appears to be free of major limitations, although a few minor problems are still being reported. The greatest current emphasis is being directed toward documentation clarification. Diagnostics had been found to be too unspecific in many instances. In others, diagnostics were found to be lacking upon the occurrence of anomolies, but the bulk of these have been attended to. Field reporting is expected to remain high in the area of diagnostics for several more years as a consequence of broadening the number of NASTRAN users.

An advantage of having the NASTRAN system centrally managed is that duplication of error discoveries will not result in duplication of corrective action. Subsequent periodic releases of NASTRAN can then incorporate all corrections processed at the time of the release. Field reporting can be channeled and then correlated so that all users can benefit from having a source of reliable information about the system.

U. S. GOVERNMENT PRINTING OFFICE : 1971 O - 420-242

"The aeronautical and space activities of the United States shall be conducted so as to contribute . . . to the expansion of human knowledge of phenomena in the atmosphere and space. The Administration shall provide for the widest practicable and appropriate dissemination of information concerning its activities and the results thereof."

— NATIONAL AERONAUTICS AND SPACE ACT OF 1958

NASA SCIENTIFIC AND TECHNICAL PUBLICATIONS

TECHNICAL REPORTS: Scientific and technical information considered important, complete, and a lasting contribution to existing knowledge.

TECHNICAL NOTES: Information less broad in scope but nevertheless of importance as a contribution to existing knowledge.

TECHNICAL MEMORANDUMS: Information receiving limited distribution because of preliminary data, security classification, or other reasons.

CONTRACTOR REPORTS: Scientific and technical information generated under a NASA contract or grant and considered an important contribution to existing knowledge.

TECHNICAL TRANSLATIONS: Information published in a foreign language considered to merit NASA distribution in English.

SPECIAL PUBLICATIONS: Information derived from or of value to NASA activities. Publications include conference proceedings, monographs, data compilations, handbooks, sourcebooks, and special bibliographies.

TECHNOLOGY UTILIZATION PUBLICATIONS: Information on technology used by NASA that may be of particular interest in commercial and other non-aerospace applications. Publications include Tech Briefs, Technology Utilization Reports and Notes, and Technology Surveys.

Details on the availability of these publications may be obtained from:

SCIENTIFIC AND TECHNICAL INFORMATION OFFICE

NATIONAL AERONAUTICS AND SPACE ADMINISTRATION

Washington, D.C. 20546