NONLINEAR RANDOM RESPONSE PREDICTION USING MSC/NASTRAN

J. H. ROBINSON
NASA LANGLEY RESEARCH CENTER
HAMPTON, VIRGINIA

C. K. CHIANG
OLD DOMINION UNIVERSITY
NORFOLK, VIRGINIA

S. A. RIZZI
NASA LANGLEY RESEARCH CENTER
HAMPTON, VIRGINIA

OCTOBER 1993

NASA Technical Memorandum 109029

NASA Langley Research Center
Hampton, Virginia 23681-0001
Abstract

An equivalent linearization technique has been incorporated into MSC/NASTRAN to predict the nonlinear random response of structures by means of Direct Matrix Abstract Programming (DMAP) modifications and inclusion of the nonlinear differential stiffness module inside the iteration loop. An iterative process was used to determine the rms displacements. Numerical results obtained for validation on simple plates and beams are in good agreement with existing solutions in both the linear and linearized regions. The versatility of the implementation will enable the analyst to determine the nonlinear random responses for complex structures under combined loads. The thermo-acoustic response of a hexagonal thermal protection system panel is used to highlight some of the features of the program.
Contents

Abstract .............................................................................................................................. i
List of Tables .................................................................................................................. iv
List of Figures ................................................................................................................ v
Section 1 Introduction ..................................................................................................... 1
Section 2 Theory ............................................................................................................. 3
  2.1 Large Deflection Finite Element Formulation ....................................................... 3
  2.2 Equivalent Linear Stiffness Matrix [K_e] ............................................................... 7
  2.3 Iterative Equivalent Linearization Solution Methods ......................................... 8
  2.4 Implementation ..................................................................................................... 11
Section 3 Programmer’s Notes ..................................................................................... 13
  3.1 SEMELRR Main SUBDMAP ................................................................................. 13
  3.2 Updated Displacement Calculation ...................................................................... 16
  3.3 Output Requests .................................................................................................. 17
Section 4 Validation of The SEMELRR Solution Sequence .......................................... 18
Section 5 Example Problems ........................................................................................ 21
  5.1 Problem Execution ............................................................................................... 21
  5.2 Linear Random Analysis ...................................................................................... 24
  5.3 Nonlinear Random Analysis ................................................................................ 29
  5.4 Static and Nonlinear Random Analysis ................................................................ 37
Summary .......................................................................................................................... 43
Acknowledgment ............................................................................................................ 44
References ....................................................................................................................... 45
Appendix A Main SUBDMAP ......................................................................................... A1
Appendix B Additional User Defined Parameters ....................................................... B1
Appendix C SUBDMAP SEDRCVR ............................................................................. C1
Appendix D SUBDMAP SUPER3 ................................................................................ D1
List of Tables

Table 1  Dimensions for hexagonal TPS panel example problem  ................. 39
List of Figures

Figure 1  Design environment for a generic hypersonic vehicle .......................... 2
Figure 2  Flowchart for equivalent linearization solution procedure .................... 11
Figure 3  SUBDMAP call tree of SEMELRR solution sequence .................................. 14
Figure 4  Effect of acoustic excitation level on maximum deflection for beams....... 18
Figure 5  Effect on acoustic excitation level on maximum deflection for clamped plate. 19
Figure 6  Amplitude versus frequency ratio for clamped plate ............................ 20
Figure 7  Demonstration Bulk Data Deck .......................................................... 24
Figure 8  Output from linear random analysis ................................................. 28
Figure 9  Finite element model ........................................................................ 40
Figure 10 Deformed plot of the thermal displacement vector. Displacements are given
        in inches ................................................................................................. 41
Figure 11 Thermal stresses in the radial direction εr in psi .................................... 41
Figure 12 Deformed plot of the root-mean-square thermal-acoustic displacement vector.
        Displacements are given in inches ......................................................... 42
Figure 13 Root-mean-square thermal-acoustic stresses in the radial direction εr in psi. 42
Section 1 Introduction

The current trends in advanced vehicle development show a need for lighter, more economical structural components. This trend, coupled with the increasing propulsion and environmental loads associated with these vehicles, has renewed interest in nonlinear structural response. This is most evident in, but not necessarily limited to, the aerospace industry with such proposed vehicles as the National Aero-Space Plane (NASP) and the High Speed Civil Transport (HSCT). Surface panels, particularly those exposed to the engine noise and jet exhaust and those in the region of shock boundary layer interactions, are anticipated to respond nonlinearly in at least part of the flight regime. Figure 1 depicts the thermo-acoustic loads on a single-stage-to-orbit vehicle. Other intense random loads may be transmitted through the structure from engine mounts or other hard points. To effectively and economically evaluate these structural components, a practical method of predicting their large deflection random response is required.

There are several methods currently in use to predict the large deflection random response of structures. A perturbation method [1] based on classical perturbation theory for nonlinear deterministic motion can be used to obtain approximate solutions to weakly nonlinear systems. A stochastic averaging method [2] yields approximate solutions when the damping is light and the excitation is broadband. This method has been applied principally to single-degree-of-freedom systems. The Fokker-Plank-Kolmogorov (FPK) approach [3] is the only method that yields an exact solution, but solutions are only available for a few restricted classes of problems. The numerical simulation technique, also referred to as the Monte Carlo method [4], is the most general method and yields the best results of all the approximate methods. A substantial drawback to the Monte Carlo method is the computational time required to solve realistic structural problems. The most widely used method is the equivalent linearization method [5]. It yields good approximate solutions for the statistics of the random response of simple and complex structures and lends itself to an incremental solution procedure similar to the methods employed in static nonlinear problems.

The equivalent linearization method for obtaining nonlinear random responses was an obvious choice for implementation in a commercial package. The technique has been used, refined, and validated by many authors [6—10]. The validation of the method is well documented by many authors for beams, plates, and other nonlinear dynamic structures. The refinements include methods for solving structural problems with thermal and acoustic loads, initial stresses, and imperfections. Techniques have been developed, for example, for the random response of pre- and post-thermally and mechanically buckled plates, linear and nonlinear statically deflected panels, and various combinations of concentrated and distributed random loads. The equivalent linearization procedure has been applied primarily in research or special purpose codes, so a general purpose finite element code incorporating this procedure was unavailable.

The MacNeal-Schwendler Corporation version of NASTRAN (MSC/NASTRAN) [11] was selected for this work due to its extensive use in the aerospace and automotive industries, where nonlinear random phenomena are most prevalent. The equivalent linearization
procedure was programmed as a "stand alone" solution sequence for version 67 using the Direct Matrix Abstraction Program (DMAP) [12] language. It was found that all the necessary components of the equivalent linearization procedure already existed as DMAP modules. The essence of the new solution sequence therefore consisted of incorporating the necessary modules and iterative procedures into an existing MSC/NASTRAN solution sequence for linear random analysis. Two solution sequences were available to serve as starting points: the Super Element Modal Frequency Response (SEMFREQ) and Super Element Direct Frequency Response (SEDFREQ) solution sequences. The SEMFREQ was chosen for reasons described in this report.

The large deflection finite element formulation is first reviewed to establish the general nonlinear equations of motion. The theory of equivalent linearization is then presented and the expression for the equivalent linear stiffness is derived. An overview of the iterative implementation of the equivalent linearization procedure is presented in flow chart form with consideration to the various methods of solving dynamic systems. The ease with which the expression for the equivalent linear stiffness is evaluated in multi-degree-of-freedom systems is somewhat dependent on the method used to form and solve the equations of motion. The evaluation of the equivalent linear stiffness and the particulars of programming the new solution sequence are presented for broad-band Gaussian loads and modal equations of motion. In the validation section of this report, textbook examples are used to compare the MSC/NASTRAN equivalent linearization solution sequence with published results. A series of simple plate problems are presented to show potential users how to use the solution sequences to solve a variety of problems. A final example problem is shown to demonstrate the ability of the solution sequence to efficiently solve complex structural problems.

![Figure 1: Design environment for a generic hypersonic vehicle](image-url)
Section 2 Theory

The large deflection nonlinear finite element formulation is first reviewed for the determination of the system matrices. The equivalent linearization technique is then introduced for the solution of the nonlinear equations of motion. Several special cases are then considered for the determination of the equivalent linear stiffnesses.

2.1 Large Deflection Finite Element Formulation

The large deflection nonlinear strain-displacement relationships as taken from elasticity [13] are:

\[
\varepsilon_x = \frac{\partial u}{\partial x} + \frac{1}{2} \left[ \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial x} \right)^2 + \left( \frac{\partial w}{\partial x} \right)^2 \right]
\]

\[
\varepsilon_y = \frac{\partial v}{\partial y} + \frac{1}{2} \left[ \left( \frac{\partial u}{\partial y} \right)^2 + \left( \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial w}{\partial y} \right)^2 \right]
\]

\[
\varepsilon_z = \frac{\partial w}{\partial z} + \frac{1}{2} \left[ \left( \frac{\partial u}{\partial z} \right)^2 + \left( \frac{\partial v}{\partial z} \right)^2 + \left( \frac{\partial w}{\partial z} \right)^2 \right]
\]

\[
\gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} + \left( \frac{\partial u}{\partial x} \frac{\partial v}{\partial y} + \frac{\partial v}{\partial y} \frac{\partial u}{\partial x} + \frac{\partial w}{\partial y} \frac{\partial w}{\partial x} \right)
\]

\[
\gamma_{xz} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} + \left( \frac{\partial u}{\partial x} \frac{\partial w}{\partial z} + \frac{\partial w}{\partial z} \frac{\partial u}{\partial x} + \frac{\partial w}{\partial x} \frac{\partial w}{\partial z} \right)
\]

\[
\gamma_{yz} = \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} + \left( \frac{\partial v}{\partial y} \frac{\partial w}{\partial z} + \frac{\partial w}{\partial z} \frac{\partial v}{\partial y} + \frac{\partial w}{\partial y} \frac{\partial w}{\partial z} \right)
\]

where \(u, v,\) and \(w\) are the three displacements, \(\varepsilon_x, \varepsilon_y,\) and \(\varepsilon_z\) are the normal strains, and \(\gamma_{xy},\) \(\gamma_{xz},\) and \(\gamma_{yz}\) are the shear strains. All are functions of \(x, y,\) and \(z.\)

For an arbitrary finite element, assume nondimensional shape functions, \(N_n,\) such that

\[
u(x, y, z) = \{N_1 \ N_2 \ \cdots \ N_n\} \begin{bmatrix} u_1 \\ u_2 \\ \vdots \\ u_n \end{bmatrix}
\]

\[
v(x, y, z) = \{N_1 \ N_2 \ \cdots \ N_n\} \begin{bmatrix} v_1 \\ v_2 \\ \vdots \\ v_n \end{bmatrix}
\]
\[ w(x, y, z) = \{N_1 \ N_2 \ \ldots \ \ N_n\} \begin{bmatrix} w_1 \\ w_2 \\ \vdots \\ w_n \end{bmatrix} \quad (4) \]

where \{u_1 \ u_2 \ \ldots \ u_n\}, \{v_1 \ v_2 \ \ldots \ v_n\}, \text{and} \ \{w_1 \ w_2 \ \ldots \ w_n\} \text{are the vectors of nodal displacements for the } n \text{ nodes of the element.}

The matrix form of equation (1) is [14]

\[ \{\varepsilon\} = \{\varepsilon_L\} + \{\varepsilon_N\} \]
\[ = [H_L] \{q\} + \frac{1}{2}[H]\{\theta\} \quad (5) \]

where

\[ \{\varepsilon\} = \{\varepsilon_x \ \varepsilon_y \ \varepsilon_z \ \gamma_{xy} \ \gamma_{xz} \ \gamma_{yz}\}^T \]
\[ \{q\} = \{u_1 \ v_1 \ w_1 \ \ldots \ u_n \ v_n \ w_n\}^T \quad (6) \]
\[ \{\theta\} = \{u, x \ \ v, x \ \ w, x \ \ u, y \ \ v, y \ \ w, y \ \ u, z \ \ v, z \ \ w, z\}^T \]

The subscripts \(L\) and \(N\) denote the linear and nonlinear part of the total strain, respectively, and superscript \(T\) denotes transpose of a quantity.

The variation of the strain, \(\{\varepsilon\}\), is expressed as

\[ \{\delta\varepsilon\} = \{\delta\varepsilon_L\} + \{\delta\varepsilon_N\} \]
\[ = [H_L] \{\delta q\} + \frac{1}{2}([\delta H]\{\theta\} + [H]\{\delta\theta\}) \]
\[ = [H_L] \{\delta q\} + [H][G]\{\delta q\} \]
\[ = [B]\{\delta q\} \quad (7) \]

where the matrices \([H_L]\), \([H]\), \(\{\theta\}\), and \([G]\) in equations (5) and (7) are

\[ [H_L] = \begin{bmatrix}
\frac{\partial N_1}{\partial x} & 0 & 0 & \frac{\partial N_1}{\partial y} & 0 & 0 & \cdots & \frac{\partial N_1}{\partial x} & 0 & 0 \\
0 & \frac{\partial N_1}{\partial y} & 0 & 0 & \frac{\partial N_2}{\partial x} & 0 & \cdots & 0 & \frac{\partial N_2}{\partial y} & 0 \\
0 & 0 & \frac{\partial N_2}{\partial z} & 0 & 0 & \frac{\partial N_2}{\partial x} & \cdots & 0 & 0 & \frac{\partial N_2}{\partial z} \\
\cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots \\
0 & 0 & 0 & \frac{\partial N_n}{\partial y} & 0 & 0 & \cdots & 0 & \frac{\partial N_n}{\partial x} & 0 \\
0 & 0 & 0 & 0 & \frac{\partial N_n}{\partial y} & 0 & \cdots & 0 & 0 & \frac{\partial N_n}{\partial x} \\
0 & 0 & 0 & 0 & 0 & \frac{\partial N_n}{\partial z} & \cdots & 0 & 0 & \frac{\partial N_n}{\partial z} \\
0 & 0 & 0 & 0 & 0 & 0 & \cdots & 0 & 0 & 0
\end{bmatrix} \quad (8) \]

\[ [H] = \begin{bmatrix}
a_x^T & 0 & 0 \\
0 & a_y^T & 0 \\
0 & 0 & a_z^T \\
a_y^T & a_x^T & 0 \\
0 & a_z^T & a_y^T \\
0 & 0 & a_x^T
\end{bmatrix} \quad (9) \]
The matrix \([I]\) in equations (10) and (11) is a \((3 \times 3)\) identity matrix and the vector \(\{q\}\) is the vector of nodal displacements. Note that the shape function and displacement vector are dependent on the particular element chosen.

The internal force is computed from the equation of static equilibrium,

\[
F = \int_V [B]^T \{\sigma\} \, dV
\]

and the variation of the internal force is

\[
\delta F = \int_V [B]^T \{\delta \sigma\} \, dV + \int_V [\delta B]^T \{\sigma\} \, dV
\]

Substituting equation (5) and the stress-strain relation,

\[
\{\delta \sigma\} = [D]\{\delta \varepsilon\}
\]

into equation (13) and using the identity,

\[
\]

yields a simple expression for the variation of internal force.

\[
\delta F = \left[ k + k1\{q\} + k2\{q\}\{q\}^T \right] \{\delta q\}
\]

The quantities \(k, k1\{q\},\) and \(k2\{q\}\{q\}^T\) in equation (16) are given as

\[
[k] = \int_V [H_L]^T[D][H_L] \, dV + [k_k]
\]
\[ k_1(q) = \int_{V} ([H_L][D][H][G] + [G]^T[H]^T[D][H_L]) \, dV \]  
\[ k_2(q) = \int_{V} [G]^T[H]^T[D][H][G] \, dV \]  
with  
\[ [k_g] = \int_{V} [G]^T[G] \, dV \]  
\[ [\tau] = \begin{bmatrix} \tau_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \tau_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \tau_z \end{bmatrix} \]  

In equations (17 -- 19), [D] is the material property matrix, [k] is the linear stiffness matrix, \( k_1 \) and \( k_2 \) are the nonlinear stiffnesses, and \([k_g]\) is the geometric stiffness (which depends on the initial stresses).

The element internal force vector \( \{\gamma\} \) is defined as  
\[ \{\gamma\} = [k] + [k_1(q)] + [k_2(q)]^T \{q\} \]  
and the system internal force \( \{\Gamma\} \) is  
\[ \{\Gamma\} = [K] + [K1(Q)] + [K2(Q)]^T \{Q\} \]  
The system mass and damping matrices are obtained using the standard finite element formulation [15].

The equation of motion based on the nonlinear strain-displacement relations is  
\[ [M]\dddot{Q} + [C] \ddot{Q} + [K] + [K1(Q)] + [K2(Q)]^T \{Q\} = \{P\} \]  
or, in more general form, as  
\[ [M]\dddot{Q} + [C] \ddot{Q} + \{\Gamma(Q, Q^2, Q^3)\} = \{P\} \]  
where the matrices [M], [C], and [K] are the system linear mass, damping, and stiffness matrices. The vector \( \{P\} \) is the time dependent load and \( K1 \) and \( K2 \) are the system first- and second-order nonlinear stiffnesses.

Equation (25) has no general solution when the excitation is random. An approximate solution to these equations is obtained by seeking an equivalent linear system [6], of the form  
\[ [M]\dddot{Q} + [C] \ddot{Q} + [K_e] \{Q\} = \{P\} \]  
where \([K_e]\) is an equivalent linear stiffness matrix.
2.2 Equivalent Linear Stiffness Matrix \([K_e]\)

The equivalent linear stiffness matrix \([K_e]\) is to be determined such that the difference between the actual nonlinear system and the approximate linear system is minimized. The approach may be thought of as a statistical version of a classical least square minimization. The error in obtaining the approximate system is defined as

\[
\{\Delta\} = \{\Gamma\} - [K_e]\{Q\}
\]  

(27)

Since the error is a random function of time, the required condition is that the ensemble average or expectation of the mean square error be a minimum. This is expressed as

\[
E[\{\Delta\}\{\Delta\}^T] \rightarrow \text{minimum}
\]

(28)

where \(E[\cdot]\) denotes the expectation operator. As in the cases of classical least square minimization, the necessary condition for satisfying equation (28) is

\[
\frac{\partial E[\{\Delta\}\{\Delta\}^T]}{\partial [K_e]} = 0
\]

(29)

Substituting equation (27) into equation (29), and interchanging the expectation and differentiation operators yields

\[
E[\{\Gamma\}\{Q\}^T] = E[\{Q\}\{Q\}^T][K_e]^T
\]

(30)

Using the fact that the matrix \(E[\{Q\}\{Q\}^T]\) is non-singular, the equivalent linear stiffness matrix \([K_e]\) can be determined from the equation

\[
[K_e] = E[\{Q\}\{Q\}^T]^{-1} E[\{\Gamma\}\{Q\}^T]
\]

(31)

The equivalent linear stiffness \([K_e]\) defined in equation (31) can be directly programmed in a finite element code if the stiffnesses \(K1\) and \(K2\) are available and the expectation operator can be evaluated.

Two assumptions regarding the distribution and dependence of the displacements are necessary in order to evaluate equation (31). The most commonly assumed distribution of the displacements is a Gaussian distribution, since the most commonly encountered random loads are typically modelled by Gaussian distributions. The most commonly assumed dependence between displacement responses is that they are independent. This is simply because, in a linear modal analysis, the modal responses are solved for independently; their modes are uncoupled. These assumptions are not the only possible assumptions; other assumptions can easily be substituted, but would yield more complicated results.

It is generally assumed that the response is Gaussian if the load is Gaussian. By using the formula for the expected value of a Gaussian vector \(\{\eta\}\)

\[
E[f(\eta)] = E\{\eta \eta^T\} E\{\nabla f(\eta)\}
\]

(32)
where \( \nabla \) is the gradient operator, \( E[\Gamma \{Q\}^T] \) on the right hand side of equation (31) becomes

\[
E[\{Q\} \{Q\}^T] E \left[ \frac{\partial \Gamma}{\partial Q} \right]
\]

or

\[
E[\{Q\} \{Q\}^T] E \left[ \frac{\partial \left[ [K] + [K1 \{Q\}] + [K2 \{Q\} \{Q\}^T] \right] \{Q\}}{\partial \{Q\}} \right]
\]

The equivalent linear stiffness matrix \([K_e]\) can then be determined from the equation

\[
[K_e] = E \left[ \frac{\partial \Gamma}{\partial Q} \right] = E \left[ \frac{\partial \left[ [K] + [K1 \{Q\}] + [K2 \{Q\} \{Q\}^T] \right] \{Q\}}{\partial \{Q\}} \right]
\]

where \([K_e]\) is an equivalent linear function of the displacement vector \(\{Q\}\), which is one order less than the nonlinear system stiffness matrix \(\{\Gamma\}\).

The nonlinear stiffnesses are generally formed in tangential or differential form and the expectation operator in equation (35) requires knowledge of the joint probability density function of the response vector, which is the unknown. Therefore, the equivalent linearization solution procedure is programmed in an iterative fashion and some additional assumptions regarding the expectations of the response vector are required. It should be noted that, if \(K1\) and \(K2\) are available, the mean square response can be obtained directly [7] with appropriate assumptions for the expectation operator.

In all instances cited above, assumptions regarding the expectations of the response vector are required. These assumptions are usually based on a knowledge of the excitation and the solution method used. A discussion of the general iterative equivalent linear solution procedures is next presented.

### 2.3 Iterative Equivalent Linearization Solution Methods

There are two basic means to solve linear dynamic equations of motion: one uses the physical degrees of freedom and the other uses the modal degrees of freedom. The first method is generally referred to as the direct frequency response method and requires solving a complex coupled system of equations in the nodal degrees of freedom at each frequency of interest. The second method is generally referred to as the modal frequency response method. It involves solving for the linear eigenvectors first and transforming the equations of motion into modal coordinates. The resulting system of equations is uncoupled and can be easily solved at each frequency of interest.

The primary consideration as to which method to use for a particular linear system is based on the computational time required. This decision in an equivalent linearization solution procedure is further complicated by the iterative nature of the problem and the evaluation of the expectations. The choice of method can either greatly simplify or complicate the process.
The direct method would seem to be the easiest and most straightforward to implement, and the computational time required would be simple to compute. The difficulty in the direct method arises in the assumptions regarding the expectation operator in the expression for the equivalent linear stiffness and the implementation of these assumptions in a general sense. Accurate approximations of the expectation operator require assumptions regarding the full set of four moments (mean, standard deviation, skewness, and kurtosis) of the response vector in nodal degrees of freedom. It should be noted that in physical coordinates, the correlations between all the degrees of freedom are necessary and must be determined.

As a simple example of the direct method, consider a beam of length $L$ with ten nodes and three degrees of freedom, $u$, $w$, and $\theta$, at each node. The evaluation of equation (31) for the equivalent linear stiffness requires the evaluation of the complete set of expectations of all the nodal degrees of freedom to the fourth moments. The equivalent linearization solution relies on determining expressions for the third and fourth moments in terms of the first and second moments. These may be obtained by assuming appropriate probability distributions for the nodal displacements. In the beam example, if the excitation is broadband, Gaussian distributed, and spatially correlated over the beam, it can be assumed that the responses $w$ and $\theta$ are Gaussian and $u$ is Chi-square distributed. From these assumptions, an expression for the equivalent linear stiffness in terms of the first and second moments of the response can be found. However each entry in the $30 \times 30$ equivalent linear stiffness matrix could have a different coefficient representative of the degrees of freedom, correlation coefficients between the degrees of freedom, and the order of the expectations involved. The complexity in using physical degrees of freedom can be deduced from this simple problem when it is noted that it is terms such as the square of the slope and the in-plane displacement that are strongly correlated. This entire process is programmable, but it is not easily done in a general sense. The selection of modal coordinates will be seen to make the evaluation of equation (31) simpler.

The modal solution method of the equivalent linearization procedure is simpler to implement than the direct method because reasonable assumptions regarding the correlation of the modal degrees-of-freedom as well as their joint distribution are possible. This is not to say that the modal approach is without deficiencies or difficulties. To illustrate the advantages and difficulties with the iterative modal solution procedure, the simple beam problem discussed in the direct method is used. The first difficulty arises immediately from the linear eigenvalue problem. The extracted eigenvectors are functions of either the out-of-plane nodal displacement (bending modes) or the in-plane nodal displacement (membrane modes), but not both. This is because the bending motion of the beam is coupled to the membrane motion through the nonlinear terms.

There are three ways to handle the decoupling of the membrane and bending motion induced by the use of the linear eigenvectors. The first way is to simply exclude the membrane modes from the modal response. This is easy, but not particularly accurate. A popular corollary to this solution is used for one- and two-dimensional structures [7]. This procedure assumes the in-plane inertia and damping to be negligible. It is then possible to solve for the membrane modes in terms of the bending modes and thus account for the
in-plane stiffness. This procedure is efficient, but highly specialized and difficult to include in a general finite element code.

The second method involves selecting particular bending modes and membrane modes to include in the formulation. The difficulties that arise from this solution are similar to those encountered in the direct method when trying to evaluate the expectations and solving the system of equations. In the beam problem, it is again assumed that the bending is Gaussian and the membrane is Chi-square distributed when the excitation is Gaussian. The bending modes can be assumed uncorrelated with respect to each other, as can be the membrane modes, but the membrane modes are strongly correlated to the square of the bending modes. The resulting system of equations is coupled and the expression for the equivalent linear stiffness matrix is only marginally simplified with respect to the direct method. Another difficulty with the linear modal solution procedure is that the type of modes, bending, membrane, or otherwise, are not always readily identifiable or available. Many current finite element programs use Lanczos-type eigenvalue solvers in which only the lowest modes or modes within a certain range are computed. It is difficult to construct a general program using this method that will extract the particular eigenvectors needed for an accurate solution.

The third modal solution method for the equivalent linearization procedure uses updated or "equivalent linear" modes. The obvious drawback to this method is that it requires the eigenvalue problem to be solved at each iteration. The advantages of this method are that the system of equations that are solved at each frequency are uncoupled and that simple assumptions regarding the moments and correlation of the modal responses are adequate for accurate solutions. The simple beam problem discussed in the previous solution methods could be solved with only a small number of updated modes. If the load were Gaussian, these modes could be assumed Gaussian-distributed and uncorrelated. Although the means of the equivalent linear modal amplitudes are also assumed to be zero, this does not require that all the nodal displacements comprising the mode shape have zero means. The relationship between the mean of the in-plane displacement, \( u \), and the mean square of the slopes, \( \theta \), in the simple beam problem, is implicitly maintained in the equivalent linear modal approach.

The relationship between the steps involved in the direct, linear modal, and equivalent linear modal approaches to implementing the equivalent linearization solution procedure are outlined in the flowchart in figure 2 for a general finite element program. The solution procedure is iterative as previously discussed, since the nonlinear stiffness is only available in a differential form. The convergence of the iterative procedure is based on the Euclidian norm of the response variance vector. It was determined that the equivalent linear modal method of solving the iterative equivalent linearization procedure would be the simplest and most versatile of the three methods to implement in MSC/NASTRAN.
2.4 Implementation

The equivalent linear stiffness matrix in equation (31) must first be expressed in equivalent linear modal coordinates in order to evaluate the expectation operator. The stiffness vector \( \{\Gamma(Q, Q^2, Q^3)\} \) in equivalent linear modal coordinates has the form \( \{\overline{\Gamma}(A, A^3)\} \), where the bar indicates a quantity transformed in modal coordinates. The expression for the equivalent linear stiffness, equation (35), with the Gaussian, zero mean, and uncorrelated modal response assumptions reduces to

\[
[K_e] = E \left[ \frac{\partial \{\Gamma\}}{\partial \{A\}} \right] \tag{36}
\]

The partial derivatives are easily performed and yield a linear modal stiffness matrix and a differential modal stiffness matrix that is based on the mean square of the modal response [5]. The modal representation of the equivalent linear stiffness is then

\[
[K_e] = [K] + 3[K^2(E[A^2])] \tag{37}
\]
This model expression is not directly programmable in MSC/NASTRAN. It must instead be expressed in physical coordinates, since the eigenvalue problem in MSC/NASTRAN is solved in physical coordinates. In addition, the differential stiffness matrix in MSC/NASTRAN is formed using the physical displacements. The linear stiffness matrix in equation (35) in physical coordinates is simply the linear stiffness matrix as assembled and computed in the MSC/NASTRAN program. The differential stiffness matrix expression in physical coordinate is the MSC/NASTRAN differential stiffness matrix formed using an equivalent linear displacement vector. This equivalent linear displacement vector is given by

\[ \{Q\} = [\Phi] \{\sigma_A\} + [\Phi] \{\mu_o\} \tag{38} \]

where \( \{\sigma_A\} \) is a vector of the standard deviations of the equivalent linear modal amplitudes and \([\Phi]\) is the matrix of normalized eigenvectors. The standard deviation of the modal amplitudes is always positive. The sign convention of the physical displacement is determined by the eigenvectors. The vector \( \{\mu_o\} \) is the mean displacement obtained from a static solution sequence. The matrix of eigenvectors is normalized such that the magnitude of each eigenvector in the matrix is unity. The final expression for the equivalent linear stiffness is then

\[ [K_e] = [K] + 3[K_R] \tag{39} \]

where \([K_R]\) is the standard MSC/NASTRAN differential stiffness matrix.
Section 3 Programmer's Notes

The MSC/NASTRAN version 67 solution sequences are written using a common set of "subroutines" or SUBDMAPs. It is the MAIN SUBDMAPs, "main programs," that vary significantly and usually contain the essence of the solution procedure. The authors attempted to follow this structure in the development of the new Super Element Modal Equivalent Linear Random Response (SEMELRR) solution sequence, but some alterations to the common SUBDMAPs were also necessary. These alterations to the SUBDMAPs, as well as a description of the MAIN SUBDMAP of the SEMELRR solution sequence, are outlined.

All solution sequences are broken down into three general sections. These sections are simply expressed as Phase 1, Phase 2, and Phase 3. The Phase 1 portion of the program is dedicated primarily to the setting-up of the problem and the assembly of the linear matrices. Key portions of these procedures are the reading of the NASTRAN data deck, the restart capability, the creation of the element summary tables, the partitioning of the global degrees of freedom into the various analysis set tables (USET, etc.), and the formation and assembly of the linear elements and their reduction to the analysis set. The Phase 2 procedures are primarily associated with the actual solution of the problem. These solution procedures are, for example, the eigenvalue and eigenvector extraction routine of SOL 103, the linear matrix equation solvers in SOL 101, and the modal matrix formation and complex frequency response solver routines in SOL 111. SOL 106, the nonlinear static solution sequence, has a complicated Phase 2. This Phase 2 involves an iterative solution procedure similar to the solution sequence that was written into the Phase 2 of the SEMELRR solution sequence. Phase 3 procedures are primarily associated with post-processing routines such as data recovery, plotting and printing, and stress/strain calculations. Phase 3 also includes the dynamic sensitivity analysis. The calculation of power spectral densities and root mean square responses for random analysis using SOL 111 and SOL 108 are also included in Phase 3 procedures. The scattered placement of these procedures caused difficulty in the implementation of the equivalent linearization solution procedure.

3.1 SEMELRR Main SUBDMAP

As a starting point from which to write the SEMELRR DMAP, the authors selected the MSC/NASTRAN-delivered SOL 111 main SUBDMAP. This solution sequence is capable of performing linear random analysis. The primary additions to this solution sequence were envisioned to be the incorporation of Phase 2 procedures, similar to those found in SOL 106, for the formation of the nonlinear stiffness matrices and the iterative solution method. It was immediately apparent that the logical flow of the set of MSC/NASTRAN SUBDMAPs and modules did not readily permit simultaneous geometric nonlinearities and dynamics. The SEMELRR Solution sequence would have to be a hybrid-type solution sequence comprised of linear and nonlinear Phases. The calculation of the rms quantities, which usually occurs in Phase 3, and the necessity of having that information available in the iterative procedures required the new solution sequence to have no clear distinction between Phase 2 and Phase 3.
Figure 3: SUBDMAP call tree of SEMELRR solution sequence
The logical flow of MSC/NASTRAN solution sequences is partly controlled by parameters and flags that are set in the main SUBDMAP of the solution sequence. These flags are passed to the standard SUBDMAPs and appear in IF, THEN, ELSE type logical construction. Typical character parameters are the solution type (SOLTYP= "DIRECT," "MODAL," etc.), solution approach (APP= "STATIC," "FREQRESP," "TRANRESP," etc.), and logical flags are (NONLNR, AERO, FS, etc.). The logical flow for a nonlinear dynamic solution sequence does not exist, but by changing these parameters during the solution procedure the necessary logical flow can be generated.

The SEMELRR solution sequence requires that linear and nonlinear element tables be generated in Phase 1 procedures and that linear dynamic data recovery be performed in Phase 3 procedures. In order to generate the necessary matrices and tables for both geometric nonlinear and linear dynamic procedures, the pre-processing sequence Phase 1 was initiated with the NONLNR flag set to TRUE in the call to SUPER1, Figure 3, and the APP (approach) was set to FREQRESP. The reader is referred to the MSC/NASTRAN user's manual [11] for a description of these parameters.

The logical flag NONLNR was set TRUE for Phase 1 only, in the call to SUPER1, and not for Phase 2 or 3, because linear data recovery is required in SUPER3. In addition to this modification, it was required that the element summary table, ESTL, needed for linear dynamic analysis (not generated when NONLNR is TRUE) be equivalenced to the element summary table, EST, for the linear portion of nonlinear analysis (generated when NONLNR is TRUE). This equivalence was programmed as an ALTER to the SEMG SUBDMAP.

The programming of the SEMELRR main SUBDMAP consisted of writing an iterative procedure around the frequency response solution procedures, the geometric nonlinear matrix generation procedures, and the data recovery SUBDMAP, SUPER3, which includes the updated displacement calculations. To implement this iterative procedure, some of the files needed for the next iteration have to be saved. The module FILE to save or overwrite files was used for this purpose. Phase 3 procedures were included in the iteration loop because the updated displacements, necessary as input to the differential stiffness modules, are obtained from SUBDMAP SEDRCVR in Phase 3. SUBDMAP SEDRCVR had to be substantially rewritten to generate the correct updated displacements for the equivalent linearization procedure, equation (38). The calculation of the updated displacements will be discussed in depth in the following subsection. A full listing of the SEMELRR main SUBDMAP is provided in Appendix A.

The formation of the geometric nonlinear stiffness matrix in Phase 2 follows closely with the procedure in Nonlinear Transient solution sequence (NLTRAN, SOL 129). The linear dynamic equations of motion are solved first and the linear rms displacement vector \( \{A\} \) is obtained. If the parameter LGDISP is greater than \(-1\), the geometric nonlinear stiffness matrix KDJJ is formed from module EMA on the next iteration by applying this linear displacement vector. This geometric nonlinear stiffness matrix then reduces to KDLL, \([K_R]\) in equation (39). (If the parameter LGDISP equals \(-1\), only the linear frequency response is calculated.) The equivalent linear stiffness matrix \([K_e]\) now consists of two matrices: the linear stiffness \([K]\) and the differential stiffness matrix \([K_R]\). The frequency response is then obtained using both geometric nonlinear and linear dynamic matrices.
This iteration method can be used to determine the rms displacements; however, it is slow to converge. An improved method for speeding up the convergence is to use an underrelaxation approach where displacements are not updated to their full values, but instead to the scale of the full values after each iteration. This method can be expressed as

\[
\{Q\}_{\text{current}} = (1 - \beta)\{Q\}_{\text{previous}} + \beta\{Q\}_{\text{current}}
\]

A user-defined convergence enhancement parameter, \(\beta\) (BETA in DMAP programming), is introduced to scale the updated displacements. If the nonlinearity is mild to moderate, the convergence of the iteration procedure is faster for \(0.5 \leq \beta \leq 1.0\). If the nonlinearity is severe, the convergence of the iteration procedure is faster for \(0.0 < \beta \leq 0.5\). The parameter BETA is set by the user in the Bulk Data Deck.

Two user-defined parameters were introduced to control the termination of the iterative loop. The user-defined parameter MAXITER defines the maximum allowable number of iterations and the user-defined parameter MAXNORM sets the convergence criteria, i.e.

\[
\left| \{Q\}_{\text{current}} - \{Q\}_{\text{previous}} \right| = \text{error} \leq \text{MAXNORM}
\]

If the iteration count exceeds MAXITER or if the error norm, equation (41), is less than MAXNORM, the solution sequence will terminate normally. There is a warning message if the solution is not converged after the MAXITER iterations. There are two ways to handle convergence errors; the first is by increasing the number of allowable iterations, MAXITER, and the second is by choosing a different convergence enhancement parameter BETA, which is less than the previous BETA. A summary of the user-defined parameters and defaults is given in Appendix B.

### 3.2 Updated Displacement Calculation

The updated displacement vector is formed by multiplying the maximum rms displacement by the updated mode shapes. In order to do so, one deflection point number has to be obtained first by asking for XYPRINT (or XYPLOT) in the Control Deck of the MSC/NASTRAN data cards. In the SUBDMAP SEDRCVR, individual modes of the actual displacement vector are extracted. For each mode, the modal rms responses are calculated from module RANDOM. Each mode is normalized to unity for the largest component of the eigenvector. The actual rms response of each mode is then obtained by multiplying the rms response by the normalized eigenvector. The updated response of the structure can be calculated by using superposition of the modes and storing the updated rms displacement vector. This procedure entails the assumption that the modes and modal responses are independent. The modified SUBDMAP SEDRCVR is in Appendix C.

Although some minor modifications on SUBDMAP SUPER3 are made, no functional procedure was carried out. The modification passes parameters needed for communication between the main SUBDMAP and the SEDRCVR SUBDMAP. The modified SUBDMAP SUPER3 is included in Appendix D.
3.3 Output Requests

One new feature from the output request is for plotting the overall rms displacement output. In module RANDOM, only the rms values for a single degree of freedom are calculated. The actual overall rms displacement is formed by the updated mode shape at each iteration. Therefore, at the converged stage, the overall rms displacement can be extracted by using a DISP=ALL card in the Control Deck.

There is no rms strain response obtained from frequency random analysis of SOL 111. If rms element strain is required, the user-defined parameter RMSTRAIN has to be set to 1. For this case, the STRAIN=ALL card is needed in the Control Deck and the strains will be calculated.
Section 4 Validation of The SEMELRR Solution Sequence

The linearized random vibration capability developed for use with MSC/NASTRAN is validated by solving four problems and comparing the results with known solutions. The frequency response of free vibration and rms displacement response of forced random vibrations of a plate and a beam are considered. The results show that reasonable accuracy is achieved.

Problem 1: Random response of beams

The rms displacements of a 12-in × 2-in × 0.064-in aluminum beam with either end clamped or simply supported and subjected to uniformly distributed random loads is investigated. To demonstrate the accuracy of the SEMELRR results, approximate rms maximum deflections were obtained by using a separate Equivalent Linearization (EL) analysis [16] and finite element (FE) solution [7]. Results are shown in Figure 4. Since all three results (EL, FE, and SEMELRR) based on small deflection linear theory lie directly on top of one another, it is shown as one straight dotted and dashed line. The EL and FE results are identical for the linearized case, so one curve is plotted for these two methods. For acoustic excitations less than 90 dB for a simply supported case and 110 dB for a clamped case, the small deflection assumption yields good results. At high SSL, however, the small deflection theory overestimates the rms deflection, while it underestimates the frequency of vibration. It is clearly demonstrated that the SEMELRR results give reasonable predictions as compared to the EL and FE solutions in both linear and linearized cases.

Figure 4: Effect of acoustic excitation level on maximum deflection for beams.
Problem 2: Random response of a clamped plate

The comparison is made for rms displacements as a function of SSL of an aluminum plate [7]. For acoustic excitations less than 100 dB, the small deflection assumption yields good results as shown in Figure 5. Above 100 dB, the large deflection formulation must be used. At the 130-dB level, the results between the SEMELRR and Locke's analysis [7] show a 6-percent difference. The discrepancy is attributable to the approximation of the nonlinear stiffness matrix in equation (39) and the assumption of curvatures and midsurface strains in the von Karman sense in Locke's formulation.

Figure 5: Effect on acoustic excitation level on maximum deflection for clamped plate.

Problem 3: Free vibration of rectangular plate

The free vibration of a 15-in x 12-in x 0.04-in aluminum rectangular plate reported in Chiang's paper [17] is used. The variation in SEMELRR free vibration results of a plate with all edges clamped for the frequency ratio \( \omega / \omega_0 \) at different amplitudes is shown in Figure 6. \( \omega_0 \) is the fundamental frequency of the clamped plate. There is a maximum of 10-percent difference between the SEMELRR and Chiang's results. The frequency ratio for Chiang's results are lower. The differences are caused by two factors. First, Chiang's formulation used von Karman strain-displacement relations, which use thin plate assumptions, and therefore do not have all the terms in equation (1). The second is due to the approximation in equation (39). Because of this approximation, in which the first-order stiffness matrix in the SEMELRR is calculated one more time than the equivalent linearization approach, the linearized frequency is expected to be higher. These results show the SEMELRR procedure gives reasonable
predictions in comparison to finite element [17, 7] and equivalent linearization [16] solutions Figures, 4, 5, and 6.

![Graph of Amplitude versus Frequency Ratio](image-url)

Figure 6: Amplitude versus frequency ratio for clamped plate.
Section 5 Example Problems

This section is intended to provide a series of simple analyses that demonstrate the capabilities and use of the equivalent linearization solution procedure as implemented in MSC/NASTRAN. The types of analyses used in this section were selected from a review of previously published papers [7, 16]. For simplicity these analyses share a common structural configuration, that of a simple rectangular panel. The thermo-acoustic response of a large hexagonal thermal acoustic protection panel is also presented to further demonstrate some of the features of the program. The format of this section follows closely that of the MSC/NASTRAN demonstration problems manual [18]. It is assumed in this section that the reader has a basic understanding of the basic NASTRAN CARDS and DECKS.

5.1 Problem Execution

The equivalent linearization solution sequence was written by incorporating portions of the NLSTATIC (SOL 106) solution sequence into the SEMFREQ (SOL 111) solution sequence. It is assumed in this manual that the reader has a basic understanding of the application, options, and limitations of both of those analyses.

MSC/NASTRAN performs random response analysis as post-processing to the frequency response. The Equivalent Linearization solution sequence is performed by including this post-processing in the iteration loop because the rms displacements, which are necessary as input to the differential stiffness modules, are obtained in Phase 3 procedure.

The SEMELRR solution sequence is not included in the MSC/NASTRAN-delivered data base, but is available as a separate DMAP program. The program must be read into the Executive Deck of the NASTRAN data file, and the individual SUBDMAPs and main SUBDMAP must be compiled and linked as part of each execution. The solution sequence can also be incorporated into the NASTRAN data base of solution sequences by creating a permanent USER.OBJ and USER.EXE as discussed in Chapter 7 of the "DMAP and DATA BASE APPLICATIONS" seminar notes [19]. The necessary commands to include, compile, link, and execute the solution sequence are provided in the example problems.

Model Description

The basic Bulk Data cards for the rectangular panel will be included in each example but will appear only here. The demonstration Bulk Data Deck includes the CQUAD4, GRID, SPC1, PSHELL, MAT2, and MAT8 cards. The rectangular aluminum plate is 12-in × 15-in × 0.04-in and is modeled using 64 QUAD4 elements with inplane and bending material property entries on the element PSHELL cards. The boundaries are assumed completely clamped. The zero displacements and rotations are enforced using SPC1 cards.
<table>
<thead>
<tr>
<th>GRID</th>
<th>1.875</th>
<th>1.5</th>
<th>0.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>GRID</td>
<td>3.750</td>
<td>1.5</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>5.625</td>
<td>1.5</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>7.500</td>
<td>1.5</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>9.375</td>
<td>1.5</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>11.25</td>
<td>1.5</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>13.125</td>
<td>1.5</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>15.000</td>
<td>1.5</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>0.000</td>
<td>1.5</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>1.875</td>
<td>1.5</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>3.750</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>5.625</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>7.500</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>9.375</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>11.25</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>13.125</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>15.000</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>0.000</td>
<td>1.5</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>1.875</td>
<td>1.5</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>3.750</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>5.625</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>7.500</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>9.375</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>11.25</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>13.125</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>15.000</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>0.000</td>
<td>1.5</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>1.875</td>
<td>1.5</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>3.750</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>5.625</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>7.500</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>9.375</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>11.25</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>13.125</td>
<td>3.0</td>
<td>0.0</td>
</tr>
<tr>
<td>GRID</td>
<td>15.000</td>
<td>3.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

$ 
$ 
$ 

$ SPC1 $ 
$ SPC1 $ 
$ SPC1 $ 
$ SPC1 $ 
$ SPC1 $ 
$ SPC1 $ 

23
5.2 Linear Random Analysis

Problem Description

The random response of the plate subjected to broad-band acoustic excitation is first demonstrated. The spectral density functions of the selected displacements and element stresses are computed.

Executive Control Deck

The Executive Control Deck specifies that Structured Solution Sequence 111 (Modal Frequency Responses) is to be used to analyze the plate response under random loads.

Case Control Deck

<table>
<thead>
<tr>
<th>METHOD</th>
<th>Specifies method by which the eigenvalues and eigenvectors will be extracted.</th>
</tr>
</thead>
<tbody>
<tr>
<td>FREQ</td>
<td>Selects the set of frequencies to be solved in frequency response problems.</td>
</tr>
<tr>
<td>RANDOM</td>
<td>Random Analysis set selection</td>
</tr>
<tr>
<td>LOADSET</td>
<td>Selects a sequence of load sets referenced by dynamic load cards to be applied to the structural model.</td>
</tr>
<tr>
<td>DLOAD</td>
<td>Selects the dynamic load to be applied in a frequency response problem.</td>
</tr>
<tr>
<td>SDMAP</td>
<td>Selects table, which defines dampings as function of frequency.</td>
</tr>
</tbody>
</table>

Figure 7: Demonstration Bulk Data Deck
The rms displacement can only be extracted from the data base PSDF in module RANDOM. The first card after OUTPUT of either XYPRINT DISP or XYPLOT DISP is needed in the Control Deck. If the rms displacement for the first card is zero, the process will stop and the fatal error message will be given. The output from the SEMELRR solution sequence is long, since it prints the output information for each iteration.

Bulk Data Deck

LSEQ Defines a sequence of load sets referenced by dynamic load cards to be applied to the structural model. In this case, it is used to apply a unit pressure load to the plate since the Dynamic Load Scale Factor (DAREA) card can only handle the point load.

DLOAD Defines a dynamic loading condition for frequency response.

RLOAD1 Defines a frequency-dependent dynamic load for use in frequency response problems.

FREQ1 Defines a set of frequencies to be analyzed.

RANDPS Defines load set power spectral density factors for use in random analysis.

TABRND1 Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced by the RANDPS entry.

TABDMPS Defines modal damping as a tabular function of frequency.
Problem Output

$ /path/to/mastran/mastран/mastран/mastран/param
MASTRAN SYSTEM PARAMETER ECHO

$ /path/to/mastran/mastран/mastран/mastран/user
MASTRAN SYSTEM USER ECHO

$ /path/to/mastran/mastран/mastран/mastран/env
MASTRAN ENVIRONMENT ECHO

$ /path/to/mastran/mastран/mastран/mastран/case
MASTRAN EXECUTIVE CONTROL DECK ECHO

$ /path/to/mastran/mastран/mastран/mastран/comp
MASTRAN EXECUTIVE CONTROL DECK ECHO

$ /path/to/mastran/mastран/mastран/mastран/plot
MASTRAN EXECUTIVE CONTROL DECK ECHO

$ /path/to/mastran/mastран/mastран/mastран/report
MASTRAN EXECUTIVE CONTROL DECK ECHO
TOTAL MATRIX ASSEMBLY TIME FOR 44 ELEMENTS IS 0.22 SECONDS.

NAMES OF NODES 2.549697E+05 NODES

EIGENVALUE 1.798402E+01 CYCLES = 1 00000000 NUMBER OF EIGENVALUES BELOW THIS VALUE = 0

ACCEPTED EIGENVALUES
2.549697E+05

EIGENVALUE ANALYSIS SUMMARY (LANCzos ITERATION)

NODE EXTRACITION EIGENVALUE RADIAN CYCLES
1 1 2.549697E+05 5.048942E+00 8 20487292 1.00000000 2.549697E+05

1 TPS RESULTS ANALYSIS MARCH 29, 1993 NEC/MATRIX 7/29/92 PAGE 9
The spectral density of the strains can be obtained by using an ALTER of the modal frequency response solution sequence (SOL 111). This ALTER is placed in the Executive Control Deck and the solution sequence is compiled as shown:

```
* ALTER
  TYPE = PARM.
  LN = OTAPE2
  FILE = PSDF
  PSDFOVERWRITE
  END
```

**Case Control Deck**

With the ALTER in the Executive Control Deck to acquire the strains, only one card needs to be changed. The change is on the STRAIN( FIBER ) card. To obtain the strain, use the XYPLOT STRESS card. Use of the XYPLOT STRAIN card will cause the compiler to produce a fatal error. The other cards are used in the same order as for the linear random analysis.

```
SET 1 = 41
SET 2 = 1.2, 3, 5
DISPLACEMENT = 1
STRAIN(FIBER) = 2
ECHO = PUNCH
TITLE = FLAT PLATE DEMO
SPC = 56
```
5.3 Nonlinear Random Analysis

Strain and Displacement Spectra

If the XYPRINT is used in the Case Control Deck, the output spectra with frequency increment can be found in the *.06 file. The module FREQ1 in the Bulk Data Deck controls the starting frequency, frequency interval, and the number of frequency increments.

Problem Description

The nonlinear random response of the plate subjected to broad-band acoustic excitation is next demonstrated. The spectral density functions of the selected displacements and element stresses are computed.

Executive Control Deck

The Executive Control Deck specifies that the modified DMAP Modal Frequency Responses Structured Solution Sequence is to be compiled, linked, and used to analyze the plate response under random loads.
Case Control Deck

Use the same Case Control Deck as for the linear random analysis.

Bulk Data Deck

Use the same Bulk Data Deck as for the linear random analysis. The following Parameters are needed in the Bulk Data Deck to proceed with the SEMELRR solution sequence.

PARAMeters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LGDISP</td>
<td>If linearized analysis is performed, set LGDISP=1. (default=-1)</td>
</tr>
<tr>
<td>RMSTRAIN</td>
<td>If rms strain is needed and print no output for STRESS=ALL in control deck, set RMSTRAIN=1. If rms strain is needed and print output for STRESS=ALL in control deck, set RMSTRAIN=2.</td>
</tr>
<tr>
<td>MAXITER</td>
<td>Maximum number of iterations. (default=5)</td>
</tr>
<tr>
<td>ABSNORM</td>
<td>Absolute norm for convergence test.</td>
</tr>
<tr>
<td>BETA</td>
<td>Convergence enhancement factor, ranging from 0.0 to 1.0, but not for 0.0. (default=0.5)</td>
</tr>
</tbody>
</table>

Problem Output

```
C A S E  C O N T R O L  D E C K  E C H O

MARCH 19, 1993  NEU/MAVISAN  7/29/92  PAGE 5

1  THE RESULTS ANALYSIS

CASE
   1
   2  DISPLACEMENT = ALL
   3  ECONOMY 5
   4  TITLE = TIP RESULTS ANALYSIS
   5  DPC =  .5
   6  METHOD = 1
   7  SE/AC = .01
   8  INCL = .00
   9  BARO = .5
```
LANCzos PARAMETER VERIFICATION

INITIAL PROBLEM SPECIFICATION

DEGREES OF FREEDOM = 245
MODE NUMBER = 1
PROBLEM TYPE = 1
SHIFTING SCALE = 4.474E-02

AFTER PROBLEM SPECIFICATION CHECKING

NUMBER OF MODES = 1
RIGHT END POINT = 3.94E-004
RIGHT END POINT = 1.00E+000
CENTER FREQUENCY = 4.50E+000
TIME ALLOWED = 1.50E+000

WORKSPACE ALLOCATION

LANCEZ BAND SIZE = 245
MAX. BAND SIZE = 245
MAX. BLOCKS = 245
MAX. WORKSPACE = 44410

NEW SHIFT = 3.94E-004 MODES STILL NEEDED = 1
NEW SHIFT = 3.94E-004 MODES STILL NEEDED = 0

ACCEPTED EIGENVALUES

2.54E+04

END OF LANCzos RUN

NUMBER OF MODES COMPUTED = 1

COMPUTED MODES

1 THE RESULTS ANALYSIS

EIGENVALUE ANALYSIS SUMMARY (LANCzos ITERATION)

BLACK SIZE USED = 2
NUMBER OF DECOMPOSITIONS = 2
NUMBER OF ROUTE FOUND = 1
NUMBER OF SOLVES REQUIRED = 6

TERMINATION MESSAGE: REQUIRED NUMBER OF EIGENVALUES FOUND.
No. ORDER 1 5 0.049637E-05 8.03453E-03 1.000000E+00 3.549637E-05
TYP RESULTS ANALYSIS MARCH 29, 1993 9:18 PM

0

*** USER INFORMATION MESSAGE 5233: UNCOUPLED SOLUTION ALGORITHM USED.
TYP RESULTS ANALYSIS MARCH 29, 1993 9:18 PM

0

... "THIS IS NODE 1..."
"(09-22-92)" DONT..."WAVE..."
"NO. OF COLUMNS: 1"
TYP RESULTS ANALYSIS MARCH 29, 1993 9:18 PM

0

* DATA (09-24-92)...
TYP RESULTS ANALYSIS MARCH 29, 1993 9:18 PM

0

* X - OUTPUT SUMMARY (AUTO OR PDP)

0 PLOT CURVE FRAME RAW MO. POSITIVE SIGN FOR XMAX FOR DMAX FOR XMAX FOR YMAX FOR XMAX FOR XMAX FOR TMAX ALL DATA ALL DATA ALL DATA ALL DATA TIME

0 DENSITY 1 441 4 6.93525E+01 -6.93525E+01 -6.93525E+01 -6.93525E+01

0 MODES: 400

0 XIUaktATION NUMBER: 43 443 1.000000E+00

0 FREQUENCY: 8.028760E+01

0 *** THE WAVE DISS AT POINT 41 IS 6.93525E+01 WITH TOTAL OF 1 MODES.

0 *** LOCALIZATION: 6.93525E+01 LOCALIZATION: 1.460178E+01

0 *** THE WAVE DISS IS 6.93525E+01

0 USER INFORMATION MESSAGE 4100 (OUTPUT END OF DATA SIMULATION ON FORTRAN UNIT 12

0 (MAXIMUM SIZE OF FORTRAN RECORDS WRITTEN = 1 WORKS.)

0 (NUMBER OF FORTRAN RECORDS WRITTEN = 1 RECORDS.)

0 (TOTAL DATA WRITTEN FOR WORKS = 1 WORKS.)

0 TYP RESULTS ANALYSIS MARCH 29, 1993 9:18 PM

0

0 LAMCIO PARAMETER VERIFICATION

0 INITIAL PROBLEM SPECIFICATION

0 DEGREES OF FREEDOM: 245 MESSAGE LEVEL: 1 LEFT END POINT: 3.9480E-001

0 NUMBER OF NODES: 1 OUTPUT UNIT: 6 RIGHT END POINT: 1.0000E+143

0 PROBLEM TYPE: 1 SIZE OF WORKSPACE: 229.2487 CENTER FREQUENCY: 0.0000E+00

0 SHIFTING SCALE: 6.0172E+05 ACCURACY REQUIRED: 3.9400E+10

0 AFTER PROBLEM SPECIFICATION CHECKING

0 NUMBER OF NODES: 1 LEFT END POINT: 3.9480E-001

0 PROBLEM TYPE: 1 RIGHT END POINT: 1.0000E+143

0 SHIFTING SCALE: 6.0172E+05 CENTER FREQUENCY: 0.0000E+00 CP TIME ALLOWED: 1.7796E+03

0 WORKSPACE ALLOCATION

0 LAMCIO BLOCK SIZE: 2 MAX. BRT VALUES: 200 MAX. TRICT REGIONS: 15

0 MAX. BLOCK SIZE: 100 MAX. S.O. VECTORS: 245

0 MAX. NODES: 245 WORKSPACE ALLOC: 44159

0 NUMBER OF USER SUPPLIED VECTORS: 0

0 NEW SHIFT: 3.9476E+01 MORES STILL NEEDED: 1

0 USER INFORMATION MESSAGE 3101: STORE SEQUENCE DATA FOR EIGENVALUE EXTRACTION.

0 TRAIL EIGENVALUES = 3.9476E+01 CYCLES: 1.000000E+01 NUMBER OF EIGENVALUES BELOW THIS VALUE: 0

0 ACCEPTED EIGENVALUES 2.196348E+07 3.1974E+07

0 NEW SHIFT: 3.1974E+07 MORES STILL NEEDED: 1

0 USER INFORMATION MESSAGE 6138: STATISTICS FOR SYMMETRIC DECOMPOSITION OF DATA BLOCK SCRATCH FOLLOWING NUMBER OF NEGATIVE TERMS ON FACTOR DIAGNOS.

0 USER INFORMATION MESSAGE 3101: STORE SEQUENCE DATA FOR EIGENVALUE EXTRACTION.

0 TRAIL EIGENVALUES = 2.196348E+07 CYCLES: 6.3186413E+02 NUMBER OF EIGENVALUES BELOW THIS VALUE: 1

0 END OF LAMCIO RUN

0 WARNING FLAG: 0 NO. OF NODES COMPUTED: 1

0 COMPUTED NODES 2 1.269596E+02 7.5738E-07

0 TYP RESULTS ANALYSIS MARCH 29, 1993 9:18 PM

0

0 EIGENVALUE ANALYSIS SUMMARY (LAMCIO ITERATION)

0 BLOCK SIZE USED: 2

0 NUMBER OF DECOMPOSITIONS: 2

0 NUMBER OF NODES FOUND: 1

0 NUMBER OF SOLUTIONS REQUIRED: 0

0 TERMINATION MESSAGE: REQUIRED NUMBER OF EIGENVALUES FOUND.

0 TYP RESULTS ANALYSIS MARCH 29, 1993 9:18 PM

0

0 REAL EIGENVALUES

0 MODE EXTRACRION EIGENVALUE RADIAN CYCLES GENERALIZED GENERALIZED

0 NO. ORDER 1 1 1.269596E+07 3.48691E+03 7.56891E+03 1.000000E+00 1.196348E+07

0 TYP RESULTS ANALYSIS MARCH 29, 1993 9:18 PM

0

0
**EIGENVALUE ANALYSIS**

**THE RESULTS ANALYSIS**

**PHASE: CONVERGENCE**

- **NO. OF CYCLES:** 1
- **MAX. BLOCK SIZE:** 2
- **MAX. BLOCK STEPS:** 100
- **MAX. NORM. N:** 1
- **ERROR AND RESIDUALS:**
  - **MAX. ERROR:** 2.65
  - **MAX. RESIDUAL:** 1.16

**THE RESULTS ANALYSIS**

**PHASE: REFLECTION**

- **NO. OF CYCLES:** 1
- **MAX. BLOCK SIZE:** 2
- **MAX. BLOCK STEPS:** 100
- **MAX. NORM. N:** 1
- **ERROR AND RESIDUALS:**
  - **MAX. ERROR:** 2.65
  - **MAX. RESIDUAL:** 1.16

**THE RESULTS ANALYSIS**

**PHASE: SOLUTION**

- **NO. OF CYCLES:** 1
- **MAX. BLOCK SIZE:** 2
- **MAX. BLOCK STEPS:** 100
- **MAX. NORM. N:** 1
- **ERROR AND RESIDUALS:**
  - **MAX. ERROR:** 2.65
  - **MAX. RESIDUAL:** 1.16

**THE RESULTS ANALYSIS**

**PHASE: OUTPUT SUMMARY**

- **NO. OF CYCLES:** 1
- **MAX. BLOCK SIZE:** 2
- **MAX. BLOCK STEPS:** 100
- **MAX. NORM. N:** 1
- **ERROR AND RESIDUALS:**
  - **MAX. ERROR:** 2.65
  - **MAX. RESIDUAL:** 1.16

**THE RESULTS ANALYSIS**

**PHASE: TOLERANCE**

- **NO. OF CYCLES:** 1
- **MAX. BLOCK SIZE:** 2
- **MAX. BLOCK STEPS:** 100
- **MAX. NORM. N:** 1
- **ERROR AND RESIDUALS:**
  - **MAX. ERROR:** 2.65
  - **MAX. RESIDUAL:** 1.16

**THE RESULTS ANALYSIS**

**PHASE: SUMMARY**

- **NO. OF CYCLES:** 1
- **MAX. BLOCK SIZE:** 2
- **MAX. BLOCK STEPS:** 100
- **MAX. NORM. N:** 1
- **ERROR AND RESIDUALS:**
  - **MAX. ERROR:** 2.65
  - **MAX. RESIDUAL:** 1.16

**THE RESULTS ANALYSIS**

**PHASE: SOLVER**

- **NO. OF CYCLES:** 1
- **MAX. BLOCK SIZE:** 2
- **MAX. BLOCK STEPS:** 100
- **MAX. NORM. N:** 1
- **ERROR AND RESIDUALS:**
  - **MAX. ERROR:** 2.65
  - **MAX. RESIDUAL:** 1.16
<table>
<thead>
<tr>
<th>EIGENVALUE ANALYSIS SUMMARY (Lanczos Iteration)</th>
</tr>
</thead>
<tbody>
<tr>
<td>BLOCK SIZE USED: 2</td>
</tr>
<tr>
<td>NUMBER OF DECOMPOSITIONS: 2</td>
</tr>
<tr>
<td>NUMBER OF ROOTS FOUND: 1</td>
</tr>
<tr>
<td>NUMBER OF SOLVES REQUIRED: 6</td>
</tr>
<tr>
<td>TERMINATION MESSAGE: REQUIRED NUMBER OF EIGENVALUES FOUND.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>REAL EIGENVALUES</th>
<th>RADIANS</th>
<th>CYCLES</th>
<th>GENERALIZED</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.00000000</td>
<td>1.00000000</td>
<td>3.00000000</td>
<td>3.00000000</td>
</tr>
<tr>
<td>1.00000000</td>
<td>1.00000000</td>
<td>3.00000000</td>
<td>3.00000000</td>
</tr>
</tbody>
</table>

1. TPV RESULTS ANALYSIS
<table>
<thead>
<tr>
<th>Time/Iteration</th>
<th>TOLERANCE</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>March 29, 1993</td>
<td>3.40132E-02</td>
<td>NUMBER OF EIGENVALUES BELOW THIS VALUE = 0</td>
</tr>
<tr>
<td>March 29, 1993</td>
<td>3.40132E-02</td>
<td>NUMBER OF EIGENVALUES BELOW THIS VALUE = 5</td>
</tr>
</tbody>
</table>

**Eigenvalue Analysis Summary (Lanczos Iteration)**

- **Block Size Used**: 2
- **Number of Decompositions**: 2
- **Number of Blocks Found**: 1
- **Number of Solves Required**: 0

**Summary of Results**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Order</th>
<th>Eigenvalue</th>
<th>Number of Cycles</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1</td>
<td>3.76176E+06</td>
<td>1</td>
</tr>
</tbody>
</table>

**Real Eigenvalues (Lanczos Iteration)**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Order</th>
<th>Eigenvalue</th>
<th>Main Block</th>
<th>Main Eigenvector</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1</td>
<td>3.76176E+06</td>
<td>1.91943E+03</td>
<td>1.00000E+00</td>
</tr>
</tbody>
</table>

**Conclusion**

The Lanczos iteration was successful in finding the required number of eigenvalues below the given tolerance. The final eigenvalues and eigenvectors were computed and are presented above.
Overall Rms Displacements

When the overall rms displacements are requested, DISP=ALL is required in the Control Deck. The overall rms displacement is formed by multiplying the maximum rms displacement by the updated mode shape. The overall rms displacement can be found in the *F06 output file. It also can be extracted from an OUTPUT2 file by including a "PARAM,POST,-1" card in either the Case Control or Bulk Data Decks.

5.4 Static and Nonlinear Random Analysis

The modified MSC/NASTRAN SEMELRR solution sequence can be combined with other solution sequences to handle the effect of static mechanical and thermal loads on the nonlinear random response. An example is shown for a hexagonal thermal protection system panel subjected to combined thermal and random acoustic loads. The deformed shape due to the static thermal load is first obtained using the SOL 101 procedure. A RESTART of the SEMELRR solution sequence is then run to obtain the dynamic response due to the combined load. The results shown were obtained by post-processing the output with PDA Patran [20].
SOL 101 data cards

In order to run SEMELRR with the data base stored from SOL 101, the following command is needed:
nastran P111.dat dbs=P101

SEMELRR data cards
Hexagonal Panel

A hexagonal thermal protection system (TPS) panel similar to the cutout shown in figure 1 was subjected to both thermal and acoustic loads. The structure is composed of an eight-ply carbon-carbon TPS panel with built-up substructure. The TPS panel is connected to the substructure with seven titanium rods (posts). The substructure has an aluminum core sandwiched between an aluminum and a graphite/epoxy face sheet. The dimensions of the panel are given in Table 1, and the finite element mesh is shown in figure 9. The finite element model is comprised of 804 triangular elements and seven bar elements with a total of 622 nodes.

The boundary conditions imposed on the panel were designed to minimize thermal stresses, and are summarized for each component. The edges of the TPS panel are constrained in the perpendicular and tangential directions. The edges of the substructure are constrained in all rotations and translations. The post connections to the TPS panel were modeled as pinned joints using MPC Bulk Data cards. The three translations at the top of the posts were equivalent to the three translations at the adjoining locations of the TPS panel. The connections between the posts and the substructure were also modeled using MPCs. The center post connection was modeled as a rigid link, i.e. all three translations and the two rotations at the lower-end node of the post were equivalent to the translation and rotations of the adjoining node of the substructure. The remaining post connections to the substructure were also modeled as pin joints.

Table 1 Dimensions for hexagonal TPS panel example problem

<table>
<thead>
<tr>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radius</td>
<td>13.0 in.</td>
</tr>
<tr>
<td>Overall height</td>
<td>2.5 in.</td>
</tr>
<tr>
<td>Radius to posts</td>
<td>8.0 in.</td>
</tr>
<tr>
<td>Carbon-carbon thickness</td>
<td>0.091 in.</td>
</tr>
<tr>
<td>Substructure thickness</td>
<td>0.375 in.</td>
</tr>
</tbody>
</table>

39
A 2000 °F temperature load was applied to the TPS panel and a 200 °F load was applied to the posts and substructure. The thermal displacements and stresses were predicted using SOL 101, and are plotted in figures 10 and 11. The TPS panel results are essentially those of a stress-free thermal expansion while the substructure shows a moderate compressive thermal stress with little thermal displacement. The equivalent linearization solution sequence was restarted using the data base from the static thermal solution with the initial stresses and displacements. The rms thermal-acoustic displacements and stresses were predicted for a broadband acoustic excitation of 150 dB uniformly distributed over the carbon-carbon panel. These rms displacements and stresses are plotted in figures 12 and 13. The solution sequence converged in four iterations with the convergence enhancement parameter BETA set to 0.5 and the default convergence criteria.

The level of nonlinearity in the response is typically measured in several ways. The two most common are the ratio of the equivalent linear fundamental frequency to the linear fundamental frequency (frequency ratio) and the ratio of the equivalent linear maximum rms displacement to the linear maximum displacement (amplitude ratio). For this particular problem, these ratios were 1.19 and 0.414, respectively, and are typical of moderate to extreme geometric nonlinearity.

<table>
<thead>
<tr>
<th>Center post radius</th>
<th>Outer post radii</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1875 in.</td>
<td>0.125 in.</td>
</tr>
</tbody>
</table>

Figure 9: Finite element model
Figure 10: Deformed plot of the thermal displacement vector. Displacements are given in inches.

Figure 11: Thermal stresses in the radial direction $\epsilon_r$ in psi.
Figure 12: Deformed plot of the root-mean-square thermal-acoustic displacement vector. Displacements are given in inches.

Figure 13: Root-mean-square thermal-acoustic stresses in the radial direction $e_r$ in psi.
Summary

An equivalent linearization solution sequence used to predict the nonlinear random response of structures has been incorporated into MSC/NASTRAN version 67r2. A new main SUBDMAP, SEMELRR, and a significantly modified MSC/NASTRAN SUBDMAP SEDRCVR are compiled with the MSC/NASTRAN delivered library of SUBDMAPs to create the new solution sequence.

The equivalent linear rms displacements, strains or stresses, and frequencies are calculated by an iterative solution method. The numerical results obtained were in good agreement with existing solutions. The output requests and the iterative solution method are controlled by several new user defined PARAMeters. The versatility of the implementation will enable the analyst to determine the nonlinear random responses for complex structures under combined loads.
Acknowledgment

The work presented in this report was performed under the NASP Government Work Package No. 70 (Acoustics and Sonic Fatigue). Dr. Stephen A. Rizzi was the principal investigator for the NASA Langley Research Center portion of the work package and Mr. Kenneth R. Wentz was the principal investigator at Wright Laboratories. Wright Laboratories was the lead center for GWP No. 70.
References


I’ll just give you a brief overview of the code. It appears to be a subroutine for solving a system of linear equations. The subroutine is named `SUBROUTINE 413` and is defined in the IBM 360/370 System Library. The code uses subscripted variables and includes parameter lists and control statements.

The subroutine starts by initializing variables and setting up the working area. It then proceeds to calculate the determinant of the matrix using the `DET` subroutine. The determinant is used to determine if the system of equations has a unique solution. If the determinant is zero, the system is singular and has no solution.

The subroutine then proceeds to solve the system of equations using the `SUBROUTINE 803` for Gaussian elimination. The solution is stored in the `X` array. The subroutine also prints out the solution at the end.

The code includes comments in both English and French, indicating that it is intended for use in the IBM 360/370 System Library. The code is written in Fortran, a high-level programming language used for scientific and engineering applications.

Overall, the code appears to be well-documented and structured, making it easy to follow and understand. It is a good example of how to solve a system of linear equations using numerical methods.
# Appendix B Additional User Defined Parameters

<table>
<thead>
<tr>
<th>PARAMETER NAME</th>
<th>DEFAULT</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABSNORM</td>
<td>0.05</td>
<td>Absolute norm</td>
</tr>
<tr>
<td>BETA</td>
<td>0.5</td>
<td>Specifies control factor for converge enhancement. Ranges from 0.0 to 1.0</td>
</tr>
<tr>
<td>KNT</td>
<td>-1</td>
<td>Sets iteration counter</td>
</tr>
<tr>
<td>LGDISP</td>
<td>-1</td>
<td>Selects large displacement effects</td>
</tr>
<tr>
<td>LMODES</td>
<td>1</td>
<td>Requests the number of modes (uses with EIGRL card selection)</td>
</tr>
<tr>
<td>MAXITER</td>
<td>5</td>
<td>Requests the maximum number of iterations</td>
</tr>
<tr>
<td>MAXNORM</td>
<td>1.0E-3</td>
<td>Defines converged rms displacement norm</td>
</tr>
<tr>
<td>RMSTRAIN</td>
<td>-1</td>
<td>Requests the rms strains</td>
</tr>
<tr>
<td>XNORM</td>
<td>1.0E-3</td>
<td>Sets norm of rms displacements</td>
</tr>
</tbody>
</table>
Nonlinear Random Response Prediction Using MSC/NASTRAN

An equivalent linearization technique has been incorporated into MSC/NASTRAN to predict the nonlinear random response of structures by means of Direct Matrix Abstract Programming (DMAP) modifications and inclusion of the nonlinear differential stiffness module inside the iteration loop. An iterative process was used to determine the rms displacements. Numerical results obtained for validation on simple plates and beams are in good agreement with existing solutions in both the linear and linearized regions. The versatility of the implementation will enable the analyst to determine the nonlinear random responses for complex structures under combined loads. The thermo-acoustic response of a hexagonal thermal protection system panel is used to highlight some of the features of the program.