NX Nastran

Basic Dynamic Analysis User's Guide

Proprietary & Restricted Rights Notice

TAUCS Copyright and License

Contents

Contents

Contents

About this Book

This guide describes and gives examples of the basic types of dynamic analysis capabilities available in NX Nastran, including

- normal modes analysis

- transient response analysis

- frequency response analysis

- enforced motion

This guide only presents theoretical derivations of the mathematics used in dynamic analysis as they pertain to the proper understanding of the use of each capability.

For more information on dynamic reduction, response spectrum analysis, random response analysis, complex eigenvalue analysis, nonlinear analysis, control systems, fluid-structure coupling and the Lagrange Multiplier Method, see the *NX Nastran Advanced Dynamic Analysis User's Guide*.

Chapter

1   *Fundamentals of Dynamic Analysis*

- *Overview*

- *Equations of Motion*

- *Dynamic Analysis Process*

- *Dynamic Analysis Types*

1.1   Overview

With static structural analysis, it is possible to describe how to use NX Nastran without including a detailed discussion of the fundamental equations. However, because there are several types of dynamic analyses, each with a different mathematical form, you must have some knowledge of both the physics of dynamics and the manner in which the physics is represented to use NX Nastran efficiently for dynamic analysis.

This chapter:

* contains important information on notation and terminology used throughout the rest of the book

* introduces the equations of motion for a single degree-of-freedom dynamic system (see Equations of Motion)

* illustrates the dynamic analysis process (see "Dynamic Analysis Process")

* characterizes the types of dynamic analyses described in this guide (see "Dynamic Analysis Types".

Note: See "References and Bibliography" for a list of references for structural dynamic analysis.

Dynamic Analysis Versus Static Analysis

Two basic aspects of dynamic analysis differ from static analysis:

* Dynamic loads are applied as a function of time.

* This time-varying load application induces time-varying response (displacements, velocities, accelerations, forces, and stresses). These time-varying characteristics make dynamic analysis more complicated and more realistic than static analysis.

1.2   Equations of Motion

The basic types of motion in a dynamic system are displacement u and the first and second derivatives of displacement with respect to time. These derivatives are velocity and acceleration, respectively, given below:

$$\dot{u} \; = \; \frac{du}{dt} \; = \; v \; = \; \text{velocity}$$

$$\ddot{u} \; = \; \frac{d^2u}{dt^2} \; = \; \frac{dv}{dt} \; = \; a \; = \; \text{acceleration}$$

Equation 1-1.

Velocity and Acceleration

Velocity is the rate of change in the displacement with respect to time. Velocity can also be described as the slope of the displacement curve. Similarly, acceleration is the rate of change of the velocity with respect to time, or the slope of the velocity curve.

Single Degree-of-Freedom System

The most simple representation of a dynamic system is a single degree-of-freedom (SDOF) system (see Figure 1-1). In an SDOF system, the time-varying displacement of the structure is defined by one component of motion $u(t)$. Velocity $\dot{u}(t)$ and acceleration $\ddot{u}(t)$ are derived from the displacement.

m	=	mass (inertia)
b	=	damping (energy dissipation)
k	=	stiffness (restoring force)
p	=	applied force
u	=	displacement of mass
$\dot{u}$	=	velocity of mass
$\ddot{u}$	=	acceleration of mass
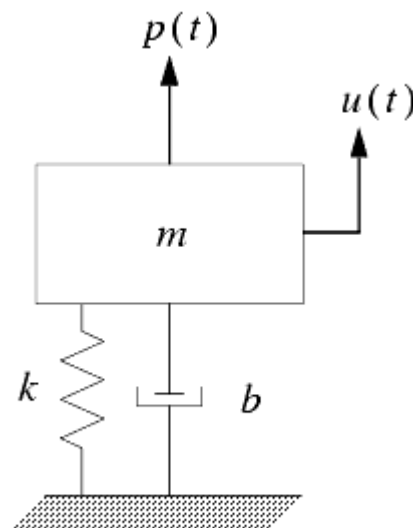
Figure 1-1. Single Degree-of-Freedom (SDOF) System

Dynamic and Static Degrees-of-Freedom

Mass and damping are associated with the motion of a dynamic system. Degrees-of-freedom with mass or damping are often called dynamic degrees-of-freedom; degrees-of-freedom with stiffness are called static degrees-of-freedom. It is possible (and often desirable) in models of complex systems to have fewer dynamic degrees-of-freedom than static degrees-of-freedom.

The four basic components of a dynamic system are mass, energy dissipation (damper), resistance (spring), and applied load. As the structure moves in response to an applied load, forces are induced that are a function of both the applied load and the motion in the individual components. The equilibrium equation representing the dynamic motion of the system is known as the equation of motion.

Equation of Motion

This equation, which defines the equilibrium condition of the system at each point in time, is represented as

$$m\ddot{u}(t) + b\dot{u}(t) + ku(t) = p(t)$$

Equation 1-2.

The equation of motion accounts for the forces acting on the structure at each instant in time. Typically, these forces are separated into internal forces and external forces. Internal forces are found on the left-hand side of the equation, and external forces are specified on the right-hand side. The resulting equation is a second-order linear differential equation representing the motion of the system as a function of displacement and higher-order derivatives of the displacement.

Inertia Force

An accelerated mass induces a force that is proportional to the mass and the acceleration. This force is called the inertia force $m\ddot{u}(t)$.

Viscous Damping

The energy dissipation mechanism induces a force that is a function of a dissipation constant and the velocity. This force is known as the viscous damping force $b\dot{u}(t)$. The damping force transforms the kinetic energy into another form of energy, typically heat, which tends to reduce the vibration.

Elastic Force

The final induced force in the dynamic system is due to the elastic resistance in the system and is a function of the displacement and stiffness of the system. This force is called the elastic force or occasionally the spring force $ku(t)$.

Applied Load

The applied load $p(t)$ on the right-hand side of Equation 1-2 is defined as a function of time. This load is independent of the structure to which it is applied (e.g., an earthquake is the same earthquake whether it is applied to a house, office building, or bridge), yet its effect on different structures can be very different.

Solution of the Equation of Motion

The solution of the equation of motion for quantities such as displacements, velocities, accelerations, and/or stresses—all as a function of time—is the objective of a dynamic analysis. The primary task for the dynamic analyst is to determine the type of analysis to be performed. The nature of the dynamic analysis in many cases governs the choice of the appropriate mathematical approach. The extent of the information required from a dynamic analysis also dictates the necessary solution approach and steps.

Dynamic analysis can be divided into two basic classifications: free vibrations and forced vibrations. Free vibration analysis is used to determine the basic dynamic characteristics of the system with the right-hand side of Equation 1-2 set to zero (i.e., no applied load). If damping is neglected, the solution is known as undamped free vibration analysis.

Free Vibration Analysis

In undamped free vibration analysis, the SDOF equation of motion reduces to

$$m\ddot{u}(t) + ku(t) = 0$$

Equation 1-3.

Equation1-3 has a solution of the form

$$u(t) = A \sin \omega_n t + B \cos \omega_n t$$

Equation 1-4.

The quantity $u\ (t)$ is the solution for the displacement as a function of time t. As shown in Equation 1-4, the response is cyclic in nature.

Circular Natural Frequency

One property of the system is termed the circular natural frequency of the structure w_n . The subscript n indicates the "natural" for the SDOF system. In systems having more than one mass degree of freedom and more than one natural frequency, the subscript may indicate a frequency number. For an SDOF system, the circular natural frequency is given by

$$\omega_n = \sqrt{\frac{k}{m}}$$

Equation 1-5.

The circular natural frequency is specified in units of radians per unit time.

Natural Frequency

The natural frequency f_n is defined by

$$f_n = \frac{\omega_n}{2\pi}$$

Equation 1-6.

The natural frequency is often specified in terms of cycles per unit time, commonly cycles per second (cps), which is more commonly known as Hertz (Hz). This characteristic indicates the number of sine or cosine response waves that occur in a given time period (typically one second).

The reciprocal of the natural frequency is termed the period of response T_n given by

$$T_n = \frac{1}{f_n} = \frac{2\pi}{\omega_n}$$

Equation 1-7.

The period of the response defines the length of time needed to complete one full cycle of response.

In the solution of Equation 1-4, A and B are the integration constants. These constants are determined by considering the initial conditions in the system. Since the initial displacement of the system u $(t = 0)$ and the initial velocity of the system $\dot{u}(t = 0)$ are known, A and B are evaluated by substituting their values into the solution of the equation for displacement and its first derivative (velocity), resulting in

$$B = u(t = 0) \quad \text{and} \quad A = \frac{\dot{u}(t = 0)}{\omega_n}$$

Equation 1-8.

These initial value constants are substituted into the solution, resulting in

$$u(t) = \frac{\dot{u}(0)}{\omega_n}\sin\omega_n t + u(0)\cos\omega_n t$$

Equation 1-9.

Equation 1-9 is the solution for the free vibration of an undamped SDOF system as a function of its initial displacement and velocity. Graphically, the response of an undamped SDOF system is a sinusoidal wave whose position in time is determined by its initial displacement and velocity as shown in Figure 1-2.
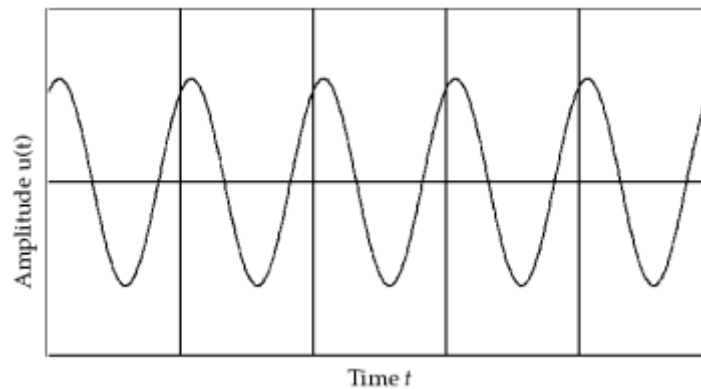
Figure 1-2. SDOF System – Undamped Free Vibrations

If damping is included, the damped free vibration problem is solved. If viscous damping is assumed, the equation of motion becomes

$$m\ddot{u}(t) + b\dot{u}(t) + ku(t) = 0$$

Equation 1-10.

Damping Types

The solution form in this case is more involved because the amount of damping determines the form of the solution. The three possible cases for positive values of b are

- Critically damped

- Overdamped

- Underdamped

Critical damping occurs when the value of damping is equal to a term called critical damping b_{cr}. The critical damping is defined as

$$b_{cr} = 2\sqrt{km} = 2m\omega_n$$

Equation 1-11.

For the critically damped case, the solution becomes

$$u(t) = (A + Bt)e^{-bt/2m}$$

Equation 1-12.

Under this condition, the system returns to rest following an exponential decay curve with no oscillation.

A system is overdamped when $b > b_{cr}$ and no oscillatory motion occurs as the structure returns to its undisplaced position.

Underdamped System

The most common damping case is the underdamped case where $b < b_{cr}$. In this case, the solution has the form

$$u(t) = e^{-bt/2m}(A\sin\omega_d t + B\cos\omega_d t)$$

Equation 1-13.

Again, A and B are the constants of integration based on the initial conditions of the system. The new term ω_d represents the damped circular natural frequency of the system. This term is related to the undamped circular natural frequency by the following expression:

$$\omega_d = \omega_n \sqrt{1 - \zeta^2}$$

Equation 1-14.

The term z is called the damping ratio and is defined by

$$\zeta = \frac{b}{b_{cr}}$$

Equation 1-15.

The damping ratio is commonly used to specify the amount of damping as a percentage of the critical damping.

In the underdamped case, the amplitude of the vibration reduces from one cycle to the next following an exponentially decaying envelope. This behavior is shown in Figure 1-3. The amplitude change from one cycle to the next is a direct function of the damping. Vibration is more quickly dissipated in systems with more damping.
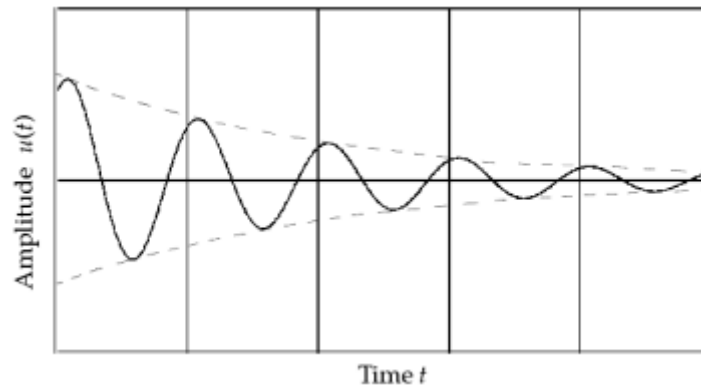
Figure 1-3. Damped Oscillation, Free Vibration

The damping discussion may indicate that all structures with damping require damped free vibration analysis. In fact, most structures have critical damping values in the 0 to 10% range, with values of 1 to 5% as the typical range. If you assume 10% critical damping, Equation 1-4 indicates that the damped and undamped natural frequencies are nearly identical. This result is significant because it avoids the computation of damped natural frequencies, which can involve a considerable computational effort for most practical problems. Therefore, solutions for undamped natural frequencies are most commonly used to determine the dynamic characteristics of the system (see "Real Eigenvalue Analysis"). However, this does not imply that damping is neglected in dynamic response analysis. Damping can be included in other phases of the analysis as presented later for frequency and transient response (see Frequency Response Analysis and "Transient Response Analysis").

Forced Vibration Analysis

Forced vibration analysis considers the effect of an applied load on the response of the system. Forced vibrations analyses can be damped or undamped. Since most structures exhibit damping, damped forced vibration problems are the most common analysis types.

The type of dynamic loading determines the mathematical solution approach. From a numerical viewpoint, the simplest loading is simple harmonic (sinusoidal) loading. In the undamped form, the equation of motion becomes

$$m\ddot{u}(t) + ku(t) = p\sin\omega t$$

Equation 1-16.

In this equation the circular frequency of the applied loading is denoted by w. This loading frequency is entirely independent of the structural natural frequency w_n, although similar notation is used.

This equation of motion is solved to obtain

$$u(t) = \underbrace{A\sin\omega_n t + B\cos\omega_n t}_{\text{Initial Condition Solution}} + \underbrace{\frac{p/k}{1 - \omega^2/\omega_n^2}\sin\omega t}_{\text{Steady-State Solution}}$$

Equation 1-17.

where:

$$A = \frac{\dot{u}(t = 0)}{\omega_n} - \frac{\omega p/k}{(1 - \omega^2/\omega_n^2)\omega_n}$$

$$B = u(t = 0)$$

Again, *A* and *B* are the constants of integration based on the initial conditions. The third term in Equation 1-17 is the steady-state solution. This portion of the solution is a function of the applied loading and the ratio of the frequency of the applied loading to the natural frequency of the structure.

The numerator and denominator of the third term demonstrate the importance of the relationship of the structural characteristics to the response. The numerator p/k is the static displacement of the system. In other words, if the amplitude of the sinusoidal loading is applied as a static load, the resulting static displacement u is p/k . In addition, to obtain the steady state solution, the static displacement is scaled by the denominator.

The denominator of the steady-state solution contains the ratio between the applied loading frequency and the natural frequency of the structure.

Dynamic Amplification Factor for No Damping

The term

$$\frac{1}{1 - \omega^2 / \omega_n^2}$$

is called the dynamic amplification (load) factor. This term scales the static response to create an amplitude for the steady state component of response. The response occurs at the same frequency as the loading and in phase with the load (i.e., the peak displacement occurs at the time of peak loading). As the applied loading frequency becomes approximately equal to the structural natural frequency, the ratio w/w_n approaches unity and the denominator goes to zero. Numerically, this condition results in an infinite (or undefined) dynamic amplification factor. Physically, as this condition is reached, the dynamic response is strongly amplified relative to the static response. This condition is known as resonance. The resonant buildup of response is shown in Figure 1-4.
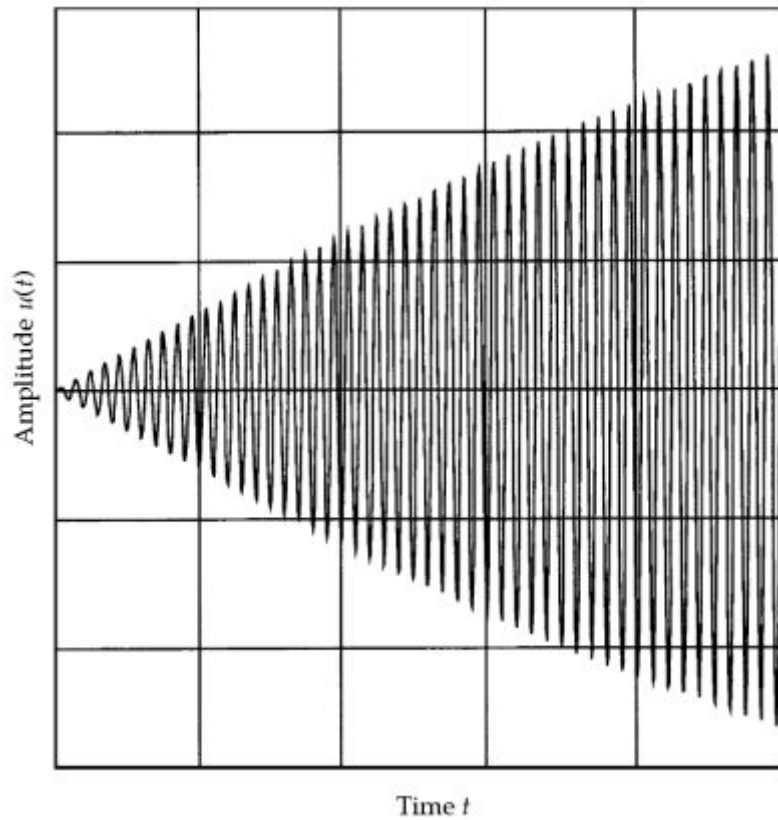
Figure 1-4. Harmonic Forced Response with No Damping

It is important to remember that resonant response is a function of the natural frequency and the loading frequency. Resonant response can damage and even destroy structures. The dynamic analyst is typically assigned the responsibility to ensure that a resonance condition is controlled or does not occur.

Solving the same basic harmonically loaded system with damping makes the numerical solution more complicated but limits resonant behavior. With damping, the equation of motion becomes

$$m\ddot{u}(t) + b\dot{u}(t) + ku(t) \; = \; p \sin \omega t$$

Equation 1-18.

In this case, the effect of the initial conditions decays rapidly and may be ignored in the solution. The solution for the steady-state response is

$$u(t) = p/k \frac{\sin(\omega t + \theta)}{\sqrt{(1 - \omega^2/\omega_n^2)^2 + (2\zeta\omega/\omega_n)^2}}$$

Equation 1-19.

The numerator of the above solution contains a term that represents the phasing of the displacement response with respect to the applied loading. In the presence of damping, the peak loading and peak response do not occur at the same time. Instead, the loading and response are separated by an interval of time measured in terms of a phase angle q as shown below:

$$\theta = -\tan^{-1}\frac{2\zeta\omega/\omega_n}{(1 - \omega^2/\omega_n^2)}$$

Equation 1-20.

The phase angle q is called the phase lead, which describes the amount that the response leads the applied force.

Note

Some texts define q as the phase lag, or the amount that the response lags the applied force. To convert from phase lag to phase lead, change the sign of q in Equation 1-19 and Equation 1-20.

Dynamic Amplification Factor with Damping

The dynamic amplification factor for the damped case is

$$\frac{1}{\sqrt{(1 - \omega^2/\omega_n^2)^2 + (2\zeta\omega/\omega_n)^2}}$$

Equation 1-21.

The interrelationship among the natural frequency, the applied load frequency, and the phase angle can be used to identify important dynamic characteristics. If w/w_n is much less than 1, the dynamic amplification factor approaches 1 and a static solution is represented with the displacement response in phase with the loading. If w/w_n is much greater than 1, the dynamic amplification factor approaches zero, yielding very little displacement response. In this case, the structure does not respond to the loading because the loading is changing too fast for the structure to respond. In addition, any measurable displacement response will be 180 degrees out of phase with the loading (i.e., the displacement response will have the opposite sign from the force). Finally if $w/w_n = 1$, resonance occurs. In this case, the magnification factor is 1/(2z), and

the phase angle is 270 degrees. The dynamic amplification factor and phase lead are shown in
Figure 1-5 and are plotted as functions of forcing frequency.
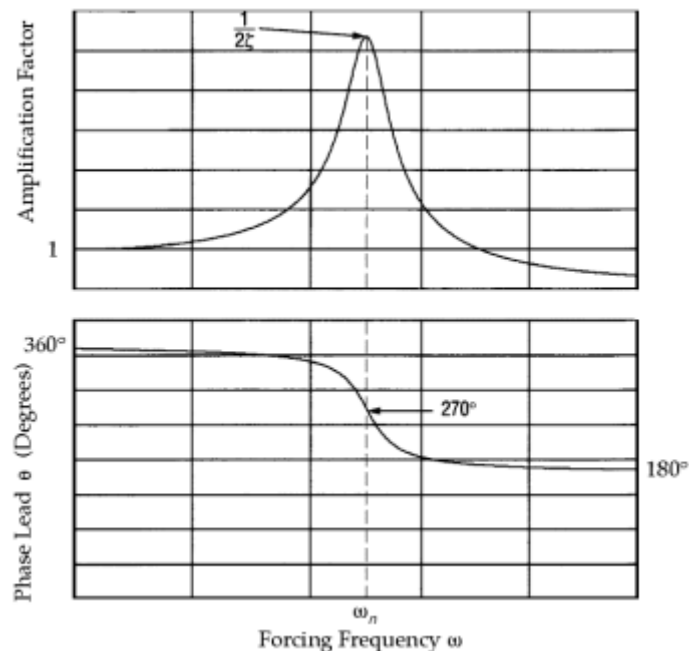
Figure 1-5. Harmonic Forced Response with Damping

In contrast to harmonic loadings, the more general forms of loading (impulses and general
transient loading) require a numerical approach to solving the equations of motion. This
technique, known as numerical integration, is applied to dynamic solutions either with or
without damping. Numerical integration is described in "Transient Response Analysis" .

1.3   Dynamic Analysis Process

Before conducting a dynamic analysis, you should first define the goal of the analysis prior to
the formulation of the finite element model. Consider the dynamic analysis process shown in
Figure 1-6. You must evaluate the finite element model in terms of the type of dynamic loading
to be applied to the structure. This dynamic load is known as the dynamic environment. The
dynamic environment governs the solution approach (i.e., normal modes, transient response,
frequency response, etc.). This environment also indicates the dominant behavior that must
be included in the analysis (i.e., contact, large displacements, etc.). Proper assessment of the
dynamic environment leads to the creation of a more refined finite element model and more
meaningful results.
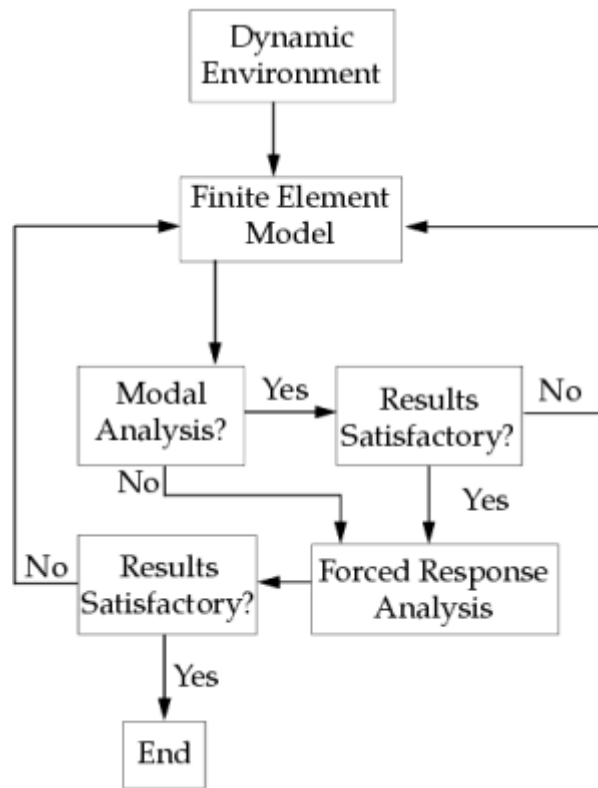
Figure 1-6. Overview of Dynamic Analysis Process

An overall system design is formulated by considering the dynamic environment. As part of the evaluation process, a finite element model is created. This model should take into account the characteristics of the system design and, just as importantly, the nature of the dynamic loading (type and frequency) and any interacting media (fluids, adjacent structures, etc.). At this point, the first step in many dynamic analyses is a modal analysis to determine the structure's natural frequencies and mode shapes (see "Real Eigenvalue Analysis").

In many cases the natural frequencies and mode shapes of a structure provide enough information to make design decisions. For example, in designing the supporting structure for a rotating fan, the design requirements may require that the natural frequency of the supporting structure have a natural frequency either less than 85% or greater than 110% of the operating speed of the fan. Specific knowledge of quantities such as displacements and stresses are not required to evaluate the design.

Forced response is the next step in the dynamic evaluation process. The solution process reflects the nature of the applied dynamic loading. A structure can be subjected to a number of different dynamic loads with each dictating a particular solution approach. The results of a forced response analysis are evaluated in terms of the system design. Necessary modifications are made to the system design. These changes are then applied to the model and analysis parameters to perform another iteration on the design. The process is repeated until an acceptable design is determined, which completes the design process.

The primary steps in performing a dynamic analysis are summarized as follows:

1. Define the dynamic environment (loading).

2. Formulate the proper finite element model.

3. Select and apply the appropriate analysis approach(es) to determine the behavior of the structure.

4. Evaluate the results.

1.4 Dynamic Analysis Types

You can perform the following types of basic dynamic analysis with NX Nastran:

*   Real eigenvalue analysis (undamped free vibrations).

*   Linear frequency response analysis (steady-state response of linear structures to loads that vary as a function of frequency).

*   Linear transient response analysis (response of linear structures to loads that vary as a function of time).

Additionally, NX Nastran allows you to perform a number of type of advanced dynamics analysis, such as shock/response spectrum analysis, random response analysis, design sensitivity, design optimization, aeroelasticity, and component mode synthesis.

Overview of Real Eigenvalue Analysis

Real eigenvalue analysis is used to determine the basic dynamic characteristics of a structure. The results of an eigenvalue analysis indicate the frequencies and shapes at which a structure naturally tends to vibrate. Although the results of an eigenvalue analysis are not based on a specific loading, they can be used to predict the effects of applying various dynamic loads. Real eigenvalue analysis is described in "Real Eigenvalue Analysis" .

Overview of Frequency Response Analysis

Frequency response analysis is an efficient method for finding the steady-state response to sinusoidal excitation. In frequency response analysis, the loading is a sine wave for which the frequency, amplitude, and phase are specified. Frequency response analysis is limited to linear elastic structures. Frequency response analysis is described in Frequency Response Analysis.

Overview of Transient Response analysis

Transient response analysis is the most general method of computing the response to time-varying loads. The loading in a transient analysis can be of an arbitrary nature, but is explicitly defined (i.e., known) at every point in time. The time-varying (transient) loading can also include nonlinear effects that are a function of displacement or velocity. Transient response analysis is most commonly applied to structures with linear elastic behavior. Transient response analysis is described in "Transient Response Analysis".

Additional types of dynamic analysis are available with NX Nastran. These types are described briefly in "Advanced Dynamic Analysis Capabilities" and will be described fully in the *NX Nastran Advanced Dynamic Analysis User's Guide*.

Chapter

2 *Finite Element Input Data*

- *Overview*

- *Mass Input*

- *Damping Input*

- *Units in Dynamic Analysis*

- *Direct Matrix Input*

2.1   Overview

When you use NX Nastran to perform a dynamic analysis, how you construct the model depends upon the desired results and the type of dynamic loading. For example, you need to obtain stress data, you would only need to define a finer finite element mesh if you needed to obtain system level displacement patterns. Many of the modeling considerations involved in a static analysis are similarly applied in dynamic analysis. However, with a dynamic analysis, you must also include additional input data to define the dynamic character of the structure.

In static analysis the stiffness properties are defined by element and material properties. These same properties are also required for dynamic analysis along with the addition of mass and damping. "Mass Input" describes mass input, and "Damping Input" describes damping input.

Correct specification of units is very important for dynamic analysis. Incorrect specification of units is probably more difficult to diagnose in dynamic analysis than in static analysis. Because NX Nastran has no built-in set of units, you must ensure their consistency (and accuracy). "Units in Dynamic Analysis" describes the common variables and units for dynamic analysis.

"Direct Matrix Input" concludes this chapter with a discussion of direct matrix input.

2.2   Mass Input

Mass input is one of the major entries in a dynamic analysis. Mass can be represented in a number of ways in NX Nastran. The mass matrix is automatically computed when mass density or nonstructural mass is specified for any of the standard finite elements (CBAR, CQUAD4, etc.) in NX Nastran, when concentrated mass elements are entered, and/or when full or partial mass matrices are entered.

Lumped and Coupled Mass

Mass is formulated as either lumped mass or coupled mass. Lumped mass matrices contain uncoupled, translational components of mass. Coupled mass matrices contain translational components of mass with coupling between the components. The CBAR, CBEAM, and CBEND elements contain rotational masses in their coupled formulations, although torsional inertias are not considered for the CBAR element. Coupled mass can be more accurate than lumped mass. However, lumped mass is more efficient and is preferred for its computational speed in dynamic analysis.

The mass matrix formulation is a user-selectable option in NX Nastran. The default mass formulation is lumped mass for most NX Nastran finite elements. The coupled mass matrix formulation is selected using PARAM,COUPMASS,1 in the Bulk Data. Table 2-1 shows the mass options available for each element type.

Table 2-1. Element Mass Types

Element Type	Lumped Mass	Coupled Mass[1]
CBAR	X	X
CBEAM	X	X
CBEND		X
CONEAX	X	
CONMi	X	X
CONROD	X	X

Table 2-1. Element Mass Types		
Element Type	Lumped Mass	Coupled Mass[1]
CRAC2D	X	X
CRAC3D	X	X
CHEXA	X	X
CMASSi	X	
CPENTA	X	X
CQUAD4	X	X
CQUAD8	X	X
CQUADR	X	X
CROD	X	X
CSHEAR	X	
CTETRA	X	X
CTRIA3	X	X
CTRIA6	X	X
CTRIAR	X	X
CTRIAX6	X	X
CTUBE	X	X
[1]Couple mass is selected by PARAM,COUPMASS,1.		

The NX Nastran coupled mass formulation is a modified approach to the classical consistent mass formulation found in most finite element texts. The NX Nastran lumped mass is identical to the classical lumped mass approach. The various formulations of mass matrices can be compared using the CROD element. Assume the following properties:

2 (Torsion)   4 (Torsion)   Length L
1 (Translation)   3 (Translation)   Area A
Torsional Constant J
Young's Modulus E
Shear Modulus G
Mass Density ρ
Polar Moment of Inertia I ρ
1-4 are
Degrees-of-Freedom

CROD Element Stiffness Matrix

The CROD element's stiffness matrix [*K*] is given by:

$$[K] = \begin{array}{cccc} 1 & 2 & 3 & 4 \end{array}$$

$$[K] = \begin{bmatrix} \dfrac{AE}{L} & 0 & \dfrac{-AE}{L} & 0 \\[2mm] 0 & \dfrac{GJ}{L} & 0 & \dfrac{-GJ}{L} \\[2mm] \dfrac{-AE}{L} & 0 & \dfrac{AE}{L} & 0 \\[2mm] 0 & \dfrac{-GJ}{L} & 0 & \dfrac{GJ}{0} \end{bmatrix} \begin{array}{c} 1 \\ 2 \\ 3 \\ 4 \end{array}$$

Equation 2-1.

The zero entries in the matrix create independent (uncoupled) translational and rotational behavior for the CROD element, although for most other elements these degrees-of-freedom are coupled.

CROD Lumped Mass Matrix

The CROD element classical lumped mass matrix is the same as the NX Nastran lumped mass matrix. This lumped mass matrix is

$$[M] = \rho AL \begin{bmatrix} \dfrac{1}{2} & 0 & 0 & 0 \\[2mm] 0 & 0 & 0 & 0 \\[2mm] 0 & 0 & \dfrac{1}{2} & 0 \\[2mm] 0 & 0 & 0 & 0 \end{bmatrix}$$

Equation 2-2.

The lumped mass matrix is formed by distributing one-half of the total rod mass to each of the translational degrees-of-freedom. These degrees-of-freedom are uncoupled and there are no torsional mass terms calculated.

The CROD element classical consistent mass matrix is

$$[M] = \rho AL \begin{bmatrix} \frac{1}{3} & 0 & \frac{1}{6} & 0 \\ 0 & \frac{I_\rho}{3A} & 0 & \frac{I_\rho}{6A} \\ \frac{1}{6} & 0 & \frac{1}{3} & 0 \\ 0 & \frac{I_\rho}{6A} & 0 & \frac{I_\rho}{3A} \end{bmatrix}$$

Equation 2-3.

This classical mass matrix is similar in form to the stiffness matrix because it has both translational and rotational masses. Translational masses may be coupled to other translational masses, and rotational masses may be coupled to other rotational masses. However, translational masses may not be coupled to rotational masses.

CROD Coupled Mass Matrix

The CROD element NX Nastran coupled mass matrix is

$$[M] = \rho AL \begin{bmatrix} \frac{5}{12} & 0 & \frac{1}{12} & 0 \\ 0 & 0 & 0 & 0 \\ \frac{1}{12} & 0 & \frac{5}{12} & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

Equation 2-4.

The axial terms in the CROD element coupled mass matrix represent the average of lumped mass and classical consistent mass. This average is found to yield the best results for the CROD element as described below. The mass matrix terms in the directions transverse to the element axes are lumped mass, even when the coupled mass option is selected. Note that the torsional inertia is not included in the CROD element mass matrix.

Lumped Mass Versus Coupled Mass Example

The difference in the axial mass formulations can be demonstrated by considering a fixed-free rod modeled with a single CROD element (Figure 2-1). The exact quarter-wave natural frequency for the first axial mode is

$$1.5708 \sqrt{\frac{E/\rho}{l}}$$

Using the lumped mass formulation for the CROD element, the first frequency is predicted to be

$$1.414 \sqrt{\frac{E/\rho}{l}}$$

which underestimates the frequency by 10%. Using a classical consistent mass approach, the predicted frequency

$$1.732 \sqrt{\frac{E/\rho}{l}}$$

is overestimated by 10%. Using the coupled mass formulation in NX Nastran, the frequency

$$1.549 \sqrt{\frac{E/\rho}{l}}$$

is underestimated by 1.4%. The purpose of this example is to demonstrate the possible effects of the different mass formulations on the results of a simple problem. Remember that not all dynamics problems have such a dramatic difference. Also, as the model's mesh becomes finer, the difference in mass formulations becomes negligible.
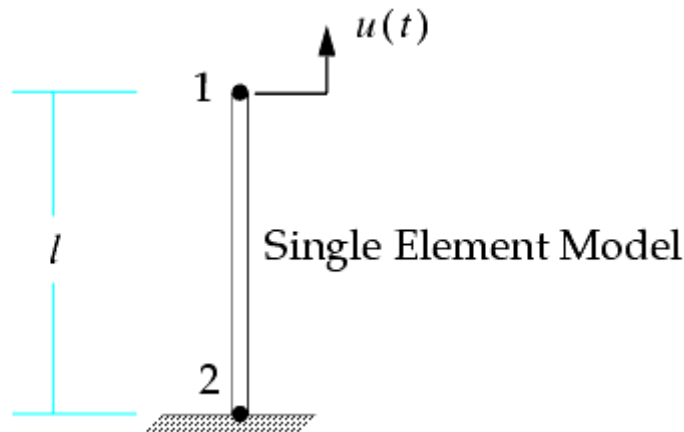
Theoretical Natural Frequency	$\omega_n = \dfrac{\pi}{2} \dfrac{\sqrt{E/\rho}}{l} = 1.5708 \dfrac{\sqrt{E/\rho}}{l}$
NX Nastran Lumped Mass	$\omega_n = \sqrt{2} \dfrac{\sqrt{E/\rho}}{l} = 1.414 \dfrac{\sqrt{E/\rho}}{l}$
Classical Consistent Mass	$\omega_n = \sqrt{3} \dfrac{\sqrt{E/\rho}}{l} = 1.732 \dfrac{\sqrt{E/\rho}}{l}$

NX Nastran Coupled Mass
$$\omega_n = \sqrt{12/5}\,\frac{\sqrt{E/\rho}}{l} = 1.549\,\frac{\sqrt{E/\rho}}{l}$$

Figure 2-1. Comparison of Mass Formulations for a ROD

CBAR, CBEAM Lumped Mass

The CBAR element lumped mass matrix is identical to the CROD element lumped mass matrix. The CBEAM element lumped mass matrix is identical to that of the CROD and CBAR mass matrices with the exception that torsional inertia is included.

CBAR, CBEAM Coupled Mass

The CBAR element coupled mass matrix is identical to the classical consistent mass formulation except for two terms: (1) the mass in the axial direction is the average of the lumped and classical consistent masses, as explained for the CROD element; and (2) there is no torsional inertia. The CBEAM element coupled mass matrix is also identical to the classical consistent mass formulation except for two terms: (1) the mass in the axial direction is the lumped mass; and (2) the torsional inertia is the lumped inertia.

Another important aspect of defining mass is the units of measure associated with the mass definition. NX Nastran assumes that consistent units are used in all contexts. You must be careful to specify structural dimensions, loads, material properties, and physical properties in a consistent set of units.

All mass entries should be entered in mass consistent units. Weight units may be input instead of mass units, if this is more convenient. However, you must convert the weight to mass by dividing the weight by the acceleration of gravity defined in consistent units:

$$\rho_m = (1/g)\rho_w$$

Equation 2-5.

where:

ρ_m	=	mass or mass density
g	=	acceleration of gravity
ρ_w	=	weight or weight density

The parameter

PARAM,WTMASS,factor

performs this conversion. The value of the factor should be entered as 1/*g*. The default value for the factor is 1.0. Hence, the default value for WTMASS assumes that mass (and mass density) is entered, instead of weight (and weight density).

When using English units if the weight density of steel is entered as RHO = 0.3 lb/in^3, using PARAM,WTMASS,0.002588 converts the weight density to mass density for the acceleration

of gravity g = 386.4 in/sec^2. The mass density, therefore, becomes 7.76E-4 lb$_f$-sec^2/in^4. If the weight density of steel is entered as RHO = 80000 N/m^3 when using metric units, then using PARAM,WTMASS,0.102 converts the weight density to mass density for the acceleration of gravity g = 9.8 m/sec^2 . The mass density, therefore, becomes 8160 kg/m^3 .

PARAM,WTMASS is used once per run, and it multiplies all weight/mass input (including CMASSi, CONMi, and nonstructural mass input). Therefore, do not mix input type; use all mass (and mass density) input or all weight (or weight density) input. PARAM,WTMASS does not affect direct input matrices M2GG or M2PP (see "Direct Matrix Input"). PARAM,CM2 can be used to scale M2GG; there is no parameter scaling for M2PP. PARAM,CM1 is similar to PARAM,WTMASS since CM1 scales all weight/mass input (except for M2GG and M2PP), but it is active only when M2GG is also used. In other words, PARAM,CM1 is used in addition to PARAM,WTMASS if M2GG is used.

NX Nastran Mass Input

Mass is input to NX Nastran via a number of different entries. The most common method to enter mass is using the RHO field on the MATi entry. This field is assumed to be defined in terms of mass density (mass/unit volume). To determine the total mass of the element, the mass density is multiplied by the element volume (determined from the geometry and physical properties). For a MAT1 entry, a mass density for steel of 7.76E-4 lb$_f$-sec^2/in^4 is entered as follows:

1	2	3	4	5	6	7	8	9	10
$MAT1	MID	E	G	NU	RHO	A	TREF	GE	
MAT1	2	30.0E6		0.3	7.76E-4				

Grid point masses can be entered using the CONM1, CONM2, and CMASSi entries. The CONM1 entry allows input of a fully coupled 6x6 mass matrix. You define half of the terms, and symmetry is assumed. The CONM2 entry defines mass and mass moments of inertia for a rigid body. The CMASSi entries define scalar masses.

Nonstructural Mass

An additional way to input mass is to use nonstructural mass, which is mass not associated with the geometric cross-sectional properties of an element. Examples of nonstructural mass are insulation, roofing material, and special coating materials. Nonstructural mass is input as mass/length for line elements and mass/area for elements with two-dimensional geometry. Nonstructural mass is defined on the element property entry (PBAR, for example).

2.3   Damping Input

Damping is a mathematical approximation used to represent the energy dissipation observed in structures. Damping is difficult to model accurately since it is caused by many mechanisms including

- Viscous effects (dashpot, shock absorber)

- External friction (slippage in structural joints)

- Internal friction (characteristic of the material type)

- Structural nonlinearities (plasticity, gaps)

Because these effects are difficult to quantify, damping values are often computed based on the results of a dynamic test. Simple approximations are often justified because the damping values are low.

Viscous and Structural Damping

Two types of damping are generally used for linear-elastic materials: viscous and structural. The viscous damping force is proportional to velocity, and the structural damping force is proportional to displacement. Which type to use depends on the physics of the energy dissipation mechanism(s) and is sometimes dictated by regulatory standards.

The viscous damping force f_v is proportional to velocity and is given by

$$f_v = b\dot{u}$$

Equation 2-6.

where:

b	=	viscous damping coefficient
$\dot{u}$	=	velocity

The structural damping force f_s is proportional to displacement and is given by

$$f_s = i \cdot G \cdot k \cdot u$$

Equation 2-7.

where:

G	=	structural damping coefficient
k	=	stiffness
u	=	displacement
i	=	$\sqrt{-1}$ (phase change of 90 degrees)

For a sinusoidal displacement response of constant amplitude, the structural damping force is constant, and the viscous damping force is proportional to the forcing frequency. Figure 2-2 depicts this and also shows that for constant amplitude sinusoidal motion the two damping forces are equal at a single frequency.

At this frequency,

$$G\,k = b\,\omega^* \quad \text{or} \quad b = \frac{Gk}{\omega^*}$$

Equation 2-8.

where ω^* is the frequency at which the structural and viscous damping forces are equal for a constant amplitude of sinusoidal motion.
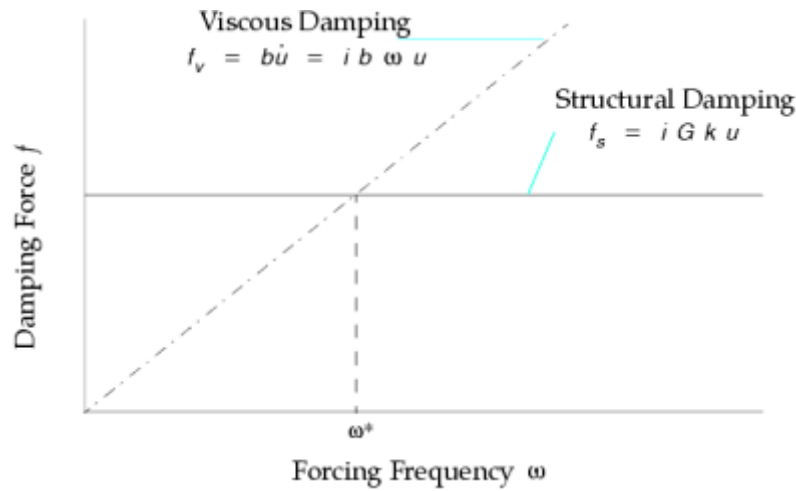
Figure 2-2. Structural Damping and Viscous Damping Forces for Constant Amplitude Sinusoidal Displacement

If the frequency w^* is the circular natural frequency w_n , Equation 2-8 becomes

$$b = \frac{G\,k}{\omega_n} = G\,\omega_n\,m$$

Equation 2-9.

Recall the definition of critical damping from Equation 1-11

$$b_{cr} = 2\sqrt{km} = 2m\omega_n$$

Equation 2-10.

Some equalities that are true at resonance (w_n) for constant amplitude sinusoidal displacement are

$$\frac{b}{b_{cr}} = \zeta = \frac{G}{2}$$

Equation 2-11.

and

$$Q = \frac{1}{2\zeta} = \frac{1}{G}$$

Equation 2-12.

where Q is the quality or dynamic magnification factor, which is inversely proportional to the energy dissipated per cycle of vibration.

The Effect of Damping

Damping is the result of many complicated mechanisms. The effect of damping on computed response depends on the type and loading duration of the dynamic analysis. Damping can often be ignored for short duration loadings, such as those resulting from a crash impulse or a shock blast, because the structure reaches its peak response before significant energy has had time to dissipate. Damping is important for long duration loadings (such as earthquakes), and is critical for loadings (such as rotating machinery) that continually add energy to the structure. The proper specification of the damping coefficients can be obtained from structural tests or from published literature that provides damping values for structures similar to yours.

As is discussed in detail in "Frequency Response Analysis"5 and "Transient Response Analysis" , certain solution methods allow specific forms of damping to be defined. The type of damping used in the analysis is controlled by both the solution being performed and the NX Nastran data entries. In transient response analysis, for example, structural damping must be converted to equivalent viscous damping.

Structural Damping Specification

Structural damping is specified on the MATi and PARAM,G Bulk Data entries. The GE field on the MATi entry is used to specify overall structural damping for the elements that reference this material entry. This definition is via the structural damping coefficient GE.

For example, the MAT1 entry:

1	2	3	4	5	6	7	8	9	10
$MAT1	MID	E	G	NU	RHO	A	TREF	GE	
MAT1	2	30.0E6		0.3	7.764E-4			0.10	

specifies a structural damping coefficient of 0.1.

An alternate method for defining structural damping is through PARAM,G,r where r is the structural damping coefficient. This parameter multiplies the stiffness matrix to obtain the structural damping matrix. The default value for PARAM,G is 0.0. The default value causes this source of structural damping to be ignored. Two additional parameters are used in transient response analysis to convert structural damping to equivalent viscous damping: PARAM,W3 and PARAM,W4.

PARAM,G and GE can both be specified in the same analysis.

Viscous Damping Specification

Viscous damping is defined by the following elements:

CDAMP1 entry	Scalar damper between two degrees-of-freedom (DOFs) with reference to a PDAMP property entry.
CDAMP2 entry	Scalar damper between two DOFs without reference to a property entry.
CDAMP3 entry	Scalar damper between two scalar points (SPOINTs) with reference to a PDAMP property entry.
CDAMP4 entry	Scalar damper between two scalar points (SPOINTs) without reference to a property entry.
CVISC entry	Element damper between two grid points with reference to a PVISC property entry.

CBUSH entry   A generalized spring-and-damper structural element that may be nonlinear or frequency dependent. It references a PBUSH entry.

Viscous damping for modal transient response and modal frequency response is specified with the TABDMP1 entry.

Note that GE and G by themselves are dimensionless; they are multipliers of the stiffness. The CDAMPi and CVISC entries, however, have damping units.

Damping is further described in Frequency Response Analysis and "Transient Response Analysis" as it pertains to frequency and transient response analyses.

2.4   Units in Dynamic Analysis

Because NX Nastran doesn't assume a particular set of units, you must ensure that the units in your NX Nastran model are consistent. Because there is more input in dynamic analysis than in static analysis, it is easier to make a mistake in units when performing a dynamic analysis. The most frequent source of error in dynamic analysis is incorrect specification of the units, especially for mass and damping.

Table 2-2 shows typical dynamic analysis variables, fundamental and derived units, and common English and metric units. Note that for English units all "lb" designations are lb_f . The use of "lb" for mass (i.e., lb_m) is avoided.

Table 2-2. Engineering Units for Common Variables			
Variable	**Dimensions**[1]	**Common English Units**	**Common Metric Units**
Length	L	in	m
Mass	M	lb-sec^2/in	kg
Time	T	sec	sec
Area	L^2	in^2	m^2
Volume	L^3	in^3	m^3
Velocity	LT^{-1}	in/sec	m/sec
Acceleration	LT^{-2}	in/sec^2	m/sec^2
Rotation	–	rad	rad
Rotational Velocity	T^{-1}	rad/sec	rad/sec
Rotational Acceleration	T^{-2}	rad/sec^2	rad/sec^2
Circular Frequency	T^{-1}	rad/sec	rad/sec
Frequency	T^{-1}	cps; Hz	cps; Hz
Eigenvalue	T^{-2}	rad^2/sec^2	rad^2/sec^2
Phase Angle	–	deg	deg
Force	MLT^{-2}	lb	N
Weight	MLT^{-2}	lb	N

Table 2-2. Engineering Units for Common Variables			
Variable	**Dimensions**[1]	**Common English Units**	**Common Metric Units**
Moment	$ML2T^{-2}$	in-lb	N-m
Mass Density	ML^{-3}	lb-sec^2/in^4	kg/m^3
Young's Modulus	$ML^{-1}T^{-2}$	lb/in2	Pa; N/m^2
Poisson's Ratio	–	–	–
Shear Modulus	$ML^{-1}T^{-2}$	lb/in^2	Pa; N/m^2
Area Moment of Inertia	L^4	in^4	m^4
Torsional Constant	L^4	in^4	m^4
Mass Moment of Inertia	ML^2	in-lb-sec^2	kg-m^2
Stiffness	MT^{-2}	lb/in	N/m
Viscous Damping Coefficient	MT^{-1}	lb-sec/in	N-sec/m
Stress	$ML^{-1}T^{-2}$	lb/in^2	Pa; N/m^2
Strain	–	–	–
L denotes length, M denotes mass , T denotes time, – denotes dimensionless			

2.5   Direct Matrix Input

The finite element approach simulates the structural properties with mathematical equations written in matrix format. The structural behavior is then obtained by solving these equations. Usually, all of the structural matrices are generated internally based on the information that you provide in your NX Nastran model. Once you provide the grid point locations, element connectivities, element properties, and material properties, NX Nastran generates the appropriate structural matrices.

External Matrices

If structural matrices are available externally, you can input the matrices directly into NX Nastran without providing all the modeling information. Normally this is not a recommended procedure since it requires additional work on your part. However, there are occasions where the availability of this feature is very useful and in some cases crucial. Some possible applications are listed below:

- Suppose you are a subcontractor on a classified project. The substructure that you are analyzing is attached to the main structure built by the main contractor. The stiffness and mass effects of this main structure are crucial to the response of your component, but geometry of the main structure is classified. The main contractor, however, can provide you with the stiffness and mass matrices of the classified structure. By reading these stiffness and mass matrices and adding them to your NX Nastran model, you can account for the effect of the attached structure without compromising security.

- Perhaps you are investigating a series of design options on a component attached to an aircraft bulkhead. Your component consists of 500 DOFs and the aircraft model consists of 100,000 DOFs. The flexibility of the backup structure is somewhat important. You can certainly analyze your component by including the full aircraft model (100,500 DOFs). However, as an approximation, you can reduce the matrices for the entire aircraft down to a manageable size using dynamic reduction (see "Advanced Dynamic Analysis Capabilities"). These reduced mass and stiffness matrices can then be read and added to your various component models. In this case, you may be analyzing a 2000-DOF system, instead of a 100,500-DOF system.

- The same concept can be extended to a component attached to a test fixture. If the finite element model of the fixture is available, then the reduced mass and stiffness matrices of the fixture can be input. Furthermore, there are times whereby the flexibility of the test fixture at the attachment points can be measured experimentally. The experimental stiffness matrix is the inverse of the measured flexibility matrix. In this instance, this experimental stiffness matrix can be input to your model.

One way of reading these external matrices is through the use of the direct matrix input feature in NX Nastran.

Direct Matrix Input

The direct matrix input feature can be used to input stiffness, mass, viscous damping, structural damping, and load matrices attached to the grid and/or scalar points in dynamic analysis. These matrices are referenced in terms of their external grid IDs and are input via DMIG Bulk Data entries. As shown in Table 2-3, there are seven standard kinds of DMIG matrices available in dynamic analysis.

Table 2-3. Types of DMIG Matrices in Dynamics

Matrix	G Type	P Type
Stiffness	K2GG	K2PP
Mass	M2GG	M2PP
Viscous Damping	B2GG	B2PP
Structural Damping	K42GG	–
Load	P2G	–

The symbols for g-type matrices in mathematical format are $[K^2_{gg}]$, $[M^2_{gg}]$, $[B^2_{gg}]$, $[K^{4,2}_{gg}]$, and $\{P^2_g\}$. The four matrices K2GG, M2GG, B2GG, and K42GG must be real and symmetric. These matrices are implemented at the g-set level (see "The Set Notation System Used in Dynamic Analysis" for a description of the set notation for dynamic analysis). In other words, these terms are added to the corresponding structural matrices at the specified DOFs prior to the application of constraints (MPCs, SPCs, etc.).

The symbols for p-type matrices in standard mathematical format are $[K^2_{pp}]$, $[M^2_{pp}]$, and $[B^2_{pp}]$. The p-set is a union of the g-set and extra points. These matrices need not be real or symmetric. The p-type matrices are used in applications such as control systems. Only the g-type DMIG input matrices are covered in this guide.

DMIG Bulk Data User Interface

In the Bulk Data Section, the DMIG matrix is defined by a single DMIG header entry followed by a series of DMIG data entries. Each of these DMIG data entries contains a column of nonzero terms for the matrix.

Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMIG	NAME	"0"	IFO	TIN	TOUT	POLAR		NCOL	

Column Entry Format:

DMIG	NAME	GJ	CJ		G1	C1	A1	B1	
	G2	C2	A2	B2	-etc.-				

Example:

DMIG	STIF	0	6	1					
DMIG	STIF	5	3		5	3	250.		
	5	5	-125.		6	3	-150.		

Field	Contents
NAME	Name of the matrix.
IFO	Form of matrix input:
	1 = Square
	9 or 2 = Rectangular
	6 = Symmetric (input only the upper or lower half)
TIN	Type of matrix being input:
	1 = Real, single precision (one field is used per element)
	2 = Real, double precision (one field per element)
	3 = Complex, single precision (two fields are used per element)
	4 = Complex, double precision (two fields per element)
TOUT	Type of matrix to be created:
	0 = Set by precision system cell (default)
	1 = Real, single precision
	2 = Real, double precision
	3 = Complex, single precision
	4 = Complex, double precision
POLAR	Input format of Ai, Bi. (Integer = blank or 0 indicates real, imaginary format; integer > 0 indicates amplitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9.
GJ	Grid, scalar, or extra point identification number for the column index or column number for IFO = 9.
CJ	Component number for GJ for a grid point.
Gi	Grid, scalar, or extra point identification number for the row index.

Field	Contents
Ci	Component number for Gi for a grid point.
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank.

DMIG Case Control User Interface

In order to include these matrices, the Case Control must contain the appropriate K2GG, M2GG, B2GG, or K42GG command. (Once again, only the g-type DMIG input matrices are included in this guide.)

Examples

1. K2GG = mystiff

   The above Case Control command adds terms that are defined by the DMIG Bulk Data entries with the name "mystiff" to the g-set stiffness matrix.

2. M2GG = yourmass

   The above Case Control command adds terms that are defined by the DMIG Bulk Data entries with the name "yourmass" to the g-set mass matrix.

3. B2GG = ourdamp

   The above Case Control command adds terms that are defined by the DMIG Bulk Data entries with the name "ourdamp" to the g-set viscous damping matrix.

4. K42GG = strdamp

   The above Case Control command adds terms that are defined by the DMIG Bulk Data entries with the name "strdamp" to the g-set structural damping matrix.

Use of the DMIG entry for inputting mass and stiffness is illustrated in one of the examples in "Real Eigenvalue Analysis".

Chapter

3   Real Eigenvalue Analysis

- *Overview*

- *Reasons to Compute Normal Modes*

- *Overview of Normal Modes Analysis*

- *Methods of Computation*

- *Comparison of Methods*

- *User Interface for Real Eigenvalue Analysis*

- *Solution Control for Normal Modes*

- *Examples*

3.1   Overview

The usual first step in performing a dynamic analysis is determining the natural frequencies and mode shapes of the structure with damping neglected. These results characterize the basic dynamic behavior of the structure and are an indication of how the structure will respond to dynamic loading.

Natural Frequencies

The natural frequencies of a structure are the frequencies at which the structure naturally tends to vibrate if it is subjected to a disturbance. For example, the strings of a piano are each tuned to vibrate at a specific frequency. Some alternate terms for the natural frequency are characteristic frequency, fundamental frequency, resonance frequency, and normal frequency.

Mode Shapes

The deformed shape of the structure at a specific natural frequency of vibration is termed its normal mode of vibration. Some other terms used to describe the normal mode are mode shape, characteristic shape, and fundamental shape. Each mode shape is associated with a specific natural frequency.

Natural frequencies and mode shapes are functions of the structural properties and boundary conditions. A cantilever beam has a set of natural frequencies and associated mode shapes (Figure 3-1). If the structural properties change, the natural frequencies change, but the mode shapes may not necessarily change. For example, if the elastic modulus of the cantilever beam is changed, the natural frequencies change but the mode shapes remain the same. If the boundary conditions change, then the natural frequencies and mode shapes both change. For example, if the cantilever beam is changed so that it is pinned at both ends, the natural frequencies and mode shapes change (see Figure 3-2).
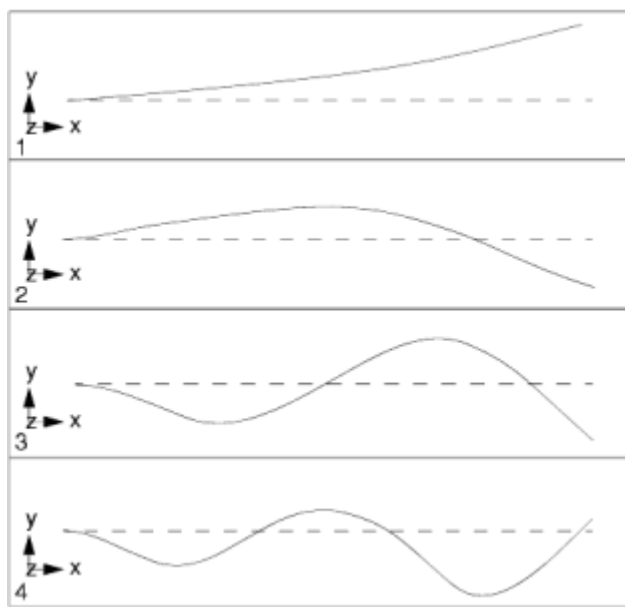
Figure 3-1. The First Four Mode Shapes of a Cantilever Beam
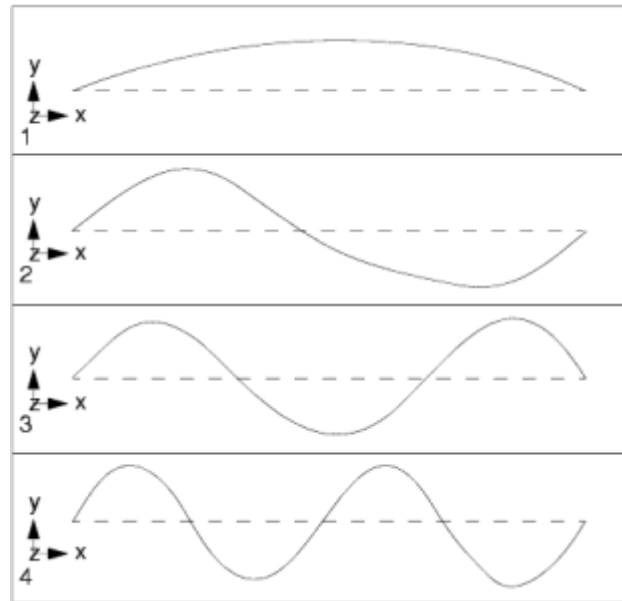
Figure 3-2. The First Four Mode Shapes of a Simply Supported Beam

Computation of the natural frequencies and mode shapes is performed by solving an eigenvalue problem as described in "Rigid-Body Mode of a Simple Structure" . Next, we solve for the eigenvalues (natural frequencies) and eigenvectors (mode shapes). Because damping is neglected in the analysis, the eigenvalues are real numbers. (The inclusion of damping makes the eigenvalues complex numbers; see "Advanced Dynamic Analysis Capabilities".) The solution for undamped natural frequencies and mode shapes is called real eigenvalue analysis or normal modes analysis.

The remainder of this chapter describes the various eigensolution methods for computing natural frequencies and mode shapes, and it concludes with several examples.

3.2   Reasons to Compute Normal Modes

There are many reasons to compute the natural frequencies and mode shapes of a structure. One reason is to assess the dynamic interaction between a component and its supporting structure. For example, if a rotating machine, such as an air conditioner fan, is to be installed on the roof of a building, it is necessary to determine if the operating frequency of the rotating fan is close to one of the natural frequencies of the building. If the frequencies are close, the operation of the fan may lead to structural damage or failure.

Decisions regarding subsequent dynamic analyses (i.e., transient response, frequency response, response spectrum analysis, etc.) can be based on the results of a natural frequency analysis. The important modes can be evaluated and used to select the appropriate time or frequency step for integrating the equations of motion. Similarly, the results of the eigenvalue analysis-the natural frequencies and mode shapes-can be used in modal frequency and modal transient response analyses (see Frequency Response Analysis and "Transient Response Analysis").

The results of the dynamic analyses are sometimes compared to the physical test results. A normal modes analysis can be used to guide the experiment. In the pretest planning stages, a normal modes analysis can be used to indicate the best location for the accelerometers. After the test, a normal modes analysis can be used as a means to correlate the test results to the analysis results.

Design changes can also be evaluated by using natural frequencies and normal modes. Does a particular design modification cause an increase in dynamic response? Normal modes analysis can often provide an indication.

In summary, there are many reasons to compute the natural frequencies and mode shapes of a structure. All of these reasons are based on the fact that real eigenvalue analysis is the basis for many types of dynamic response analyses. Therefore, an overall understanding of normal modes analysis as well as knowledge of the natural frequencies and mode shapes for your particular structure is important for all types of dynamic analysis.

3.3   Overview of Normal Modes Analysis

The solution of the equation of motion for natural frequencies and normal modes requires a special reduced form of the equation of motion. If there is no damping and no applied loading, the equation of motion in matrix form reduces to

$$[M]\{\ddot{u}\} + [K][u] = 0$$

Equation 3-1.

where:

$[M]$	=	mass matrix
$[K]$	=	stiffness matrix

This is the equation of motion for undamped free vibration. To solve Equation 3-1 assume a harmonic solution of the form

$$\{u\} = \{\phi\}\sin\omega t$$

Equation 3-2.

where:

$\{\phi\}$	=	the eigenvector or mode shape
ω	=	is the circular natural frequency

Aside from this harmonic form being the key to the numerical solution of the problem, this form also has a physical importance. The harmonic form of the solution means that all the degrees-of-freedom of the vibrating structure move in a synchronous manner. The structural configuration does not change its basic shape during motion; only its amplitude changes.

If differentiation of the assumed harmonic solution is performed and substituted into the equation of motion, the following is obtained:

$$-\omega^2[M]\{\phi\}\sin\omega t + [K]\{\phi\}\sin\omega t = 0$$

Equation 3-3.

which after simplifying becomes

$$([K] - \omega^2[M])\{\phi\} = 0$$

Equation 3-4.

This equation is called the eigenequation, which is a set of homogeneous algebraic equations for the components of the eigenvector and forms the basis for the eigenvalue problem. An eigenvalue problem is a specific equation form that has many applications in linear matrix algebra. The basic form of an eigenvalue problem is

$$[A - \lambda I]x = 0$$

Equation 3-5.

where:

A   =   square matrix

λ   =   eigenvalues

I   =   identity matrix

x   =   eigenvector

In structural analysis, the representations of stiffness and mass in the eigenequation result in the physical representations of natural frequencies and mode shapes. Therefore, the eigenequation is written in terms of K, ω, and M as shown in Equation 3-4 with $\omega^2 = \lambda$.

There are two possible solution forms for Equation 3-4:

1. If det $([K] - \omega^2[M]) \neq 0$, the only possible solution is

$$\{\phi\} = 0$$

Equation 3-6.

This is the trivial solution, which does not provide any valuable information from a physical point of view, since it represents the case of no motion. ("det" denotes the determinant of a matrix.)

2. If det $([K] - \omega^2[M]) = 0$, then a non-trivial solution ($\{\phi\} \neq 0$) is obtained for

$$([K] - \omega^2[M])\{\phi\} = 0$$

Equation 3-7.

From a structural engineering point of view, the general mathematical eigenvalue problem reduces to one of solving the equation of the form

$$\det\left([K] - \omega^2[M]\right) = 0$$

Equation 3-8.

or

$$\det\left([K] - \lambda[M]\right) = 0$$

Equation 3-9.

where $\lambda = \omega^2$

The determinant is zero only at a set of discrete eigenvalues λ_i or ω^2_i . There is an eigenvector $\{\phi_i\}$ which satisfies Equation 3-7 and corresponds to each eigenvalue. Therefore, Equation 3-7 can be rewritten as

$$[K - \omega_i^2 M]\{\phi_i\} = 0 \qquad i = 1, 2, 3\ldots$$

Equation 3-10.

Each eigenvalue and eigenvector define a free vibration mode of the structure. The i-th eigenvalue λ_i is related to the i-th natural frequency as follows:

$$f_i = \frac{\omega_i}{2\pi}$$

Equation 3-11.

where:

f_i	=	i-th natural frequency
ω_i	=	$\sqrt{\lambda_i}$

The number of eigenvalues and eigenvectors is equal to the number of degrees-of-freedom that have mass or the number of dynamic degrees-of-freedom.

There are a number of characteristics of natural frequencies and mode shapes that make them useful in various dynamic analyses. First, when a linear elastic structure is vibrating in free or forced vibration, its deflected shape at any given time is a linear combination of all of its normal modes

$$\{u\} = \sum_i (\phi_i)\xi_i$$

Equation 3-12.

where:

$\{u\}$ =   vector of physical displacements

$\{\phi_i\}$ =   i-th mode shape

x_i =   i-th modal displacement

Second, if $[K]$ and $[M]$ are symmetric and real (as is the case for all the common structural finite elements), the following mathematical properties hold:

$$\{\phi_i\}^T [M]\{\phi_j\} \; = \; 0 \qquad \text{if } i \neq j$$

Equation 3-13.

$$\{\phi_j\}^T [M]\{\phi_j\} \; = \; m_j \; = \; \text{j-th generalized mass}$$

Equation 3-14.

and

$$\{\phi_j\}^T [K]\{\phi_j\} \; = \; 0 \qquad \text{if } i \neq j$$

Equation 3-15.

$$\{\phi_j\}^T [K]\{\phi_j\} \; = \; k_j \; = \; \text{j-th generalized stiffness} \; = \; \omega^2 m_j$$

Equation 3-16.

Also, from Equation 3-14 and Equation 3-16 Rayleigh's equation is obtained

$$\omega_j^2 \; = \; \frac{\{\phi_j\}^T [K]\{\phi_j\}}{\{\phi_j\}^T [M]\{\phi_j\}}$$

Equation 3-17.

Equation 3-13 and Equation 3-15 are known as the orthogonality property of normal modes, which ensures that each normal mode is distinct from all others. Physically, orthogonality of modes means that each mode shape is unique and one mode shape cannot be obtained through a linear combination of any other mode shapes.

In addition, a natural mode of the structure can be represented by using its generalized mass and generalized stiffness. This is very useful in formulating equivalent dynamic models and in component mode synthesis (see "Advanced Dynamic Analysis Capabilities").

If a structure is not totally constrained in space, it is possible for the structure to displace (move) as a rigid body or as a partial or complete mechanism. For each possible component of rigid-body motion or mechanism, there exists one natural frequency which is equal to zero.

The zero-frequency modes are called rigid-body modes. Rigid-body motion of all or part of a structure represents the motion of the structure in a stress-free condition. Stress-free, rigid-body modes are useful in conducting dynamic analyses of unconstrained structures, such as aircraft and satellites. Also, rigid-body modes can be indicative of modeling errors or an inadequate constraint set.

For example, the simple unconstrained structure in Figure 3-3 has a rigid-body mode.

$$\omega_1 = 0 \qquad \{\phi_1\} = \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} 1 \\ 1 \end{Bmatrix}$$
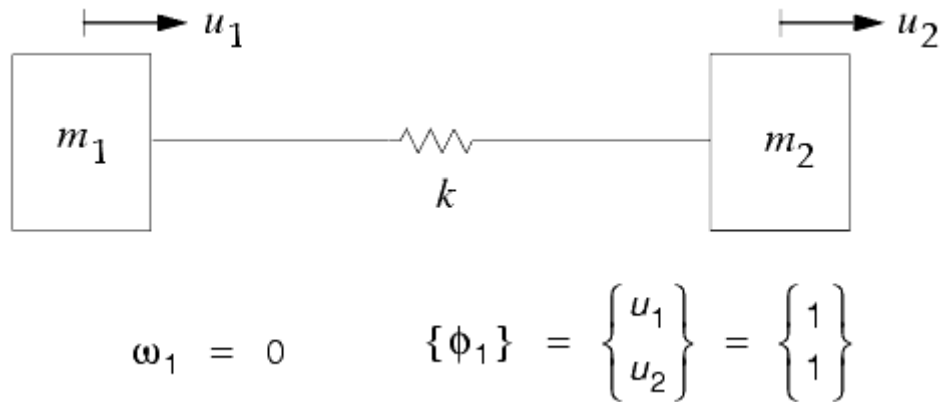
Figure 3-3. Rigid-Body Mode of a Simple Structure

When both masses move the same amount (as a rigid body), there is no force induced in the connecting spring. A detailed discussion of rigid-body modes is presented in "Rigid-body Modes".

An important characteristic of normal modes is that the scaling or magnitude of the eigenvectors is arbitrary. Mode shapes are fundamental characteristic shapes of the structure and are therefore relative quantities. In the solution of the equation of motion, the form of the solution is represented as a shape with a time-varying amplitude. Therefore, the basic mode shape of the structure does not change while it is vibrating; only its amplitude changes.

For example, three different ways to represent the two modes of a two-DOF structure are shown in Figure 3-4. The graphical representation of the eigenvectors in the figure shows the modal displacements rotated by 90 degrees in order to view the deformation better.

Mode 1   Mode 2
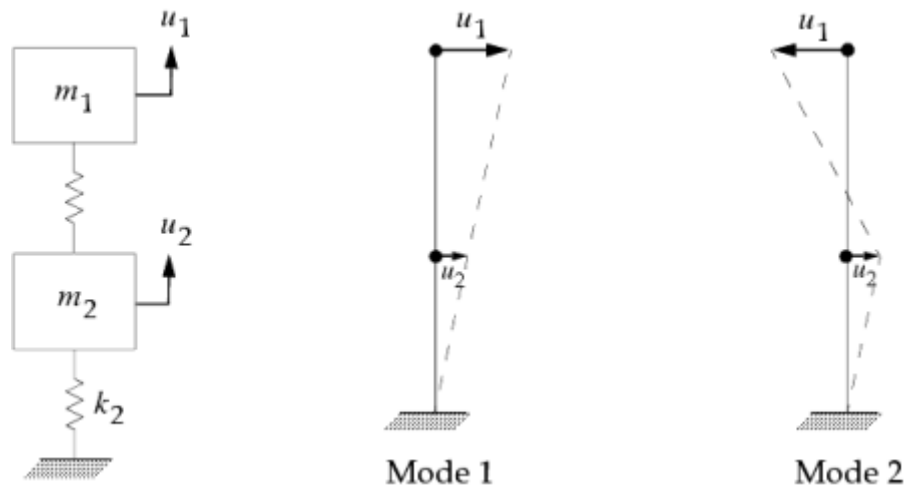
$$\phi_1 \; = \; \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} 3 \\ 1 \end{Bmatrix} = \begin{Bmatrix} 300 \\ 100 \end{Bmatrix} = \begin{Bmatrix} -0.6 \\ -0.2 \end{Bmatrix} \{\phi_2\} = \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} -1.6 \\ 0.8 \end{Bmatrix} = \begin{Bmatrix} 160 \\ -80 \end{Bmatrix} = \begin{Bmatrix} 1.0 \\ -0.5 \end{Bmatrix}$$

Figure 3-4. Representations of Mode Shapes for a Two-DOF System

A common misconception about mode shapes is that they define the structural response. Again, mode shapes are relative quantities. They cannot be used alone to evaluate dynamic behavior. As described earlier, it is the relation between the structural loading and the natural frequencies that determines the absolute magnitude of dynamic response. The relation of a specific loading to a set of natural frequencies provides explicit scale factors that are used to determine the extent to which each particular mode is excited by the loading. After the individual modal responses to a particular loading are determined, only then can the various engineering design decisions be made with the actual (absolute) values of stress and/or displacement. Methods that use the modal results to determine forced response are termed modal methods or modal superposition methods. Modal frequency response analysis and modal transient response analysis are described in Frequency Response Analysis and "Transient Response Analysis," respectively.

Mode Shape Normalization

Although the scaling of normal modes is arbitrary, for practical considerations mode shapes should be scaled (i.e., normalized) by a chosen convention. In NX Nastran there are three normalization choices, MASS, MAX, and POINT normalization.

MASS normalization is the default method of eigenvector normalization. This method scales each eigenvector to result in a unit value of generalized mass

$$\{\phi_j\}^T [M] \{\phi_j\} \; = \; 1.0$$

Equation 3-18.

Numerically this method results in a modal mass matrix that is an identity matrix. This normalization approach is appropriate for modal dynamic response calculations because it simplifies both computational and data storage requirements. When mass normalization is used with a model of a heavy, massive structure, the magnitude of each of the terms of the eigenvectors is very small.

In MAX normalization, each eigenvector is normalized with respect to the largest a-set component. ("Advanced Dynamic Analysis Capabilities" and "The Set Notation System Used in Dynamic Analysis" provide discussions of the a-set.) This normalization results in the largest a-set displacement value being set to a unit (1.0) value. This normalization approach can be very useful in the determination of the relative participation of an individual mode. A small generalized mass obtained using MAX normalization may indicate such things as local modes or isolated mechanisms.

POINT normalization of eigenvectors allows you to chose a specific displacement component at which the modal displacement is set to 1 or -1. This method is not recommended because for complex structures the chosen component in the non-normalized eigenvector may have a very small value of displacement (especially in higher modes). This small value can cause larger numbers to be normalized by a small number, resulting in possible numerical roundoff errors in mode shapes.

Although mode shapes are relative quantities, a number of modal quantities can be helpful in predicting qualitative responses or in isolating troublesome modal frequencies. Since relative

strains, internal loads, and stresses develop when a structure deforms in a mode shape, you may recover these quantities during a normal modes analyses. Basically, any quantity that you can recover for static analysis is also available for normal modes analysis.

It is important to remember that these output quantities are based on the relative displacements of a mode shape. The output quantities can be compared for a given mode, but not necessarily from one mode to another. However, they can still be effectively used in the analysis/design process.

Modal quantities can be used to identify problem areas by indicating the more highly stressed elements. Elements that are consistently highly stressed across many or all modes will probably be highly stressed when dynamic loads are applied.

Modal strain energy is a useful quantity in identifying candidate elements for design changes to eliminate problem frequencies. Elements with large values of strain energy in a mode indicate the location of large elastic deformation (energy). These elements are those which most directly affect the deformation in a mode. Therefore, changing the properties of these elements with large strain energy should have more effect on the natural frequencies and mode shapes than if elements with low strain energy were changed.

Structures with two or more identical eigenvalues are said to have repeated roots. Repeated roots occur for structures that have a plane of symmetry or that have multiple, identical pieces (such as appendages). The eigenvectors for the repeated roots are not unique because many sets of eigenvectors can be found that are orthogonal to each other. An eigenvector that is a linear combination of the repeated eigenvectors is also a valid eigenvector. Consequently, small changes in the model can make large changes in the eigenvectors for the repeated roots. Different computers can also find different eigenvectors for the repeated roots. Rigid-body modes (see "Rigid-body Modes") represent a special case of repeated roots.

3.4 Methods of Computation

Seven methods of real eigenvalue extraction are provided in NX Nastran. These methods are numerical approaches to solving for natural frequencies and modes shapes. The reason for seven different numerical techniques is because no one method is the best for all problems. While most of the methods can be applied to all problems, the choice is often based on the efficiency of the solution process.

The methods of eigenvalue extraction belong to one or both of the following two groups:

•   Transformation methods

•   Tracking methods

In the transformation method, the eigenvalue equation is first transformed into a special form from which eigenvalues may easily be extracted. In the tracking method, the eigenvalues are extracted one at a time using an iterative procedure.

The recommended real eigenvalue extraction method in NX Nastran is the Lanczos method. The Lanczos method combines the best characteristics of both the tracking and transformation methods. For most models the Lanczos method is the best method to use.

Two of the real eigenvalue extraction methods available in NX Nastran are transformation methods:

•   Householder method

•   Modified Householder method

Tthe real eigenvalue extraction method available in NX Nastran is classified as tracking methods:

- Sturm modified inverse power method

The remainder of this section briefly describes the various methods. The theory and algorithms behind each method can be found in the *NX Nastran Numerical Methods User's Guide*.

Lanczos Method

The Lanczos method overcomes the limitations and combines the best features of the other methods. It requires that the mass matrix be positive semidefinite and the stiffness be symmetric. Like the transformation methods, it does not miss roots, but has the efficiency of the tracking methods, because it only makes the calculations necessary to find the roots requested by the user. This method computes accurate eigenvalues and eigenvectors. Unlike the other methods, its performance has been continually enhanced since its introduction giving it an advantage. The Lanczos method is the preferred method for most medium- to large-sized problems, since it has a performance advantage over other methods.

Householder Methods

The Householder modal extraction method require a positive definite mass matrix (all degrees-of-freedom must have mass). There is no restriction on the stiffness matrix except that it must be symmetric. These matrices always result in real (positive) eigenvalues. The Householder method is the most efficient method for small problems and problems with dense matrices when a large portion of the eigenvectors are needed. This method finds all of the eigenvalues and as many eigenvectors as requested. While this method do not take advantage of sparse matrices, they are efficient with the dense matrices sometimes created using dynamic reduction (see "Advanced Dynamic Analysis Capabilities").

The Householder method fail if the mass matrix is not positive definite. To minimize this problem, degrees-of-freedom with null columns are removed by the automatic application of static condensation (see "Advanced Dynamic Analysis Capabilities") called auto-omit. Applying the auto-omit process is a precaution and may not remove all possible causes of mass matrix singularity, such as a point mass offset from a grid point, but it greatly improves the reliability and convenience of the Householder method.

The Householder method uses different transformation schemes to obtain the eigenvalues. The Householder method can take advantage of parallel processing computers.

Modified Householder Methods

The modified Householder method issimilar to the standard method with the exception that the mass matrix can be singular. Although the mass matrix is not required to be nonsingular in the modified method, a singular mass matrix can produce one or more infinite eigenvalues. Due to roundoff error, these infinite eigenvalues appear in the output as very large positive or negative eigenvalues. To reduce the incidence of such meaningless results, degrees-of-freedom with null masses are eliminated by automatic static condensation as in the case of the unmodified method.

The modified methods requires more computer time than the standard method.

Automatic Householder Methods

Many times you may not know whether the mass matrix is singular. To assist you in choosing the appropriate method, automatic Householder, is available. Initially the automatic method uses the standard method. In the first step of the method, if the mass matrix is not well-conditioned for decomposition, the method shifts to the corresponding modified method. The modified method is more expensive and may introduce numerical noise due to the shift, but they resolve most of the numerical problems of the ill-conditioned mass matrix. The automatic method, therefore, uses the modified method when necessary for numerical stability but uses the standard method when the numerical stability is accurate.

Sturm Modified Inverse Power Method

This method uses Sturm sequence logic to ensure that all modes are found. The Sturm sequence check determines the number of eigenvalues below a trial eigenvalue, then finds all of the eigenvalues below this trial eigenvalue until all modes in the designed range are computed. This process helps to ensure that modes are not missed.

The Sturm modified inverse power method is useful for models in which only the lowest few modes are needed. This method is also useful as a backup method to verify the accuracy of other methods.

3.5   Comparison of Methods

Since NX Nastran provides a variety of real eigensolution methods, you must decide which is best for your application. The best method for a particular model depends on four factors: the size of the model (the total number of degrees-of-freedom as well as the number of dynamic degrees-of-freedom), the number of eigenvalues desired, the available real memory of your computer, and the conditioning of the mass matrix (whether there are massless degrees-of-freedom). In general, the Lanczos method is the most reliable and efficient, and is the recommended choice.

For small, dense models whose matrices fit into memory, we recommend using the automatic method (automatic Householder). The automatic Householder runs modified methods if the mass matrix is singular; however, it runs the unmodified methods, which isfaster, if the mass matrix is not singular. The automatic Householder method runs faster on computers with vector processing and also supports parallel processing computers. Note that most real world problems are not small and dense, unless you use reductive methods such as superelements.

The Sturm modified inverse power method can be the best choice when the model is too large to fit into memory, only a few modes are needed, and a reasonable eigenvalue search range is specified. This method is also a backup method for the other methods and is used when a check of the other methods' results is needed.

For medium to large models the Lanczos method is the recommended method. In addition to its reliability and efficiency, the Lanczos method supports sparse matrix methods that substantially increase its speed and reduce disk space requirements.

Table 3-1. Comparison of Eigenvalue Methods

	Method			
	Householder	**Modified Householder**	**Sturm Modified Inverse Power**	**Lanczos**
Reliability	High	High	High	High

Table 3-1. Comparison of Eigenvalue Methods

	Method			
	Householder	**Modified Householder**	**Sturm Modified Inverse Power**	**Lanczos**
Relative Cost: Few Modes Many Modes	Medium High	Medium High	Low High	Medium Medium
Limitations	Cannot analyze singular [M] Expensive for problems that do not fit in memory	Expensive for many modes Expensive for problems that do not fit in memory	Expensive for many modes	Difficulty with massless mechanisms
Best Application	Small, dense matrices that fit in memory Use with dynamic reduction (Chapter 11)	Small, dense matrices that fit in memory Use with dynamic reduction (Chapter 11)	To determine a few modes Backup method	Medium to large models

3.6 User Interface for Real Eigenvalue Analysis

The EIGR and EIGRL Bulk Data entries define the method and select the parameters that control the eigenvalue extraction procedure. The EIGRL entry is used for the Lanczos method, and the EIGR entry is used for all of the other methods.

User Interface for the Lanczos Method

The EIGRL entry has the following format:

Format:

1	2	3	4	5	6	7	8	9	10
EIGRL	SID	V1	V2	ND	MSGLVL	MAXSET	SHFSCL	NORM	
	option_1=value_1, option_2=value_2, etc.								

Example:

EIGRL	1	0.1	3.2						

Field	Contents
SID	Set identification number. (Unique Integer > 0)
V1, V2	The V1 field defines the lower frequency bound; the V2 field defines the upper frequency field. For vibration analysis: frequency range of interest. For buckling analysis: eigenvalue range of interest. (Real or blank, $-5 \times 10^{16} \leq V1 < V2 \leq 5 \times 10^{16}$)
ND	Number of eigenvalues and eigenvectors desired. (Integer > 0 or blank)
MSGLVL	Diagnostic level. ($0 \leq$ Integer ≤ 4; Default = 0)
MAXSET	Number of vectors in block or set. ($1 \leq$ Integer ≤ 15; Default = 7)
SHFSCL	Estimate of the first flexible mode natural frequency. (Real or blank)
NORM	Method for normalizing eigenvectors (Character: "MASS" or "MAX")

Field	Contents
MASS	Normalize to unit value of the generalized mass. Not available for buckling analysis. (Default for normal modes analysis.)
MAX	Normalize to unit value of the largest displacement in the analysis set. Displacements not in the analysis set may be larger than unity. (Default for buckling analysis.)

Examples of the results of using explicit or default values for the V1, V2, and ND fields are shown in Table 3-2. The defaults on the EIGRL entry are designed to provide the minimum number of roots in cases where the input is ambiguous. Alternatively, you can write the entry above using the new free field format. You must specify certain new parameters such as ALPH,NUMS, and Fi when using the free field format. See the NX Nastran Quick Reference Guide for details.

1	2	3	4	5	6	7	8	9	10
EIGRL	1								
	V1 = 0.1, V2 = 3.2								

Table 3-2. Number and Type of Roots Found with the EIGRL Entry				
Case	V1	V2	ND	Number and Type of Roots Found
1	V1	V2	ND	Lowest ND in range or all in range, whichever is smaller
2	V1	V2		All in range
3	V1		ND	Lowest ND in range [V1, ∞]
4	V1			Lowest root in range [V1, ∞]
5			ND	Lowest ND roots in [-∞, ∞]
6				Lowest root
7		V2	ND	Lowest ND or all in range [-∞, V2], whichever is smaller
8		V2		All below V2

The MSGLVL field of the EIGRL entry is used to control the amount of diagnostic output. The value of 0 produces no diagnostic output. The values 1, 2, or 3 provide more output with the higher values providing increasingly more output. In some cases, higher diagnostic levels may help to resolve difficulties with special modeling problems.

The MAXSET field is used to control the block size. The default value of 7 is recommended for most applications. There may be special cases where a larger value may result in quicker convergence of many multiple roots or a lower value may result in more efficiency when the structure is lightly coupled. However, the default value has been chosen after reviewing the results from a wide range of problems on several different computer types with the goal of minimizing the computer time.

A common occurrence is for the block size to be reset by NX Nastran during the run because there is insufficient memory for a block size of 7. Computational efficiency tends to degrade as the block size decreases. Therefore, you should examine the eigenvalue analysis summary output to determine whether NX Nastran has sufficient memory to use an efficient block size. A smaller block size may be more efficient when only a few roots are requested. The minimum recommended block size is 2.

The SHFSCL field allows a user-designated shift to be used to improve performance especially when large mass techniques are used in enforced motion analysis (see "Enforced Motion"). Large mass techniques can cause a large gap between the rigid body (see "Rigid-body Modes")

and flexible frequencies, which can degrade performance of the Lanczos method or cause System Fatal Message 5299. When SHFSCL is used, its value should be set close to the expected first nonzero natural frequency.

The Lanczos method normalizes (i.e., scales) the computed eigenvectors using the MASS or MAX method. These methods are specified using the NORM field (new for Version 68). The MASS method normalizes to a unit value of the generalized mass (i.e., $m_j = 1.0$). The MAX method normalizes to a unit value of the largest component in the a-set (see "Advanced Dynamic Analysis Capabilities"). The default is MASS.

You can use the continuation entry to specify V1, V2, ND, MSGLVL, MAXSET, SHFSCL and NORM if you have not specified them on the parent entry. To apply the continuation entry use the following format: 'option_i=value_i', e.g., ND=6. Using the continuation entry is the only way to specify the three new options, ALPH, NUMS and Fi.

NUMS	The number of segments that a frequency range will be broken into for parallel processing. You must define a value greater than 1 to take advantage of parallel processing. You may also specify NUMS using the NUMSEG keyword on the NASTRAN statement. If you specify both, then NUMS takes precedence.
Fi	Directly specifies the upper frequencies of each segment, such that V1 < F1 < F2 < ... F15 < V2.
ALPH	Automatically generates the Fi values based on the following formula:

$$ Fi \ = \ (V2 - V1) \cdot (1.0 - ALPH^i) / (1.0 - ALPH^{NUMS}) $$

If you specify both ALPH and Fi, then Fi takes precedence over ALPH as long as they are consistent. If ALPH is multiplied by 100, it may be specified on the FRQSEQ keyword of the NASTRAN statement.

For a detailed description of the EIGRL input, see the *NX Nastran Numerical Methods User's Guide*.

User Interface for the Other Methods

The data entered on the EIGR entry selects the eigenvalue method and the frequency range or number of required roots. The basic format of the Bulk Data entry is as follows:

1	2	3	4	5	6	7	8	9	10
EIGR	SID	METHOD	F1	F2	NE	ND			
	NORM	G	C						

The METHOD field selects the eigenvalue method from the following list:

AHOU	Automatic Householder method
HOU	Householder method
MHOU	Modified Householder method
SINV	Sturm modified inverse power method

The F1 field specifies the lowest frequency of interest in the eigenvalue extraction. The F2 field specifies the highest frequency of interest in the eigenvalue extraction. The units are cycles per unit time. The value for F1 must be greater than or equal to 0.0.

The ND field is used to specify the desired number of roots (for tracking methods) or eigenvectors (for transformation methods), beginning with F1.

The NORM field on the continuation entry is used to specify the method of eigenvector normalization. The choices are

MASS	Mass normalization (default–if used, the continuation entry is not required).
MAX	Normalization to a unit value of the largest component in the analysis set.
POINT	Normalization to a unit value at a user-specified a-set grid point G and component C.

There is an interrelationship among the F1, F2, and ND fields on the EIGR entry as defined in Table 3-3.

Table 3-3. Relationship Between the METHOD Field and Other Fields

Field	METHOD Field	
	SINV	**HOU, MHOU, or AHOU**
F1, F2	Frequency range of interest. F1 must be input. If METHOD = "SINV" and ND is blank, then F2 must be input. (Real ≥ 0.0)	Frequency range of interest. If ND is not blank, F1 and F2 are ignored. If ND is blank, eigenvectors are found whose natural frequencies lie in the range between F1 and F2. (Real ≥ 0.0; F1 < F2)
NE	Estimate of number of roots in range. Not used by "SINV" method. (Integer > 0)	Not used
ND	Desired number of roots. If this field is blank and METHOD = "SINV", then all roots between F1 and F2 are searched, and the limit is 600 roots.	Desired number of eigenvectors. If ND is zero, the number of eigenvectors is determined from F1 and F2. If all three are blank, then ND is automatically set to one more than the number of degrees-of-freedom listed on SUPORT entries. (Integer ≥ 0; Default = 0)

The rules for METHOD = HOU, MHOU, and AHOU are identical. If any of these methods are selected, NX Nastran finds all of the eigenvalues but only computes the eigenvectors specified by F1 and F2 or those specified by ND (the desired number). F1 and F2 specify the lower and upper bounds of the frequency range in which eigenvectors are computed, and ND specifies the number of eigenvectors, beginning with the lowest (or the first rigid-body mode, if present). If F1, F2, and ND entries are present, ND takes precedence.

If METHOD = SINV, the values of F1, F2, and ND determine the number of eigenvalues and eigenvectors that are computed. These entries also provide hints to help NX Nastran find the eigenvalues. F1 and F2 specify the frequency range of interest within which NX Nastran searches for modes. NX Nastran attempts to find all of the modes in the range between F1 and F2 or the number specified by ND, whichever is less. If searching stops because ND modes are found, there is no guarantee that they are the lowest eigenvalues. If ND modes are not found in the range of interest, SINV usually finds one mode (or possibly more) outside the range F1 and F2 before stopping the search.

The SINV method is particularly efficient when only a small number of eigenvalues and eigenvectors are to be computed. Often only the lowest mode is of interest. The following example illustrates an EIGR entry which extracts only the lowest nonzero eigenvalue.

1	2	3	4	5	6	7	8	9	10
EIGR	13	SINV	0.0	0.01		1			

It is assumed in the example above that the frequency of the lowest mode is greater than 0.01 cycles per unit time. NX Nastran finds one eigenvalue outside the range F1, F2, and then stops the search. The eigenvalue found is the lowest nonzero eigenvalue (or a member of the lowest closely spaced cluster of eigenvalues in cases with close roots) provided that there are no negative eigenvalues and that the SUPORT entry has been used to specify the correct number of zero eigenvalues (see "Rigid-body Modes")

The following examples demonstrate the use of the EIGR data entry.

1	2	3	4	5	6	7	8	9	10
EIGR	1	AHOU				10			

In this example, the automatic Householder method is selected, and the lowest 10 modes are requested. Since the default MASS eigenvector normalization is requested, no continuation entry is needed.

EIGR	2	AHOU		100.					
	MAX								

In this example, the same method is requested, but all the modes below 100 cycles per unit time are requested with MAX vector normalization.

EIGR	3	SINV	0.1	100.0		6			
	POINT	32	3						

In this example, the Sturm modified inverse power method is requested for the first six modes found in the range specified (0.1 to 100 Hz). The POINT normalization method is requested with each eigenvector scaled such that grid point 32 in the T3 direction has a magnitude of 1.0. Note that this degree-of-freedom must be in the a-set.

3.7 Solution Control for Normal Modes Analysis

This section describes input required for the selection and control of normal modes analysis.

Executive Control Section

You can run a normal modes analysis, as an independent solution, in SOL 103 of the Structured Solution Sequences. The Executive Control Section can also contain diagnostic DIAG16, which prints the iteration information used in the SINV method. You may also run a normal modes analysis as part of the other solution sequences (such as modal transient response, modal frequency response, design optimization, and aeroelasticity).

Case Control Section

The most important eigenanalysis command in the Case Control is the METHOD command. This command is required. The set identification number specified by the METHOD Case Control command refers to the set identification number of an EIGR or EIGRL entry in the Bulk Data.

When you perform a modal analysis, the NX Nastran output file contains various diagnostic messages and an eigenvalue analysis summary. Optional grid and element output are available using standard Case Control output requests. Eigenvectors are printed only if a DISPLACEMENT or VECTOR command is included. These requests are summarized in Table 3-4.

Table 3-4. Eigenvalue Extraction Output Requests	
Grid Output	
DISPLACEMENT (or VECTOR)	Requests the eigenvector (mode shape) for a set of grid points.
GPFORCE	Requests the grid point force balance table to be computed for each mode for a set of grid points.
GPSTRESS	Requests grid point stresses to be computed for a set of grid points. This request must be accompanied by the ELSTRESS Case Control re quest and the definition of stress surfaces and/or stress volumes in the OUTPUT(POST) section of the Case Control.
SPCFORCE	Requests forces of single-point constraint to be computed for a set of grid points for each mode.
Element Output	
ELSTRESS (or STRESS)	Requests the computation of modal stresses for a set of elements for each mode.
ESE	Requests the computation of modal element strain energies for a set of elements for each mode.
ELFORCE (or FORCE)	Requests the computation of modal element forces for a set of elements for each mode.
STRAIN	Requests the computation of modal element strains for a set of elements.
Miscellaneous	
MODES	A special Case Control request that permits selective output requests to be processed on selective modes.

Bulk Data Section

In addition to Bulk Data entries required to define the structural model, the only other required Bulk Data entry is the eigenvalue selection entry EIGR or EIGRL. The EIGR entry is used to select the modal extraction parameters for the inverse power, Sturm modified inverse power, Householder, modified Householder, and automatic Householder methods. The EIGRL entry is used to select the modal extraction parameters for the Lanczos method.

3.8   Examples

This section provides several normal-modes analysis examples showing the input and output. These examples are as follows:

Model	Number of Grid Points	Element Types	Output Requests	Analysis Method	Units
bd03two	3	CELAS2 CONM2	DISPLACEMENT SPCFORCE ELFORCE	AHOU	Metric
bd03bar1, bd03bar2	11	CBAR	DISPLACEMENT	SINV	Metric
bd03bkt	236	CQUAD4 CONM2 RBE2	DISPLACEMENT STRESS ESE MODES	Lanczos	English
bd03car	972	CQUAD4 CTRIA3 CELAS2	DISPLACEMENT ESE	Lanczos	English
bd03fix	8157	CHEXA CPENTA	DISPLACEMENT	Lanczos	English
bd03plt1, bd03plt2	81	CQUAD4	DISPLACEMENT	Lanczos	Metric
bd03dmi	7	CBAR DMIG	None	Lanczos	Metric

These examples are described in the sections that follow.

Two-DOF Model

This example is a restrained two-DOF model with two springs and two masses as illustrated in Figure 3-5.
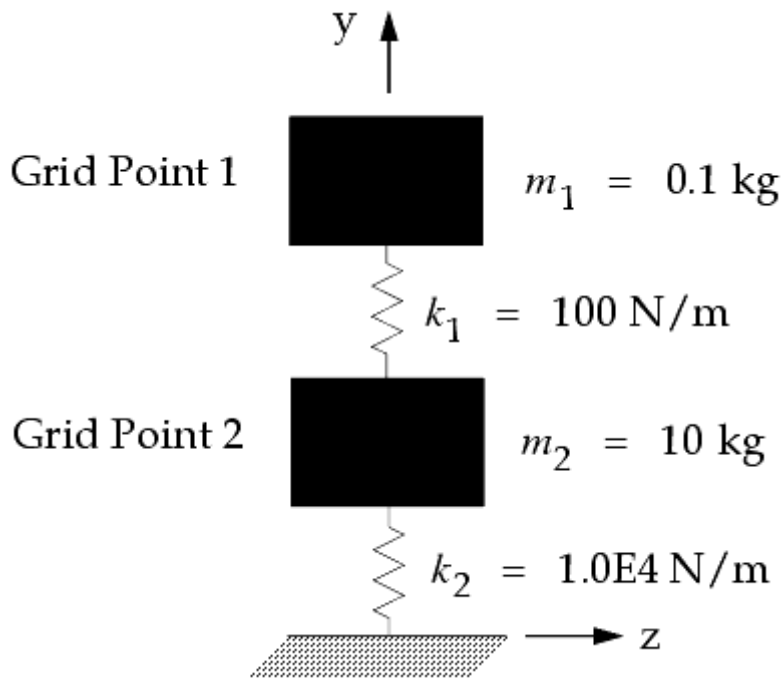
Figure 3-5. Two-DOF Model

The masses are constrained to deflect in only the y-direction. The example illustrates normal modes analysis (SOL 103) using automatic selection of the Householder or modified Householder method (METHOD = AHOU on the EIGR entry). The eigenvectors are normalized to the unit value of the largest displacement component (NORM = MAX on the EIGR entry). The input file is shown in .

```
$ FILE bd03two.dat
$
$ TWO DOF SYSTEM
$ CHAPTER 3, NORMAL MODES
$
TIME 5
SOL 103 $ NORMAL MODES ANALYSIS
CEND
$
TITLE = TWO DOF SYSTEM
SUBTITLE = NORMAL MODES ANALYSIS
$
$ SELECT SPC
SPC = 10
$
$ SELECT EIGR ENTRY
METHOD = 99
$
$ SELECT OUTPUT
SET 1 = 1,2
DISPLACEMENT = 1
SET 2 = 3
SPCFORCE = 2
SET 3 = 11,12
ELFORCE = 3
$
BEGIN BULK
$
$.......2.......3.......4.......5.......6.......7.......8.......9.......10.....$
$
$EIGR SID METHOD F1 F2 NE ND +EIG
$+EIG NORM G C
EIGR 99 AHOU 0. 200. +EIG1
+EIG1 MAX
$
GRID 1 0. 2. 0.
GRID 2 0. 1. 0.
GRID 3 0. 0. 0.
GRDSET 13456
CONM2 1 1 0.1
CONM2 2 2 10.0
```

```
CELAS2 11 100.0 1 2 2 2
CELAS2 12 1.0E4 2 2 3 2
SPC 10 3 2
$
ENDDATA
```

Figure 3-6. Input File for the Two-DOF Model

The printed output is shown in Figure 3-7. The eigenvalue summary lists the eigenvalue w^2_n, circular frequency w_n(radians per second), natural frequency f_n (cycles per second), generalized mass (see Equation 3-14), and generalized stiffness (see Equation 3-16) for each mode. The eigenvectors, SPC forces, and spring forces are shown for each mode.

```
                              R E A L   E I G E N V A L U E S
 MODE    EXTRACTION       EIGENVALUE          RADIANS           CYCLES          GENERALIZED        GENERALIZ
  NO.      ORDER                                                                   MASS            STIFFNES
   1         1         9.048751E+02       3.008114E+01      4.787562E+00       1.904875E-01       1.723674E
   2         2         1.105125E+03       3.324342E+01      5.290854E+00       2.105125E-01       2.326426E

    EIGENVALUE =  9.048751E+02
       CYCLES =  4.787562E+00       R E A L   E I G E N V E C T O R   N O .        1
    POINT ID.   TYPE          T1                T2              T3              R1                R2                R3
       1         G         0.0         1.000000E+00   0.0             0.0             0.0               0.0
       2         G         0.0         9.512492E-02   0.0             0.0             0.0               0.0


    EIGENVALUE =  1.105125E+03
       CYCLES =  5.290854E+00       R E A L   E I G E N V E C T O R   N O .        2

    POINT ID.   TYPE          T1                T2              T3              R1                R2                R3
       1         G         0.0         1.000000E+00   0.0             0.0             0.0               0.0
       2         G         0.0        -1.051249E-01   0.0             0.0             0.0               0.0


    EIGENVALUE =  9.048751E+02
                    F O R C E S   O F   S I N G L E - P O I N T   C O N S T R A I N T

    POINT ID.   TYPE          T1                T2              T3              R1                R2                R3
       3         G         0.0        -9.512491E+02   0.0             0.0             0.0               0.0


  EIGENVALUE =  1.105125E+03
                    F O R C E S   O F   S I N G L E - P O I N T   C O N S T R A I N T

    POINT ID.   TYPE          T1                T2              T3              R1                R2                R3
       3         G         0.0         1.051249E+03   0.0             0.0             0.0               0.0


    EIGENVALUE =  9.048751E+02
                    F O R C E S   I N   S C A L A R   S P R I N G S      ( C E L A S 2 )
    ELEMENT           FORCE            ELEMENT           FORCE            ELEMENT           FORCE            ELEMENT
     ID.                                ID.                               ID.                               ID.
      11       9.048751E+01             12       9.512491E+02


    EIGENVALUE =  1.105125E+03
                    F O R C E S   I N   S C A L A R   S P R I N G S      ( C E L A S 2 )
    ELEMENT           FORCE            ELEMENT           FORCE            ELEMENT           FORCE            ELEMENT
     ID.                                ID.                               ID.                               ID.
      11       1.105125E+02             12      -1.051249E+03
```

Figure 3-7. Output from the Two-DOF System

Cantilever Beam Model

This example is a fixed-free aluminum cantilever beam with properties as shown in Figure3-8.
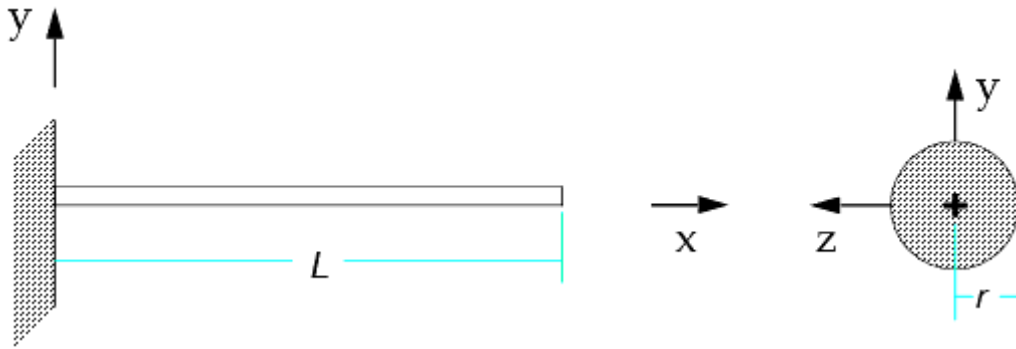
Figure 3-8. Cantilever Beam Model

L = 3.0m r = 0.014m J= 6.0E-8m^4

A = 6.158E-4m^2 $I1$ =$I2$ = 3.0E-8m^4 r_w= 2.65E4N/m^3

E = 7.1E10N/m^2 u=0.33

Nonstructural Weight =
2.414N/m

The r_w term is the weight density and must be converted to mass density r_m for consistency of units. PARAM,WTMASS is used to convert this weight density to mass density.

$$\text{WTMASS} = 1/g = 1/9.81 = 0.102 \text{ sec}^2/\text{m}$$ where g is the acceleration of gravity in

m/sec^2 . Therefore, $$\rho_m = \rho_w \cdot \text{WTMASS} = 2.65\text{E}4 \cdot (0.102) = 2703 \text{ kg/m}^3$$. The nonstructural weight of 2.414 N/m is added to the beam. This nonstructural weight per length is also scaled by PARAM,WTMASS.

The example illustrates normal modes analysis (SOL 103) using the Sturm modified inverse power method (METHOD = SINV on the EIGR entry). Mass normalization (the default) is chosen for the eigenvectors. All frequencies between 0 and 50 Hz are requested. Two models are run. In the first model, manufacturing tolerances make the cross section slightly out of round, making I1 and I2 slightly different. In the second model, the cross section is perfectly round, making I1 and I2 identical.

Consider the first model. Due to the manufacturing tolerances, I1 = 2.9E-8m^4 and I2 = 3.1E-8m^4. The input file is shown in Figure 3-9.

```
$ FILE bd03bar1.dat
$
$ CANTILEVER BEAM MODEL
$ CHAPTER 3, NORMAL MODES
$
SOL 103 $ NORMAL MODES ANALYSIS
TIME 10
CEND
$
TITLE = CANTILEVER BEAM
SUBTITLE = NORMAL MODES
LABEL = MODEL 1 (I1 NE I2)
$
SPC = 1
$
$ OUTPUT REQUEST
DISPLACEMENT = ALL
$
$ SELECT EIGR ENTRY
METHOD = 10
$
BEGIN BULK
$
$.......2.......3.......4.......5.......6.......7.......8.......9.......10.....$
$
```

```
$EIGR SID METHOD F1 F2 NE ND +EIG
$+EIG NORM G C
EIGR 10 SINV 0. 50.
$
$ ALUMINUM PROPERTIES:
$ E = 7.1E10 N/m**2, NU = 0.33, RHO = 2.65E4 N/m**3 (W E I G H T DENSITY)
$MAT1 MID E G NU RHO
MAT1 1 7.1+10 0.33 2.65+4
$
$ CONVERT WEIGHT TO MASS: MASS = (1/G)*WEIGHT
$ G = 9.81 m/sec**2 --> WTMASS = 1/G = 0.102
PARAM WTMASS 0.102
$
$ I1 AND I2 SLIGHTLY DIFFERENT DUE TO MANUFACTURING TOLERANCE
$ ADD NONSTRUCTURAL WEIGHT OF 2.414 N/M
PBAR 1 1 6.158-4 2.9-8 3.1-8 6.-8 2.414
$
$.......2.......3.......4.......5.......6.......7.......8.......9.......10.....$
$
CBAR 1 1 1 2 0. 1. 0.
CBAR 2 1 2 3 0. 1. 0.
CBAR 3 1 3 4 0. 1. 0.
CBAR 4 1 4 5 0. 1. 0.
CBAR 5 1 5 6 0. 1. 0.
CBAR 6 1 6 7 0. 1. 0.
CBAR 7 1 7 8 0. 1. 0.
CBAR 8 1 8 9 0. 1. 0.
CBAR 9 1 9 10 0. 1. 0.
CBAR 10 1 10 11 0. 1. 0.
GRID 1 0.0 0. 0.
GRID 2 0.3 0. 0.
GRID 3 0.6 0. 0.
GRID 4 0.9 0. 0.
GRID 5 1.2 0. 0.
GRID 6 1.5 0. 0.
GRID 7 1.8 0. 0.
GRID 8 2.1 0. 0.
GRID 9 2.4 0. 0.
GRID 10 2.7 0. 0.
GRID 11 3.0 0. 0.
SPC1 1 123456 1
$
ENDDATA
```

Figure 3-9. Input File for the First Beam Model

The first two resulting y-direction modes are illustrated in Figure 3-10. Displacements in the y-direction displacements are controlled by the I1 term. Because the structure is also free to displace in the z-direction, similar modes occur in that direction and are controlled by the I2 term.
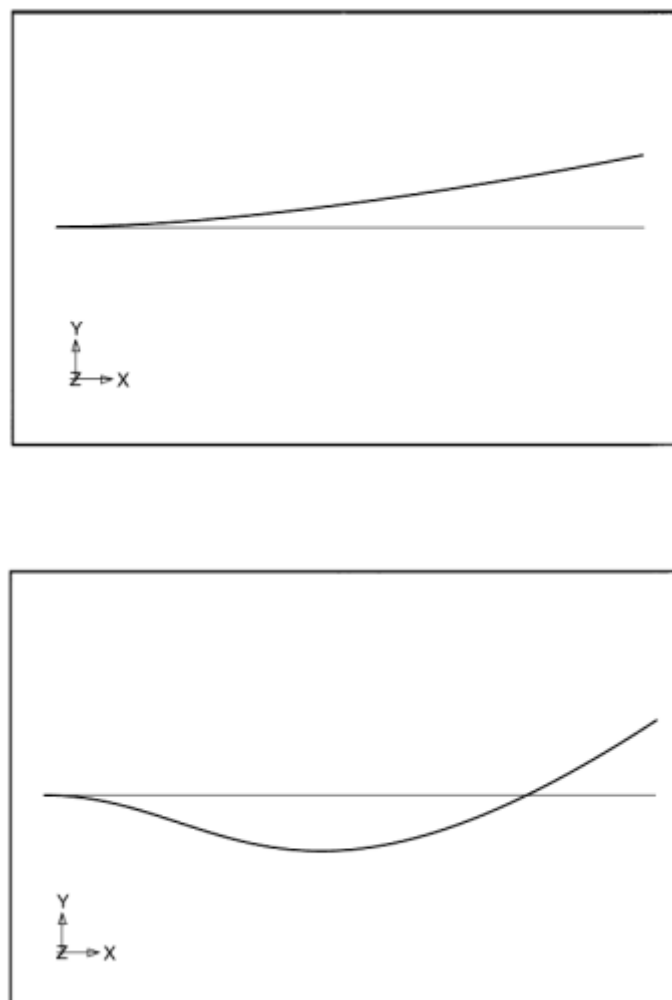
Figure 3-10. First Two Mode Shapes in the Y-Direction

Printed output is shown in Figure 3-11. Note that modes 1 and 3 are y-direction (T2) modes and modes 2 and 4 are z-direction (T3) modes.

```
                   E I G E N V A L U E   A N A L Y S I S   S U M M A R Y   (STURM INVERSE POWER)
                           NUMBER OF EIGENVALUES EXTRACTED . . . . . .       6
                           NUMBER OF TRIANGULAR DECOMPOSITIONS . . . .       9
                           TOTAL NUMBER OF VECTOR ITERATIONS . . . . .      59
                           REASON FOR TERMINATION:  ALL EIGENVALUES FOUND IN RANGE.

                                          R E A L   E I G E N V A L U E S
        MODE      EXTRACTION      EIGENVALUE          RADIANS             CYCLES           GENERALIZED      GENERALIZE
        NO.         ORDER                                                                     MASS           STIFFNESS
          1           1         1.629657E+02       1.276580E+01       2.031740E+00        1.000000E+00      1.629657E+
          2           2         1.742047E+02       1.319866E+01       2.100632E+00        1.000000E+00      1.742047E+
          3           3         6.258656E+03       7.911166E+01       1.259101E+01        1.000000E+00      6.258656E+
          4           4         6.690287E+03       8.179417E+01       1.301795E+01        1.000000E+00      6.690287E+
          5           6         4.809111E+04       2.192968E+02       3.490218E+01        1.000000E+00      4.809111E+
          6           5         5.140773E+04       2.267327E+02       3.608563E+01        1.000000E+00      5.140773E+

        EIGENVALUE =  1.629657E+02
           CYCLES =  2.031740E+00      R E A L   E I G E N V E C T O R   N O .        1

        POINT ID.    TYPE          T1                T2               T3            R1            R2                R3
            1         G         0.0              0.0              0.0             0.0         0.0               0.0
            2         G         0.0              1.391872E-02     2.259793E-10    0.0        -1.470464E-09      9.057010E-02
            3         G         0.0              5.301210E-02     8.606855E-10    0.0        -2.725000E-09      1.678406E-01
            4         G         0.0              1.133067E-01     1.839608E-09    0.0        -3.765826E-09      2.319480E-01
            5         G         0.0              1.908986E-01     3.099362E-09    0.0        -4.598456E-09      2.832321E-01
            6         G         0.0              2.820258E-01     4.578870E-09    0.0        -5.232973E-09      3.223138E-01
            7         G         0.0              3.831632E-01     6.220901E-09    0.0        -5.685052E-09      3.501587E-01
            8         G         0.0              4.911331E-01     7.973862E-09    0.0        -5.976744E-09      3.681249E-01
            9         G         0.0              6.032288E-01     9.793808E-09    0.0        -6.137029E-09      3.779972E-01
```

```
       10    G      0.0          7.173455E-01  1.164657E-08   0.0           -6.202180E-09  3.820101E-01
       11    G      0.0          8.321185E-01  1.350998E-08   0.0           -6.215970E-09  3.828595E-01

EIGENVALUE =   1.742047E+02
    CYCLES =   2.100632E+00       R E A L   E I G E N V E C T O R   N O .        2

POINT ID.   TYPE        T1            T2            T3            R1            R2            R3
        1    G      0.0           0.0           0.0           0.0           0.0           0.0
        2    G     -1.363453E-33  2.249974E-10 -1.391872E-02  0.0           9.057010E-02  1.464532E-09
        3    G     -2.693333E-33  8.574880E-10 -5.301210E-02  0.0           1.678406E-01  2.716750E-09
        4    G     -3.956893E-33  1.834005E-09 -1.133067E-01  0.0           2.319480E-01  3.758543E-09
        5    G     -5.123021E-33  3.092054E-09 -1.908986E-01  0.0           2.832321E-01  4.594735E-09
        6    G     -6.163003E-33  4.571193E-09 -2.820258E-01  0.0           3.223138E-01  5.234397E-09
        7    G     -7.051232E-33  6.214489E-09 -3.831632E-01  0.0           3.501587E-01  5.692031E-09
        8    G     -7.765834E-33  7.970303E-09 -4.911331E-01  0.0           3.681249E-01  5.988579E-09
        9    G     -8.289217E-33  9.794348E-09 -6.032288E-01  0.0           3.779972E-01  6.152242E-09
       10    G     -8.608492E-33  1.165196E-08 -7.173455E-01  0.0           3.820101E-01  6.219055E-09
       11    G     -8.715798E-33  1.352052E-08 -8.321185E-01  0.0           3.828595E-01  6.233260E-09

EIGENVALUE =   6.258656E+03
    CYCLES =   1.259101E+01       R E A L   E I G E N V E C T O R   N O .        3

POINT ID.   TYPE        T1            T2            T3            R1            R2            R3
        1    G      0.0           0.0           0.0           0.0           0.0           0.0
        2    G      0.0          -7.568120E-02 -9.828895E-08   0.0           5.937006E-07 -4.571418E-01
        3    G      0.0          -2.464387E-01 -3.200556E-07   0.0           8.257535E-07 -6.358200E-01
        4    G      0.0          -4.318525E-01 -5.608564E-07   0.0           7.289605E-07 -5.612913E-01
        5    G      0.0          -5.632111E-01 -7.314543E-07   0.0           3.723918E-07 -2.867391E-01
        6    G      0.0          -5.916957E-01 -7.684467E-07   0.0          -1.426249E-07  1.098159E-01
        7    G      0.0          -4.941767E-01 -6.417950E-07   0.0          -6.984853E-07  5.378201E-01
        8    G      0.0          -2.743728E-01 -3.563287E-07   0.0          -1.184615E-06  9.121327E-01
        9    G      0.0           4.170797E-02  5.417504E-08   0.0          -1.522753E-06  1.172494E+00
       10    G      0.0           4.159041E-01  5.401546E-07   0.0          -1.689206E-06  1.300660E+00
       11    G      0.0           8.124724E-01  1.055190E-06   0.0          -1.730573E-06  1.332512E+00

EIGENVALUE =   6.690287E+03
    CYCLES =   1.301795E+01       R E A L   E I G E N V E C T O R   N O .        4

POINT ID.   TYPE        T1            T2            T3            R1            R2            R3
        1    G      0.0           0.0           0.0           0.0           0.0           0.0
        2    G      3.698388E-22  1.207686E-08  7.568131E-02  0.0          -4.571424E-01  1.427123E-08
        3    G      7.305710E-22 -2.529794E-08  2.464390E-01  0.0          -6.358204E-01 -3.100643E-07
        4    G      1.073314E-21 -1.866785E-07  4.318528E-01  0.0          -5.612913E-01 -7.615683E-07
        5    G      1.389628E-21 -4.615435E-07  5.632114E-01  0.0          -2.867385E-01 -1.006117E-06
        6    G      1.671725E-21 -7.409781E-07  5.916957E-01  0.0           1.098168E-01 -7.569575E-07
        7    G      1.912658E-21 -8.602044E-07  4.941765E-01  0.0           5.378208E-01  4.733732E-08
        8    G      2.106496E-21 -6.796384E-07  2.743725E-01  0.0           9.121327E-01  1.181005E-06
        9    G      2.248464E-21 -1.591279E-07 -4.170818E-02  0.0           1.172493E+00  2.239405E-06
       10    G      2.335068E-21  6.217935E-07 -4.159040E-01  0.0           1.300659E+00  2.875692E-06
       11    G      2.364175E-21  1.520816E-06 -8.124720E-01  0.0           1.332511E+00  3.057266E-06
```

Figure 3-11. Printed Results from the First Model

Now, consider the second model for which I1 and I2 are identical. Printed output is shown in
Figure 3-12. Note that modes 1 and 2; 3 and 4; 5 and 6; etc., have identical frequencies; this is
a case of repeated roots. Also note that the eigenvectors are not pure y- or pure z-translation
(as they were in the first model); the eigenvectors are linear combinations of the y and z modes
since this model has repeated roots.

```
          E I G E N V A L U E  A N A L Y S I S   S U M M A R Y   (STURM INVERSE POWER)
                         NUMBER OF EIGENVALUES EXTRACTED . . . . . .      8
                         NUMBER OF TRIANGULAR DECOMPOSITIONS . . . .      6
                         TOTAL NUMBER OF VECTOR ITERATIONS . . . . .     68
                         REASON FOR TERMINATION:  ALL EIGENVALUES FOUND IN RANGE.

                                   R E A L   E I G E N V A L U E S
MODE    EXTRACTION      EIGENVALUE        RADIANS           CYCLES          GENERALIZED      GENERALIZ
 NO.      ORDER                                                               MASS           STIFFNES
   1        1         1.685851E+02     1.298403E+01      2.066473E+00      1.000000E+00      1.685851E
   2        3         1.685851E+02     1.298403E+01      2.066473E+00      1.000000E+00      1.685851E
   3        2         6.474471E+03     8.046410E+01      1.280626E+01      1.000000E+00      6.474471E
   4        5         6.474471E+03     8.046410E+01      1.280626E+01      1.000000E+00      6.474471E
   5        4         4.974941E+04     2.230458E+02      3.549883E+01      1.000000E+00      4.974941E
   6        6         4.974941E+04     2.230458E+02      3.549883E+01      1.000000E+00      4.974941E
   7        7         1.870792E+05     4.325266E+02      6.883875E+01      1.000000E+00      1.870792E
   8        8         1.870792E+05     4.325266E+02      6.883875E+01      1.000000E+00      1.870792E
EIGENVALUE =   1.685851E+02
    CYCLES =   2.066473E+00       R E A L   E I G E N V E C T O R   N O .        1

POINT ID.   TYPE        T1            T2            T3            R1            R2            R3
        1    G      0.0           0.0           0.0           0.0           0.0           0.0
        2    G      4.198658E-20  9.109908E-03  1.052335E-02  0.0          -6.847622E-02  5.927882E-02
        3    G      8.293447E-20  3.469681E-02  4.008020E-02  0.0          -1.268971E-01  1.098529E-01
        4    G      1.218317E-19  7.416008E-02  8.566642E-02  0.0          -1.753660E-01  1.518116E-01
        5    G      1.577188E-19  1.249444E-01  1.443303E-01  0.0          -2.141397E-01  1.853772E-01
```

```
         6     G     1.897125E-19    1.845877E-01   2.132276E-01   0.0       -2.436876E-01   2.109563E-01
         7     G     2.170278E-19    2.507827E-01   2.896933E-01   0.0       -2.647398E-01   2.291808E-01
         8     G     2.389961E-19    3.214496E-01   3.713246E-01   0.0       -2.783232E-01   2.409396E-01
         9     G     2.550813E-19    3.948168E-01   4.560753E-01   0.0       -2.857873E-01   2.474010E-01
        10     G     2.648916E-19    4.695067E-01   5.423540E-01   0.0       -2.888212E-01   2.500274E-01
        11     G     2.681884E-19    5.446261E-01   6.291288E-01   0.0       -2.894634E-01   2.505833E-01
 EIGENVALUE =  1.685851E+02
      CYCLES =  2.066473E+00        R E A L   E I G E N V E C T O R   N O .        2

 POINT ID.   TYPE        T1              T2             T3           R1           R2              R3
         1     G     0.0             0.0            0.0            0.0        0.0             0.0
         2     G     1.219015E-33   -1.052334E-02   9.109882E-03   0.0       -5.927866E-02   -6.847615E-02
         3     G     2.407892E-33   -4.008016E-02   3.469673E-02   0.0       -1.098526E-01   -1.268970E-01
         4     G     3.537263E-33   -8.566635E-02   7.415994E-02   0.0       -1.518114E-01   -1.753659E-01
         5     G     4.579275E-33   -1.443302E-01   1.249442E-01   0.0       -1.853771E-01   -2.141396E-01
         6     G     5.508284E-33   -2.132276E-01   1.845875E-01   0.0       -2.109563E-01   -2.436876E-01
         7     G     6.301480E-33   -2.896932E-01   2.507825E-01   0.0       -2.291809E-01   -2.647399E-01
         8     G     6.939436E-33   -3.713246E-01   3.214495E-01   0.0       -2.409399E-01   -2.783234E-01
         9     G     7.406566E-33   -4.560753E-01   3.948168E-01   0.0       -2.474014E-01   -2.857874E-01
        10     G     7.691474E-33   -5.423541E-01   4.695069E-01   0.0       -2.500278E-01   -2.888214E-01
        11     G     7.787220E-33   -6.291289E-01   5.446264E-01   0.0       -2.505838E-01   -2.894636E-01
 EIGENVALUE =  6.474471E+03
      CYCLES =  1.280626E+01        R E A L   E I G E N V E C T O R   N O .        3

 POINT ID.   TYPE        T1              T2             T3           R1           R2              R3
         1     G     0.0             0.0            0.0            0.0        0.0             0.0
         2     G    -1.606174E-26    6.893317E-02   3.123881E-02   0.0       -1.886937E-01    4.163812E-01
         3     G    -3.172614E-26    2.244652E-01   1.017221E-01   0.0       -2.624463E-01    5.791277E-01
         4     G    -4.660607E-26    3.933468E-01   1.782550E-01   0.0       -2.316833E-01    5.112443E-01
         5     G    -6.033448E-26    5.129929E-01   2.324757E-01   0.0       -1.183568E-01    2.611722E-01
         6     G    -7.257351E-26    5.389377E-01   2.442332E-01   0.0        4.532863E-02   -1.000244E-01
         7     G    -8.302282E-26    4.501138E-01   2.039804E-01   0.0        2.219953E-01   -4.898660E-01
         8     G    -9.142668E-26    2.499085E-01   1.132523E-01   0.0        3.764997E-01   -8.308035E-01
         9     G    -9.757999E-26   -3.798927E-02  -1.721591E-02   0.0        4.839685E-01   -1.067950E+00
        10     G    -1.013329E-25   -3.788206E-01  -1.716723E-01   0.0        5.368715E-01   -1.184688E+00
        11     G    -1.025940E-25   -7.400293E-01  -3.353632E-01   0.0        5.500190E-01   -1.213700E+00
 EIGENVALUE =  6.474471E+03
      CYCLES =  1.280626E+01        R E A L   E I G E N V E C T O R   N O .        4

 POINT ID.   TYPE        T1              T2             T3           R1           R2              R3
         1     G     0.0             0.0            0.0            0.0        0.0             0.0
         2     G    -6.181717E-19    3.123822E-02  -6.893279E-02   0.0        4.163791E-01    1.886905E-01
         3     G    -1.221121E-18    1.017205E-01  -2.244643E-01   0.0        5.791260E-01    2.624437E-01
         4     G    -1.794002E-18    1.782531E-01  -3.933455E-01   0.0        5.112445E-01    2.316835E-01
         5     G    -2.322707E-18    2.324743E-01  -5.129921E-01   0.0        2.611744E-01    1.183602E-01
         6     G    -2.794219E-18    2.442331E-01  -5.389376E-01   0.0       -1.000213E-01   -4.532376E-02
         7     G    -3.196926E-18    2.039817E-01  -4.501146E-01   0.0       -4.898637E-01   -2.219917E-01
         8     G    -3.520915E-18    1.132542E-01  -2.499097E-01   0.0       -8.308033E-01   -3.764996E-01
         9     G    -3.758207E-18   -1.721460E-02   3.798841E-02   0.0       -1.067952E+00   -4.839722E-01
        10     G    -3.902959E-18   -1.716725E-01   3.788207E-01   0.0       -1.184692E+00   -5.368777E-01
        11     G    -3.951610E-18   -3.353654E-01   7.400307E-01   0.0       -1.213704E+00   -5.500259E-01
```

Figure 3-12. Printed Results from the Second Model

This second model was rerun, changing the mass from lumped (the default) to coupled by adding
PARAM,COUPMASS,1 to the Bulk Data. The resulting frequencies are shown below and are
compared to those of the lumped mass model and the theoretical results. Note that the frequency
difference is greater at higher frequencies. For most production-type models (i.e., complex
three-dimensional structures), this difference is negligible.

Frequencies (Hz)		
Theory	**Lumped Mass Model**	**Coupled Mass Model**
2.076	2.066	2.076
13.010	12.806	13.010
36.428	35.499	36.437
71.384	68.838	71.451

Bracket Model

This example is a steel bracket as shown in Figure 3-13.
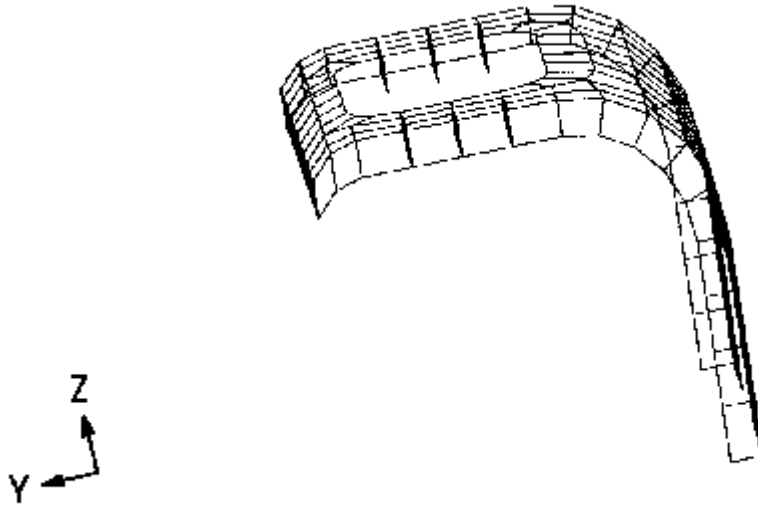
Figure 3-13. Bracket Model

A concentrated mass is suspended from the center of the hole in bracket. This mass has the following properties:

$$m = 0.0906 \text{ lb-sec}^2/\text{in}$$

$$I11 = 0.35 \text{ in-lb-sec}^2$$

$$I22 = 0.56 \text{ in-lb-sec}^2$$

$$I33 = 0.07 \text{ in-lb-sec}^2$$

The concentrated mass (grid point 999) is connected to the bracket by an RBE2 element connecting 24 grid points, as shown in Figure 3-14.
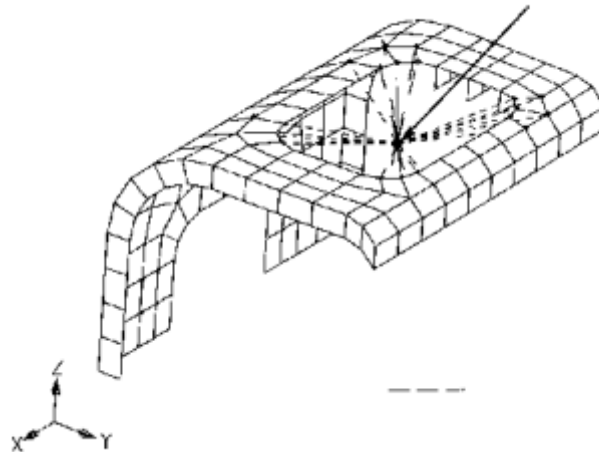
Figure 3-14. Bracket Model Showing RBE2 Element (Dashed Lines)

The bracket is clamped by constraining six degrees-of-freedom for each of 12 grid points near the base.

This example illustrates a normal modes analysis (SOL 103) using the Lanczos method (EIGRL entry). All frequencies below 100 Hz are requested. The MODES Case Control command is used to specify the number of times a subcase is repeated and therefore enables different output requests for each mode. The output requests for this problem are eigenvectors for all modes (DISPLACEMENT = ALL above the subcase level), corner stresses for the first two modes (STRESS(CORNER) = ALL and MODES = 2 in Subcase 1), and element strain energies for the third mode (ESE = ALL). An abridged version of the input file is shown in Figure 3-15.

```
$   FILE bd03bkt.dat
$
$   BRACKET MODEL
$   CHAPTER 3, NORMAL MODES
$
TIME 10
SOL 103      $  NORMAL MODES ANALYSIS
CEND
$
TITLE =  BRACKET MODEL
SUBTITLE = NORMAL MODES ANALYSIS
$
SPC   = 1
$
$   SELECT EIGRL
METHOD = 777
$
$   OUTPUT REQUESTS
DISPLACEMENT = ALL
SUBCASE 1
  MODES = 2  $ USE FOR FIRST TWO MODES
  STRESS(CORNER) = ALL
SUBCASE 3
  ESE = ALL
$
BEGIN BULK
$
$.......2.......3.......4.......5.......6.......7.......8.......9.......10.....$
$
$EIGRL  SID     V1      V2      ND      MSGLVL  MAXSET  SHFSCL  NORM
EIGRL   777             100.
$
$   CONCENTRATED MASS, SUPPORTED WITH AN RBE2 ELEMENT
$
GRID    999             3.0     3.0     3.8
$
$CONM2  EID     G       CID     M       X1      X2      X3              +CON1
$+CON1  I11     I21     I22     I31     I32     I33
CONM2   999     999             0.0906                                  +CM1
+CM1    0.35            0.56                    0.07
$
RBE2    999     999     123456  126     127     91      81      80      +RB1
```

```
+RB1      90       95      129      128       96       86       85      228      +RB2
+RB2     229      199      189      190      200      195      227      226      +RB3
+RB3     194      184      185
$
$  STEEL,  M A S S  DENSITY FOR RHO
MAT1     1        3.+7    1.153+7            7.76-4
$
... basic model ...
$
ENDDATA
```

Figure 3-15. Abridged Input File for the Bracket Model

Figure 3-16 shows an abridged version of the resulting NX Nastran output. The circular total element strain energy (ESE) for each mode of the entire model is ESE = $w^2_i/2$ for the i-th mode when $f^T_i M f_i$ = 1. The frequency of the third mode is 234.49 radians per second squared; therefore, the total strain energy of the model is ESE_{total} = $(234.49)^2/2$ = 27493 for the third mode. The printed ESE_{total} = 27492. The eigenvectors are printed for each mode, the element corner stresses are printed for the first and second modes, and the element strain energies are printed for the third mode. (Only the headers are shown in the figure in order to save space.)

```
               E I G E N V A L U E   A N A L Y S I S   S U M M A R Y   (LANCZOS ITERATION)
                       BLOCK SIZE USED ......................    6
                       NUMBER OF DECOMPOSITIONS .............    2
                       NUMBER OF ROOTS FOUND ................    3
                       NUMBER OF SOLVES REQUIRED ............    3
                       TERMINATION MESSAGE :   REQUIRED NUMBER OF EIGENVALUES FOUND.


                               R E A L   E I G E N V A L U E S
  MODE    EXTRACTION      EIGENVALUE          RADIANS              CYCLES          GENERALIZED      GENERALIZ
   NO.      ORDER                                                                     MASS          STIFFNES
    1         1         3.930304E+03        6.269214E+01        9.977763E+00       1.000000E+00     3.930304E
    2         2         2.878402E+04        1.696586E+02        2.700200E+01       1.000000E+00     2.878402E
    3         3         5.498442E+04        2.344876E+02        3.731985E+01       1.000000E+00     5.498442E


    EIGENVALUE =  3.930304E+03
        CYCLES =  9.977763E+00      R E A L   E I G E N V E C T O R   N O .        1

    EIGENVALUE =  2.878402E+04
        CYCLES =  2.700200E+01      R E A L   E I G E N V E C T O R   N O .        2

    EIGENVALUE =  5.498442E+04
        CYCLES =  3.731985E+01      R E A L   E I G E N V E C T O R   N O .        3

    EIGENVALUE =  3.930304E+03
              S T R E S S E S   I N   Q U A D R I L A T E R A L   E L E M E N T S   ( Q U A D 4 )      OPTIO

    EIGENVALUE =  2.878402E+04
              S T R E S S E S   I N   Q U A D R I L A T E R A L   E L E M E N T S   ( Q U A D 4 )      OPTIO


                            E L E M E N T   S T R A I N   E N E R G I E S

          ELEMENT-TYPE = QUAD4            * TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM   =  2.749221E+04
                MODE            3           TOTAL ENERGY OF ALL ELEMENTS IN SET    -1 =  2.749221E+04
                                                                                    *
                 TYPE = QUAD4    SUBTOTAL      2.749221E+04               100.0000
```

Figure 3-16. Abridged Output from the Bracket Model

The deformed shape resulting from the first mode is illustrated in Figure 3-17 and is overlaid on the undeformed shape. Figure3-18 illustrates the stress contours plotted on the deformed shape of the second mode. The element strain energy contour plot for the third mode is shown in Figure 3-19.
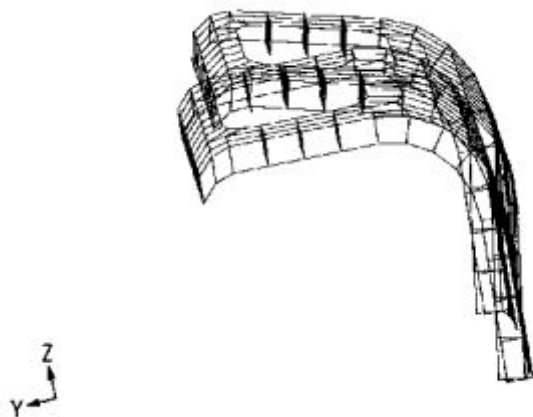
Figure 3-17. Deformed Shape of the First Mode
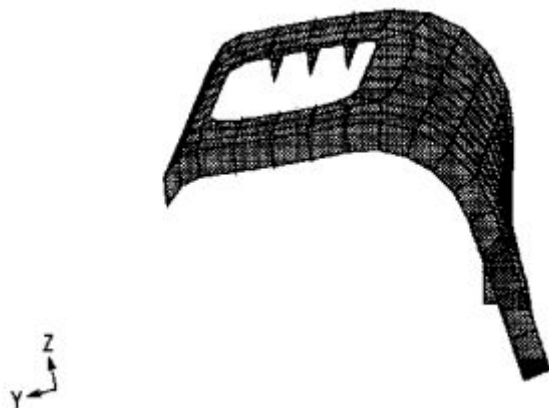
Figure 3-18. Second Mode Deformation with Element Stress Contours
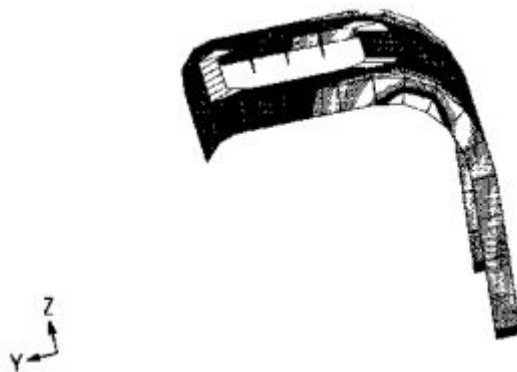
Figure 3-19. Element Strain Energy Contours for the Third Mode

Car Frame Model

Figure 3-20 shows a model of an aluminum car frame. The frame model is comprised of plate elements (CQUAD4 and CTRIA3), with springs (CELAS2) representing the suspension. Spring stiffnesses are input in the three translational directions; a stiffness of 500 lb/in is used in the vertical direction (T2), and stiffnesses of 1000 lb/in are used in the other translational directions (T1 and T3). When using CELASi elements to connect two grid points, it is recommended that the coordinates of the two grid points be identical in order to represent coaxial springs (noncoincident coordinates can lead to errors). The goal of the analysis is to compute resonant frequencies up to 50 Hz using the Lanczos method. Element strain energies are computed for the springs in order to help characterize the resulting modes.

Figure 3-21 shows the input file. Modal displacements are written to the plot file and are not printed. The rigid body mass matrix is computed via the PARAM,GRDPNT,0 entry. "Include" files are used to partition the input file into several smaller files. The INCLUDE statement inserts an external file into the input file. The basic file is bd03car.dat. The springs are contained in file bd03cars.dat (Figure 3-22), and the rest of the input file is contained in file bd03carb.dat (not shown).
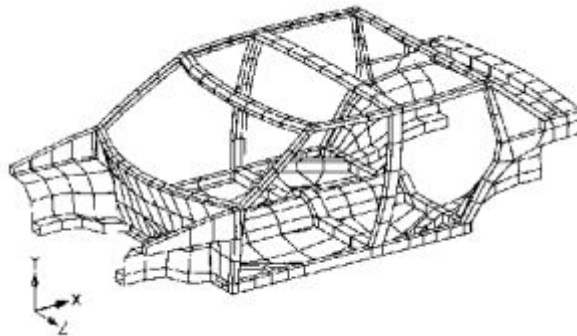
Figure 3-20. Car Frame Model

```
$ FILE bd03car.dat
$
$ CAR FRAME MODEL
$ CHAPTER 3, NORMAL MODES
$
$ MODEL COURTESY LAPCAD ENGINEERING
$ CHULA VISTA, CALIFORNIA
$
SOL 103 $ NORMAL MODES ANALYSIS
TIME 30
CEND
$
TITLE = CAR MODEL WITH SUSPENSION SPRINGS
SUBTITLE = MODAL ANALYSIS CASE CONTROL
ECHO = UNSORT
$
METHOD = 1
$
DISPLACEMENT(PLOT) = ALL
SET 99 = 1001,1002,1003,1011,1012,1013,
1021,1022,1023,1031,1032,1033
ESE = 99
$
BEGIN BULK
$
```

```
$.......2.......3.......4.......5.......6.......7.......8.......9.......10.....$
$
INCLUDE 'bd03cars.dat' $ Car springs
$
$EIGRL SID V1 V2 ND MSGLVL MAXSET SHFSCL NORM
EIGRL 1 -1. 50.
$
$MAT1 MID E G NU RHO
MAT1 1 1.0E7 0.33 2.59E-4
$
$ PRINT RIGID-BODY MASS
PARAM,GRDPNT,0
$
ECHOOFF
INCLUDE 'bd03carb.dat' $ Rest of Bulk Data
$
ENDDATA
```

Figure 3-21. Basic Input File for the Car Model

```
$ FILE bd03cars.dat
$
$ SPRINGS CONNECTED TO GROUND
$ GRIDS 1059,1562,1428,1895 HAVE THE SAME COORDS. AS 59,562,428,895
$
GRID 1059 152.012 32.7964 -2.90000 123456
GRID 1562 152.012 32.7964 -67.1000 123456
GRID 1428 35.6119 30.8257 -0.66667 123456
GRID 1895 35.6119 30.8257 -69.3333 123456
$
CELAS2 1001 1000. 59 1 1059 1
CELAS2 1002 500. 59 2 1059 2
CELAS2 1003 1000. 59 3 1059 3
CELAS2 1011 1000. 562 1 1562 1
CELAS2 1012 500. 562 2 1562 2
CELAS2 1013 1000. 562 3 1562 3
CELAS2 1021 1000. 428 1 1428 1
CELAS2 1022 500. 428 2 1428 2
CELAS2 1023 1000. 428 3 1428 3
CELAS2 1031 1000. 895 1 1895 1
CELAS2 1032 500. 895 2 1895 2
CELAS2 1033 1000. 895 3 1895 3
```

Figure 3-22. Input File for the Springs

Figure 3-23 shows the grid point weight generator output. The grid point weight generator indicates that the translational mass is 0.8027 lb-sec^2/in . This type of information is useful in verifying your model. Figure 3-24 shows some of the rest of the output. The eigenvalue analysis summary indicates that there are 12 modes below 50 Hz. The large element strain energies in the first six modes indicate that these are primarily suspension modes, comprised of the car frame acting as a rigid body supported by the flexible springs. Approximately 95% of the total strain energy is represented by spring deformation as opposed to frame deformation. Modes 7 and above show insignificant spring strain energy, indicating that these are primarily frame modes.

```
          O U T P U T   F R O M   G R I D   P O I N T   W E I G H T   G E N E R A T O R
                        REFERENCE POINT =        0
                                M O
    *  8.027376E-01  0.000000E+00  0.000000E+00  0.000000E+00 -2.808118E+01 -2.179473E+01 *
    *  0.000000E+00  8.027376E-01  0.000000E+00  2.808118E+01  0.000000E+00  7.610537E+01 *
    *  0.000000E+00  0.000000E+00  8.027376E-01  2.179473E+01 -7.610537E+01  0.000000E+00 *
    *  0.000000E+00  2.808118E+01  2.179473E+01  2.325008E+03 -2.153940E+03  2.662697E+03 *
    * -2.808118E+01  0.000000E+00 -7.610537E+01 -2.153940E+03  1.020870E+04  7.626398E+02 *
    * -2.179473E+01  7.610537E+01  0.000000E+00  2.662697E+03  7.626398E+02  9.437676E+03 *
                                 S
                    *  1.000000E+00  0.000000E+00  0.000000E+00 *
                    *  0.000000E+00  1.000000E+00  0.000000E+00 *
                    *  0.000000E+00  0.000000E+00  1.000000E+00 *
          DIRECTION
```

```
MASS AXIS SYSTEM (S)      MASS           X-C.G.          Y-C.G.          Z-C.G.
           X        8.027376E-01     0.000000E+00   2.715050E+01   -3.498177E+01
           Y        8.027376E-01     9.480728E+01   0.000000E+00   -3.498177E+01
           Z        8.027376E-01     9.480728E+01   2.715050E+01    0.000000E+00
                                         I(S)
                    *  7.509408E+02   8.764180E+01  -3.962963E-01 *
                    *  8.764180E+01   2.011031E+03  -2.217236E-01 *
                    * -3.962963E-01  -2.217236E-01   1.630595E+03 *
                                         I(Q)
                    *  2.017097E+03                                *
                    *                 7.448741E+02                 *
                    *                                1.630595E+03  *
                                          Q
                    *  6.905332E-02   9.976128E-01   5.012719E-04 *
                    * -9.976128E-01   6.905355E-02  -4.555999E-04 *
                    * -4.891269E-04  -4.686146E-04   9.999998E-01 *
```

Figure 3-23. Output from the Grid Point Weight Generator

```
        E I G E N V A L U E   A N A L Y S I S   S U M M A R Y   (LANCZOS ITERATION)
                        BLOCK SIZE USED ......................    7
                        NUMBER OF DECOMPOSITIONS .............    3
                        NUMBER OF ROOTS FOUND ................   12
                        NUMBER OF SOLVES REQUIRED ............   11
                        TERMINATION MESSAGE :  REQUIRED NUMBER OF EIGENVALUES FOUND.
```

```
                                   R E A L   E I G E N V A L U E S
   MODE    EXTRACTION      EIGENVALUE         RADIANS            CYCLES         GENERALIZED        GENERALIZ
    NO.      ORDER                                                                MASS              STIFFNES
     1         1         2.346479E+03      4.844047E+01      7.709540E+00      1.000000E+00        2.346479E
     2         2         2.654886E+03      5.152559E+01      8.200551E+00      1.000000E+00        2.654886E
     3         3         3.769821E+03      6.139887E+01      9.771934E+00      1.000000E+00        3.769821E
     4         4         4.633242E+03      6.806792E+01      1.083335E+01      1.000000E+00        4.633242E
     5         5         5.078395E+03      7.126286E+01      1.134184E+01      1.000000E+00        5.078395E
     6         6         8.485758E+03      9.211817E+01      1.466106E+01      1.000000E+00        8.485758E
     7         7         2.805541E+04      1.674975E+02      2.665805E+01      1.000000E+00        2.805541E
     8         8         5.350976E+04      2.313218E+02      3.681600E+01      1.000000E+00        5.350976E
     9         9         5.940912E+04      2.437399E+02      3.879240E+01      1.000000E+00        5.940912E
    10        10         8.476198E+04      2.911391E+02      4.633622E+01      1.000000E+00        8.476198E
    11        11         9.134271E+04      3.022296E+02      4.810133E+01      1.000000E+00        9.134271E
    12        12         9.726959E+04      3.118807E+02      4.963736E+01      1.000000E+00        9.726959E
```

```
             E L E M E N T   S T R A I N   E N E R G I E S

   ELEMENT-TYPE = ELAS2              * TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM    =   1.173240E+03
        MODE             1             TOTAL ENERGY OF ALL ELEMENTS IN SET    99 =   1.104569E+03
                                                                             *
                    ELEMENT-ID          STRAIN-ENERGY          PERCENT OF TOTAL    STRAIN-ENERGY-DE
                       1002            2.735009E+02               23.3116
                       1003            4.059090E-02                 .0035
                       1012            2.685884E+02               22.8929
                       1013            3.240471E-02                 .0028
                       1021            8.017746E-02                 .0068
                       1022            2.833448E+02               24.1506
                       1023            3.963123E-01                 .0338
                       1031            7.903841E-02                 .0067
                       1032            2.781467E+02               23.7076
                       1033            3.573737E-01                 .0305

      TYPE = ELAS2    SUBTOTAL       1.104569E+03               94.1469
```

```
             E L E M E N T   S T R A I N   E N E R G I E S

   ELEMENT-TYPE = ELAS2              * TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM    =   1.402770E+04
        MODE             7             TOTAL ENERGY OF ALL ELEMENTS IN SET    99 =   3.172818E+03
                                                                             *
                    ELEMENT-ID          STRAIN-ENERGY          PERCENT OF TOTAL    STRAIN-ENERGY-DE
                       1001            1.278503E+02                 .9114
                       1002            5.884620E+02                4.1950
                       1003            8.020268E+01                 .5717
                       1011            1.278420E+02                 .9114
                       1012            5.883779E+02                4.1944
                       1013            8.022697E+01                 .5719
                       1021            7.525866E+01                 .5365
                       1022            4.885996E+02                3.4831
                       1023            2.255234E+02                1.6077
                       1031            7.528390E+01                 .5367
                       1032            4.896509E+02                3.4906
                       1033            2.255398E+02                1.6078

      TYPE = ELAS2    SUBTOTAL       3.172818E+03               22.6182
```

Figure 3-24. Abridged Output from the Car Model

Mode shapes for modes 7, 8, 9, and 10 are shown in Figure 3-25. Mode 7 is an overall twisting mode; mode 8 is a "roof collapse" mode; mode 9 is a local (front) roof mode; and mode 10 is a local rear mode. Plots such as these, in conjunction with element strain energies, help to illustrate each of the mode shapes.
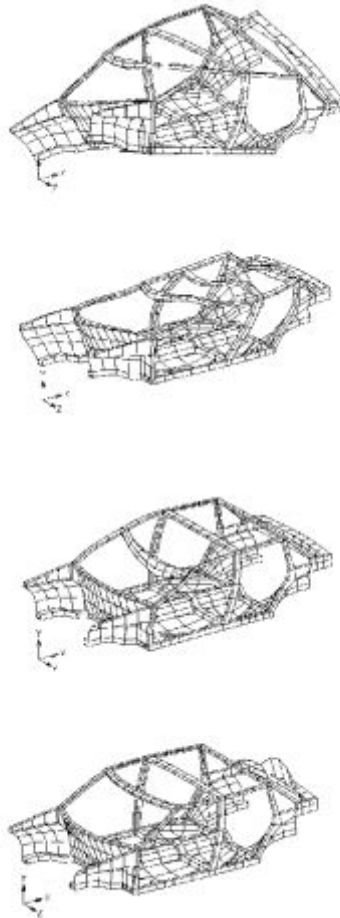
Figure 3-25. Mode Shapes for Modes 7, 8, 9, and 10

Test Fixture Model

This example is an aluminum test fixture, which is shown in Figure 3-26. The model is comprised of 8157 grid points, 5070 CHEXA elements, and 122 CPENTA elements. The primary plates are 1 inch thick, and the gusset plates are 0.5 inch thick. The base of the fixture is constrained to have no vertical (y) motion, and the bolt holes at the base are constrained to also have no horizontal (x and z) motion.
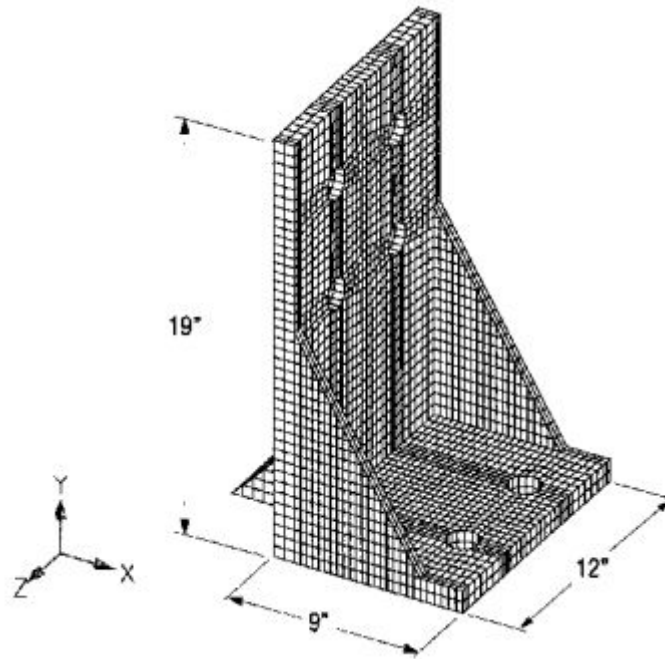
Figure 3-26. Test Fixture Model

A portion of the input file is shown in Figure 3-27. The Lanczos method is used to compute the modes. The first six modes are requested (ND is 6, with V1 and V2 blank). The GRDSET Bulk Data entry removes the rotational DOFs (456) from the analysis since the solid elements have no rotational stiffness. The Bulk Data is in free format.

```
$ FILE bd03fix.dat
$
$ TEST FIXTURE
$ CHAPTER 3, NORMAL MODES
$
TIME 240
SOL 103
CEND
ECHO = NONE
DISPLACEMENT(PLOT) = ALL
SPC = 1
$
METHOD = 1
$
BEGIN BULK
$
$GRDSET, ,CP, , , , CD, PS
GRDSET , , , , , , , , 456
$
$EIGRL, SID, V1, V2, ND, MSGLVL, MAXSET, SHFSCL, NORM
EIGRL , 1, , , 6
$
$MAT1, MID, E, G, NU, RHO
MAT1 , 3, 1.00E7, , 0.334, 2.5383-4
$
... basic model ...
$
ENDDATA
```

Figure 3-27. Abridged Input File for Test Fixture Model

Figure 3-28 shows the first four mode shapes. The first mode is a bending mode, the second and third modes are twist modes, and the fourth mode is a bending mode.
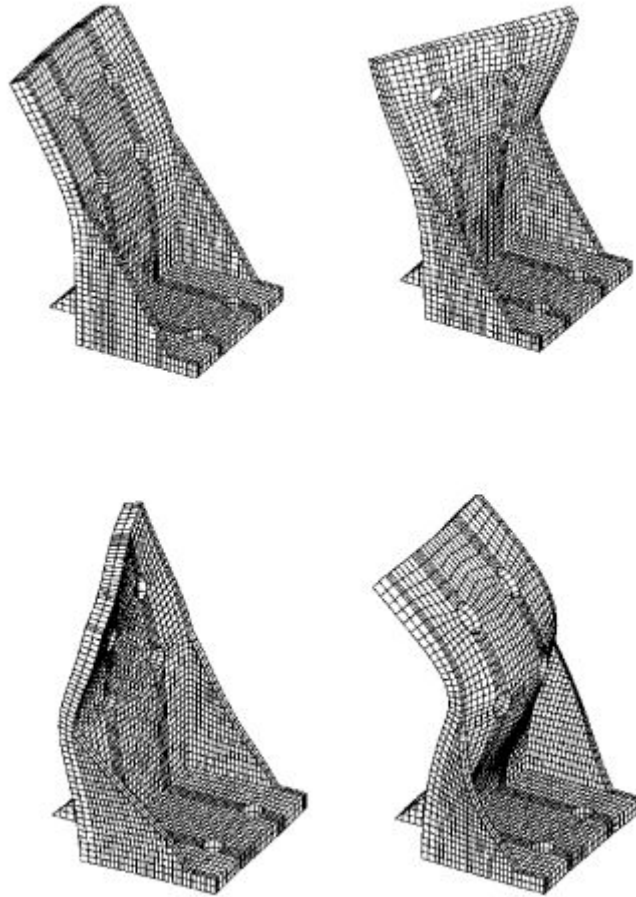
Figure 3-28. Test Fixture Mode Shapes

Quarter Plate Model

This example is a quarter model of a simply supported flat plate, shown in Figure 3-29. This example illustrates the use of multiple boundary conditions (new in Version 68) for modeling symmetric structures. In this case the plate is doubly symmetric.
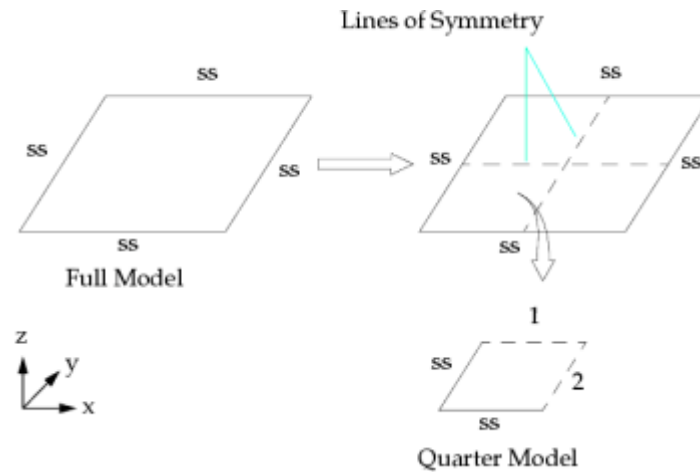
Figure 3-29. Derivation of Quarter Plate Model

A portion of the input file is shown in Figure 3-30. Four subcases are used—one for each of the following sets of boundary conditions for the quarter model:

- Symmetric-antisymmetric

- Antisymmetric-symmetric

- Symmetric-symmetric

- Antisymmetric-antisymmetric

The BC Case Control command identifies multiple boundary conditions. The SPCADD Bulk Data entry defines a union of SPC sets.

```
$ FILE bd03plt1.dat
$
$ QUARTER PLATE MODEL
$ CHAPTER 3, NORMAL MODES
$
SOL 103 $ NORMAL MODES ANALYSIS
TIME 10
CEND
TITLE = SIMPLY SUPPORTED PLATE USING SYMMETRY
SUBTITLE = NORMAL MODES CASE CONTROL
LABEL = QUARTER PLATE MODEL
$
DISPLACEMENT = ALL
$
SUBCASE 1
LABEL = SYM-ASYM
BC = 1
METHOD = 1
SPC = 101
SUBCASE 2
LABEL = ASYM-SYM
BC = 2
SPC = 102
METHOD = 1
SUBCASE 3
LABEL = SYM-SYM
BC = 3
SPC = 103
METHOD = 1
```

```
SUBCASE 4
LABEL = ASYM-ASYM
BC = 4
SPC = 104
METHOD = 1
BEGIN BULK
$
$.......2.......3.......4.......5.......6.......7.......8.......9.......10......
$
$ SYM-ASYM
SPCADD 101 11 1 4
$ ASYM-SYM
SPCADD 102 11 2 3
$ SYM-SYM
SPCADD 103 11 1 3
$ ASYM-ASYM
SPCADD 104 11 2 4
$
$EIGRL SID V1 V2
EIGRL 1 -0.1 100.
$
SPC 1 1 246 0.00
SPC 1 2 246 0.00
... etc. ...
SPC 11 80 12356 0.00
SPC 11 81 123456 0.00
$
... basic model ...
$
ENDDATA
```

Figure 3-30. Input File (Abridged) for the Quarter Plate Model

Figure 3-31 shows the quarter plate mode shapes and the corresponding mode shapes for a full model of the same structure. Note that the quarter plate modes match the full plate modes.
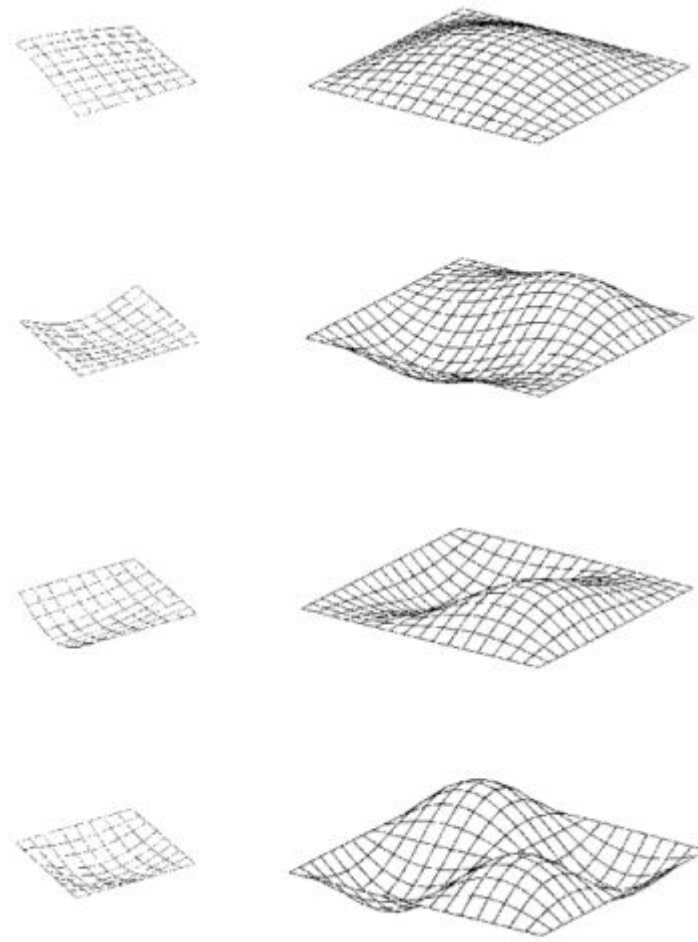
Figure 3-31. Mode Shapes for the Quarter Model (Left) and Full Model (Right)

DMIG Example

This example illustrates the use of a DMIG entry to input external mass and stiffness. The cantilever beam model shown in Figure 3-32 is used for this purpose. The model consists of CBAR elements 1 through 4. Element 5 is a model from another subcontractor that is input via DMIG entries. The model contains two DOFs (R2 and T3) per grid point.
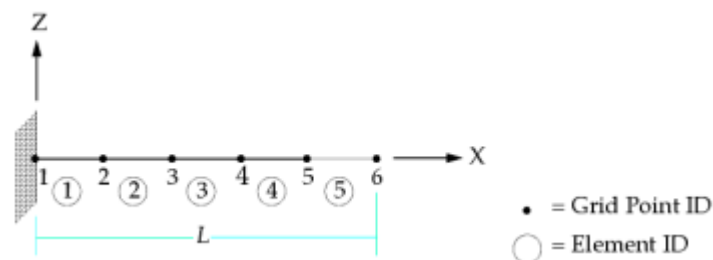
Figure 3-32. Planar Cantilever Beam

The stiffness and mass matrices from the contractor for element 5 are as follows:

$$
K_5 \;=\;
\begin{array}{c}
 \\
5,3 \\
5,5 \\
6,3 \\
6,5
\end{array}
\begin{array}{cccc}
5,3 & 5,5 & 6,3 & 6,5 \\
\left(\begin{array}{cccc}
500038.8 & & & \\
-250019.4 & 166679.6 & & \\
-500038.8 & 250019.4 & 500038.8 & \\
-250019.4 & 83339.8 & 250019.4 & 166679.6
\end{array}\right)
\end{array}
$$

Equation 3-19.

$$
M_5 \;=\;
\begin{array}{c}
 \\
5,3 \\
5,5 \\
6,3 \\
6,5
\end{array}
\begin{array}{cccc}
5,3 & 5,5 & 6,3 & 6,5 \\
\left(\begin{array}{cccc}
3.5829 & 0. & 0. & 0. \\
0. & 0. & 0. & 0. \\
0. & 0. & 3.5829 & 0. \\
0. & 0. & 0. & 0.
\end{array}\right)
\end{array}
$$

Equation 3-20.

Since the matrices are symmetric, only the lower or upper triangular portion of the matrices need to be provided via the DMIG entries.

The corresponding input file is shown in Figure 3-33.

```
$ FILE bd03dmi.dat
$
$ DMIG EXAMPLE
$ CHAPTER 3, NORMAL MODES
$
SOL 103 $ NORMAL MODES ANALYSIS
TIME 10
CEND
TITLE = DMIG TO READ STIFFNESS AND MASS FOR ELEM 5
SUBTITLE = PLANAR PROBLEM
$
SPC = 10
$
$ SPECIFY K2GG AND M2GG
K2GG = EXSTIF
M2GG = EXMASS
$
METHOD = 10
$
BEGIN BULK
$
$.......2.......3.......4.......5.......6.......7.......8.......9.......10.....$
$
$EIGRL SID V1 V2 ND
EIGRL 10 2
$
CBAR 1 1 1 2 10
CBAR 2 1 2 3 10
CBAR 3 1 3 4 10
CBAR 4 1 4 5 10
$
$ HEADER ENTRY FOR STIFFNESS
DMIG EXSTIF 0 6 1
$
DMIG EXSTIF 5 3 5 3 500039. +000001
++0000015 5 -250019. 6 3 -500039. +000002
++0000026 5 -250019. +000003
$
DMIG EXSTIF 5 5 5 5 166680. +000004
++0000046 3 250019. 6 5 83340. +000005
$
```

```
DMIG EXSTIF 6 3 6 3 500039. +000006
++0000066 5 250019. +000007
$
DMIG EXSTIF 6 5 6 5 166680.
$
$ HEADER ENTRY FOR MASS
DMIG EXMASS 0 6 1
$
$ DATA ENTRIES FOR MASS
$
DMIG EXMASS 5 3 5 3 3.5829
DMIG EXMASS 6 3 6 3 3.5829
$
GRID 1 0. 0. 0. 1246
GRID 2 1. 0. 0. 1246
GRID 3 2. 0. 0. 1246
GRID 4 3. 0. 0. 1246
GRID 5 4. 0. 0. 1246
GRID 6 5. 0. 0. 1246
GRID 10 0. 0. 10. 123456
MAT1 1 7.1+10 .33 2700.
PBAR 1 1 2.654-3 5.869-7
SPC1 10 123456 1
$
ENDDATA
```

Figure 3-33. Input File for the DMIG Example

In this example, EXSTIF is chosen as the name of the input stiffness matrix. Therefore, in order to bring in this stiffness matrix and add it to the global stiffness matrix, the Case Control command K2GG = EXSTIF is required. Note that these stiffness terms are additions to any existing terms in the global stiffness matrix at that location and not replacements of the stiffness terms at that location. In the Bulk Data Section, five DMIG entries are required—one for the header entry and four for the data column entries since there are four non-null columns in the above matrix.

For the header entry, the same name EXSTIF must be used to match the name called out in the Case Control Section. The third field is "0", which must be the value used for the header entry. The fourth field (IFO) is set to "6" to denote a symmetric matrix input. The fifth field (TIN) is set to "1" to denote that the matrix is real, single precision.

The terms in the matrix are referenced in terms of their external grid IDs when using the DMIG entries. Physically, each term in a particular column of the stiffness matrix (K_{ij}) represents the induced reactive load in the i-th degree-of-freedom due to a unit displacement in the j-th direction with all other displacement degrees-of-freedom held to zero. Since the matrix is symmetric, only the lower triangular portion of the matrix is input.

The first DMIG data column entry defines the first column of the above matrix. Field 2 of this DMIG entry must have the same name EXSTIF as referenced by the Case Control K2GG = EXSTIF command. Fields 3 and 4 of this entry identify this column in terms of its external grid ID and corresponding degree-of-freedom, respectively. In this case, it is grid point5, degree-of-freedom 3 (z-translation at grid point 5). Once this column is defined, follow the format description as described in the section for column data entry format, and then you can input the four terms in this column row-by-row. These four terms are defined by sets of three fields. They are the external grid ID number, corresponding degree-of-freedom, and the actual matrix term, respectively. The first term of column one is defined by external grid ID 5, degree-of-freedom 3 (z-translation at grid point 5) with a stiffness value of 500039. The second term of column one is defined by external grid ID 5, degree-of-freedom 5 (y-rotation at grid point 5) with a stiffness value of -250019. The third term of column one is defined by external grid ID 6, degree-of-freedom 3 (z-translation at grid point 6) with a stiffness value of -500039. The fourth term of column one is defined by external grid ID 6, degree-of-freedom 5 (y-rotation at grid point 6) with a stiffness value of -250019.

The next DMIG entry defines the second column of the above matrix. Fields 3 and 4 of this entry identify this column in terms of its external grid ID and corresponding degree-of-freedom, respectively. In this case, it is grid point 5, degree-of-freedom 5 (y-rotation at grid point 5). The

rest of the procedure is similar to that of column one with the exception that only three terms need to be input due to symmetry.

The next two DMIG entries defines columns three and four of the stiffness matrix, respectively. Note that due to symmetry, one less row needs to be defined for each additional column.

The mass matrix is input in a similar manner as the stiffness matrix with the following exceptions:

- The command M2GG = EXMASS instead of K2GG = EXSTIF is used in the Case Control Section. In this case, EXMASS is the name of the mass matrix referenced in field two of the DMIG Bulk Data entries.

- The matrix defined in the DMIG entries is expressed in the mass matrix terms rather than in stiffness matrix terms.

- Since there are only two non-null columns for the mass matrix, only two DMIG data entries are required instead of the four entries needed for the stiffness matrix.

Mass matrices input using DMIG are not scaled by PARAM,WTMASS.

In this example the small-field input format is used, and the maximum number of characters that can be input are eight (including sign and decimal point). Greater input precision can be achieved by using the large-field format and by changing the TIN field to 2 for the DMIG entries.

The first two computed natural frequencies for this example are 1.676 Hz and 10.066 Hz.

Chapter

4   Rigid Body Modes

- *Overview*

- *SUPORT Entry*

- *Examples*

4.1 Overview

A structure or a portion of a structure can displace without developing internal loads or stresses if it is not sufficiently tied to ground (constrained). These stress-free displacements are categorized as rigid-body modes or mechanism modes.

Rigid-body Modes

Rigid-body modes occur in unconstrained structures, such as satellites and aircraft in flight. For a general, unconstrained 3-D structure without mechanisms, there are six rigid-body modes often described as T1, T2, T3, R1, R2, and R3, or combinations thereof. Rigid-body modes can also be approximated for certain kinds of dynamic or modal tests in which the test specimen is supported by very flexible supports, such as bungee cords or inflatable bags. In this case the test specimen itself does not distort for the lowest mode(s) but instead displaces as a rigid body with all of the deformation occurring in the flexible support. Rigid-body modes can improperly occur if a structure that should be constrained is not fully constrained (for example, in a building model for which the boundary conditions (SPCs) were forgotten).

Mechanism Modes

A mechanism mode occurs when a portion of the structure can displace as a rigid body, which can occur when there is an internal hinge in the structure. An example of a mechanism is a ball and socket joint or a rudder in an airplane. A mechanism mode can also occur when two parts of a structure are improperly joined. A common modeling error resulting in a mechanism is when a bar is cantilevered from a solid element; the bar has rotational stiffness and the solid has no rotational stiffness, resulting in a pinned connection when the two are joined.

The presence of rigid-body and/or mechanism modes is indicated by zero frequency eigenvalues. Due to computer roundoff, the zero frequency eigenvalues are numerical zeroes on the order of 1.0E-4 Hz or less for typical structures. The same unconstrained model may give different values of the rigid-body frequencies when run on different computer types.

Rigid-body modes generated by NX Nastran are orthogonal with respect to each other and with respect to the flexible (or elastic) modes. Each rigid-body mode has the following property:

$$\{\phi_{rig}\}^T [M] \{\phi_{rig}\} > 0$$

$$\{\phi_{rig}\}^T [K] \{\phi_{rig}\} = 0$$

Equation 4-1.

where $\{\phi_{rig}\}$ denotes the rigid-body mode shapes, $[M]$ denotes the mass matrix, and $[K]$ denotes the stiffness matrix. The rigid-body modes contain no strain energy.

Figure 4-1 depicts the rigid-body modes for a two-dimensional model of a straight beam. Because the two-dimensional model has three DOFs per grid point (x-translation, y-translation, and z-rotation), there are three rigid-body modes. The figure shows classical rigid-body modes, whereby one mode is purely x-translation, another is purely y-translation, and another is purely z-rotation about the center of the beam. Because rigid-body modes are a special case of

repeated roots, any linear combination of these displacement shapes also comprises a valid set of rigid-body modes.
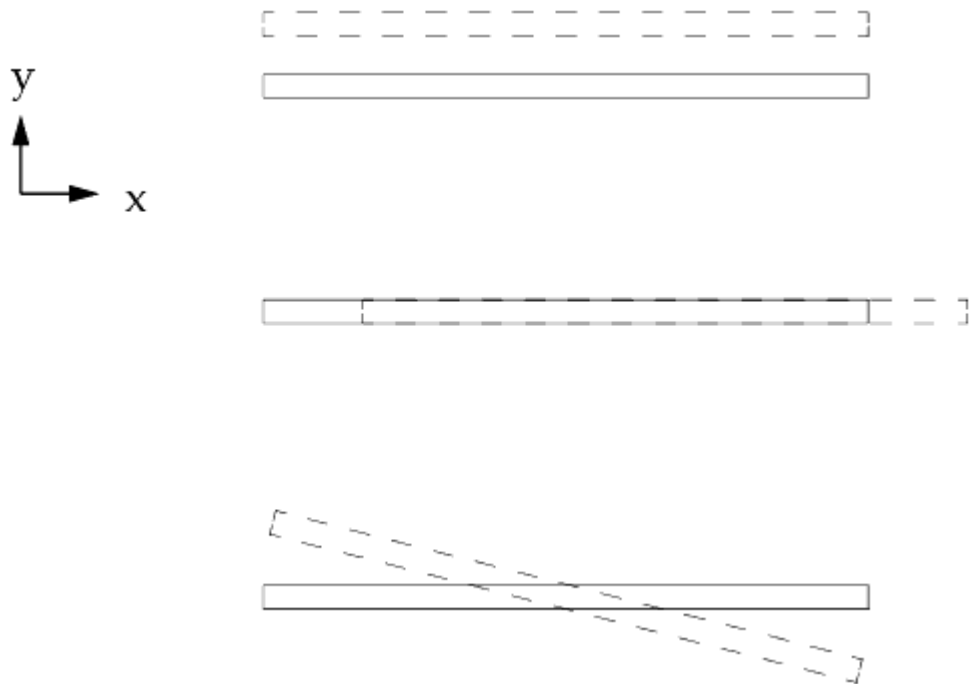
Figure 4-1. Rigid-Body Modes of a 2-D Beam

4.2   SUPORT Entry

Rigid-body modes are computed in NX Nastran without requiring special user intervention, although the use of a SUPORT Bulk Data entry makes the mode shapes look cleaner. The SUPORT entry also makes the rigid-body mode shapes repeatable when the mass or stiffness of the model changes and the model is reanalyzed. The SUPORT (note spelling!) entry does not constrain the model; it simply provides a frame of reference for the rigid-body shape calculations by defining the r-set components of motion. The SUPORT entry is not required for any of the dynamic analysis methods except for response spectrum analysis (see "Advanced Dynamic Analysis Capabilities").

If the SUPORT is used, one DOF should be specified for each rigid-body mode or mechanism. The form of the SUPORT entry is shown below:

1	2	3	4	5	6	7	8	9	10
SUPORT	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Field	Contents
IDi	Grid or scalar point identification number.
Ci	Component numbers. (0 or blank for scalar points and any unique combination of the integers 1 through 6 for grid points).

Treatment of SUPORT by Eigenvalue Analysis Methods

The eigenvalue extraction methods treat the SUPORT entry differently as described below.

EIGR Methods

Each of the eigenvalue extraction methods selected on the EIGR Bulk Data entry (AHOU, HOU, MHOU, and SINV) treats the SUPORT in the same manner. Eigenvalues are first computed using the information on the EIGR Bulk Data entry. The frequencies of the first N_r modes (where N_r is the number of SUPORT DOFs) are replaced with a value of 0.0 Hz. The first N_r eigenvectors are replaced by modes that are calculated by moving each SUPORT DOF a unit distance and then mass orthogonalizing them with respect to the other modes. The fact that the eigenvectors are calculated via kinematics is the reason that the SUPORT entry produces cleaner rigid-body modes (as opposed to the rigid-body modes computed without the use of the SUPORT entry).

Note that NX Nastran has no built-in checks to warn if any of the N_r eigenvalues are not rigid-body modes. This replacement can hide potential modeling problems. The results from UIM 3035 should be carefully checked (see "Examples"). The strain energy term for each SUPORT DOF should be zero.

A poor choice of DOFs on the SUPORT entry can adversely affect the computation of the flexible modes for the SINV method. Flexible modes computed with the other methods (AHOU, HOU, and MHOU) are not adversely affected by a poor choice of SUPORT DOFs. Again, the results of UIM 3035 should be carefully checked to ensure that a proper choice of SUPORT DOFs is made.

Lanczos Method

In the Lanczos method, the SUPORT entry attempts to give 0.0 Hz modes. The Lanczos routine compares the values of the calculated modes (without the SUPORT entry) and determines if the calculated frequencies for the N_r modes are near 0.0 Hz. If the computed frequencies are near 0.0 Hz, the computed values are replaced with 0.0 Hz. If a computed frequency is not near 0.0 Hz, then it is retained. (Note that this may cause problems in response spectrum analysis where 0.0 Hz is required for the SUPORT frequencies.) The Lanczos-computed, rigid-body eigenvectors are used, i.e., the rigid-body eigenvectors computed by the SUPORT entry are not used.

In the above paragraph, "near" means that the eigenvalues are less than 0.01 times the shift scale (the SHFSCL field on the EIGRL entry). All computed eigenvalues less than this threshold are candidate rigid-body modes. There is some logic to determine the relationship of these candidate rigid-body modes to the number of DOFs on the SUPORT entry. Suppose that there are three eigenvalues less than the threshold of 0.01 times the shift scale. If your SUPORT entry defines two DOFs, then the first two frequencies become 0.0 Hz, and the third becomes something that is nonzero but small (on the order of 1.0E-6, for example). On the other hand, if your SUPORT entry defines four DOFs, then only the first three are treated as rigid-body modes, and the fourth is not replaced. Furthermore, the use of a SUPORT entry forces an extra decomposition, which increases the computer run time.

Because the SUPORT entry is not used to compute the rigid-body eigenvectors, there is probably little to be gained by using the SUPORT entry with the Lanczos method unless response spectrum analysis is being performed.

Theoretical Considerations

Degrees-of-freedom defined on the SUPORT entry are placed in the r-set. When an r-set is present, static rigid-body vectors are calculated in NX Nastran by first partitioning the a-set into the r- and *l*-sets

$$\{u_a\} = \begin{Bmatrix} u_l \\ u_r \end{Bmatrix}$$

Equation 4-2.

Introducing this partitioning in the stiffness matrix results in

$$\begin{bmatrix} K_{ll} & K_{lr} \\ K_{rl} & K_{rr} \end{bmatrix} \begin{Bmatrix} u_l \\ u_r \end{Bmatrix} = \begin{Bmatrix} 0 \\ P_r \end{Bmatrix}$$

Equation 4-3.

for the rigid-body modes defined in the r-set.

There is no load on *l*-set DOFs. The load $\{P_r\}$ on the r-set is not needed in subsequent equations. Then solve for u_l in terms of u_r

$$\{u_l\} = [D]\{u_r\}$$

Equation 4-4.

where:

$$[D] = -[K_{ll}]^{-1}[K_{lr}]$$

The matrix $[D]$ is used to construct a set of rigid-body vectors $[y\,rig]$

$$[\psi_{rig}] = \begin{bmatrix} D \\ I_r \end{bmatrix}$$

Equation 4-5.

where $[y\,rig]$ represents the motion of the a-set for a unit motion of each SUPORT DOF with all other SUPORT DOFs constrained and $[I_r]$ is an r • r identity matrix.

The rigid-body vectors can be used to create a rigid-body mass matrix $[M_r]$

$$[M_r] = [\psi_{rig}]^T [M_{aa}][\psi_{rig}] = \begin{bmatrix} D \\ I_r \end{bmatrix}^T [M_{aa}] \begin{bmatrix} D \\ I_r \end{bmatrix}$$

Equation 4-6.

To improve the quality of the rigid-body mode shapes, orthogonalization is applied to create a diagonal mass matrix $[M_o]$ by

$$[M_o] = [\phi_{ro}^T][M_r] = [\phi_{ro}]$$

Equation 4-7.

where:

$[\phi_{ro}]$ is a transformation matrix.

This transformation matrix is used to construct the final set of rigid-body mode shape vectors by

$$[\phi_{rig}] = [\psi_{rig}][\phi_{ro}] = \begin{bmatrix} D\phi_{ro} \\ \phi_{ro} \end{bmatrix}$$

Equation 4-8.

such that

$$[\phi_{rig}]^T[K_{aa}][\phi_{rig}] \approx 0$$

Equation 4-9.

$$[\phi_{rig}]^T[M_{aa}][\phi_{rig}] = \begin{bmatrix} \ddots M_o \end{bmatrix}$$

Equation 4-10.

where:

$$\begin{bmatrix} \ddots M_o \end{bmatrix}$$

is a diagonal matrix.

Care must be taken when selecting SUPORT DOFs. Each SUPORT DOF must be able to displace independently without developing internal stresses. In other words, the SUPORT DOFs must be statically determinate. The SUPORT is used only to facilitate the calculation of rigid-body vectors. If you do not specify the r-set DOFs, the rigid-body modes are calculated directly by the method selected for the flexible frequency modes. If an insufficient number of r-set DOFs are specified, the calculation of rigid-body modes is unreliable.

As a modeling aid, NX Nastran calculates equivalent internal strain energy (work) for each rigid-body vector as follows:

$$[X] = \begin{bmatrix} D^T & I_r \end{bmatrix} \begin{bmatrix} K_{ll} & K_{lr} \\ K_{rl} & K_{rr} \end{bmatrix} \begin{bmatrix} D \\ I_r \end{bmatrix}$$

Equation 4-11.

which can be simplified as

$$[X] = [D]^T [K_{ll}][D] + [K_{rr}]$$

Equation 4-12.

When r-set DOFs exist, the printed strain energies are the diagonal elements of $[X]$ divided by 2 and should be approximately zero.

Note that $[X]$ is the transformation of the stiffness matrix $[K_{aa}]$ to r-set coordinates, which by definition of rigid-body (i.e., zero frequency) vector properties should be null. If this is not the case, the equilibrium may be violated by the r-set choice or other modeling errors may exist. The matrix $[X]$ is also called the rigid-body check matrix.

NX Nastran also calculates a rigid-body error ratio

$$\varepsilon = \frac{\left\| [K_{rr}] + [K_{lr}]^T [D] \right\|}{\left\| K_{rr} \right\|}$$

Equation 4-13.

where: $\| \ \|$ = Euclidean norm of the matrix $= \sqrt{\sum_i \sum_i x_{ij}^2}$

One value of ϵ is calculated using Equation 4-13 based on all SUPORT DOFs. Therefore, in UIM 3035 the same ϵ is printed for every supported DOF.

The rigid-body error ratio and the strain energy should be zero if a set of statically determinate SUPORT DOFs is chosen. Roundoff error may lead to computational zero values for these quantities. ("Computational zero" is a small number (10^{-5} , for example) that normally is 0.0 except for numerical roundoff.) The rigid-body error ratio and strain energy may be significantly nonzero for any of the following reasons:

- Roundoff error accumulation.

- The u_r set is overdetermined, leading to redundant supports. The condition gives high strain energy.

- The u_r set is underspecified, leading to a singular reduced-stiffness matrix and a MAXRATIO error. This condition gives a high rigid-body error ratio.

- The multipoint constraints are statically indeterminate. This condition gives high strain energy and a high rigid-body error ratio.

- There are too many single-point constraints. This condition gives high strain energy and a high rigid-body error ratio.

- $[K_{rr}]$ is null. This condition gives a unit value for the rigid-body error but low strain energy (see "Advanced Dynamic Analysis Capabilities").

Modeling Considerations

When using a SUPORT you must select a set of DOFs that is capable of constraining all the rigid-body modes. Another way to state this requirement is that the r-set must be able to constrain the structure in a statically determinate manner. There are usually many choices of DOFs that satisfy this requirement. Two choices that work for simple three-dimensional structures are:

- Six DOFs on one grid point when all its degrees-of-freedom have stiffness.

- Three translation DOFs normal to one plane, two translation DOFs normal to an orthogonal plane, and then one translational DOF normal to the last orthogonal plane. Such a system can be used, for instance, on a model composed entirely of solid elements that have no inherent stiffness for grid point rotation. See Figure 4-2.

1, 2, 3 Normal to Plane 1
4, 5 Normal to Plane 2
6 Normal to Plane 3
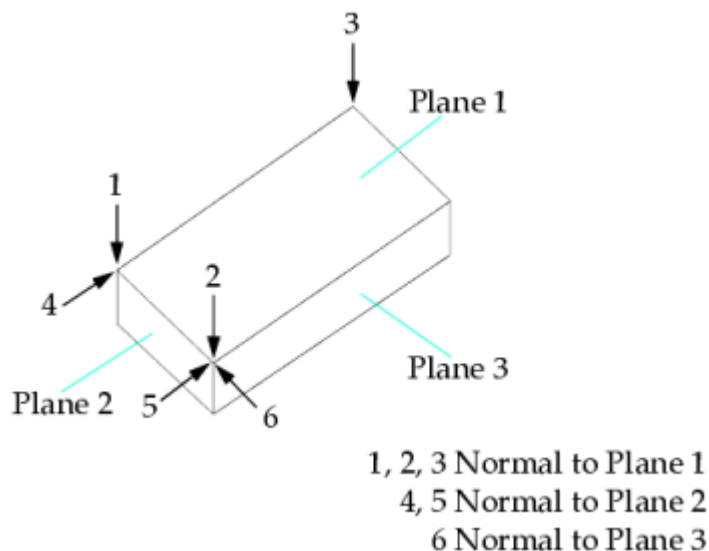
Figure 4-2. Statically Determinate r-set

There are special cases where a model need not have six rigid-body modes. A planar model has only three rigid-body modes, while an airplane with a free rudder has seven, for example. If you use the SUPORT, it is your responsibility to determine all the modes of rigid-body motion, then provide r-set DOFs that define these rigid-body modes. Another special case is the application of enforced motion by the large mass technique (see "Enforced Motion"). If the input points describe redundant load paths, diagnostics are produced that indicate overconstraint. For this case, these diagnostics may be safely ignored.

Poorly-constrained rigid-body modes result from either constraining DOFs with relatively little stiffness or from constraining a set of DOFs that are almost linearly dependent on one another. An example of the former is a model of a very thin cylindrical shell. The degrees-of-freedom normal to the shell and their associated bending degrees-of-freedom may all be too soft to avoid numerical conditioning problems. A modeling cure for this condition is to connect many grid

points to a new reference grid point with an RBE3 element and then to place the reference grid point in the r-set. The RBE3 element does not affect the flexible modes when applied in this manner.

An example of a structure whose r-set shows poor linear independence is a slightly curved bar modeled using coordinate systems that follow the curve of the bar such that the x-axis is always tangent to the bar. The x DOFs at each end of the bar describe linearly independent DOFs in a mathematical sense. However, numerical truncation produces poor conditioning if the angle between the ends is less than a few degrees. This condition is detected by the automatic diagnostics discussed earlier. This problem can be corrected (or better yet, avoided) by making a careful sketch of all r-set DOFs, including their locations in space and the orientation of their global coordinates. Then apply the three-plane test described earlier.

Using a physical analogy, a good r-set can be chosen by finding one grid point that sustains all possible loadings well if it is tied to ground in an actual hardware test. If there is no such grid point, the ties to ground should be spread over enough grid points to sustain the loads without damaging the structure. An RBE3 element used for this purpose can provide good rigid-body modes without affecting the flexible modes.

4.3   Examples

This section provides several rigid-body modes examples showing input and output. These examples are as follows:

Table 4-1. Unconstrained Beam Model Summary			
Model	**Analysis Method**	**SUPORT/ No SUPORT**	**Redundancy of SUPORT**
bd04bar1	Lanczos	No SUPORT	–
bd04bar2	Lanczos	SUPORT	Statically determinate
bd04bar3	Lanczos	SUPORT	Underdetermined
bd04bar4	Lanczos	SUPORT	Overdetermined
bd04bar5	SINV	No SUPORT	–
bd04bar6	SINV	SUPORT	Statically determinate
bd04bar7	SINV	SUPORT	Underdetermined
bd04bar8	SINV	SUPORT	Overdetermined
bd04bkt	Lanczos	No SUPORT	–

These examples are described in the sections that follow.

Unconstrained Beam Model

The constraints (SPCs) on the example cantilever beam model from "Cantilever Beam Model" are removed to create an unconstrained structure as shown in Figure 4-3. A GRDSET entry is added with the z-translation, x-rotation, and y-rotation directions constrained to make the problem two-dimensional. Therefore, there are three DOFs per grid point (x-translation, y-translation, and z-rotation) and three rigid-body modes.

Figure 4-3. Unconstrained Beam Model

Modes are computed using two methods (Lanczos and SINV), with and without a SUPORT entry. The SUPORT entry is used in three ways:

• Statically determinate (grid point 1, components 1, 2, and 6)

• Underdetermined (grid point 1, components 1 and 2)

• Overdetermined (grid point 1, components 1, 2, and 6, plus grid point 11, component1)

Figure 4-4 shows a portion of the input file for the statically determinate SUPORT and the Lanczos method.

```
$ FILE bd04bar2.dat
$
$ CANTILEVER BEAM MODEL
$ CHAPTER 4, RIGID-BODY MODES
$
SOL 103
TIME 10
CEND
TITLE = CANTILEVER BEAM
SUBTITLE = NORMAL MODES
LABEL = USE SUPORT, STATICALLY DETERMINATE
$
$ OUTPUT REQUEST
DISPLACEMENT = ALL
$
$ SELECT EIGRL ENTRY
METHOD = 10
$
BEGIN BULK
$
$.......2.......3.......4.......5.......6.......7.......8.......9.......10.....$
$
$ STATICALLY DETERMINATE SUPORT
$SUPORT G C
SUPORT 1 126
$
$ MAKE 2D MODEL
GRDSET 345
$
$EIGRL SID V1 V2
EIGRL 10 -0.1 50.
$
... basic model ...
$
ENDDATA
```

Figure 4-4. Input File for Cantilever Beam Model

Table 4-2 lists the computed frequencies. The overdetermined run for the Lanczos method works well; the same run for the SINV method gives an extra zero frequency mode that does not really exist. The overdetermined runs have redundant SUPORTs in the x-direction; therefore, two rigid-body modes are computed in this direction when using the SINV method. In all cases the flexible (greater than 0) frequencies are correct.

Table 4-2. Frequencies for the Unconstrained Beam Models

Mode	Lanczos Method Frequencies (Hz)				SINV Method Frequencies (Hz)			
	No SUPORT	Under Det.	Stat. Det. SUPORT	Over- Det. SUPORT	No SUPORT	Under Det.	Stat. Det. SUPORT	Over Det. SUPORT
1	2.15E-7	0.0	0.0	0.0	6.88E-7	0.0	0.0	0.0
2	4.02E-7	0.0	0.0	0.0	6.79E-6	0.0	0.0	0.0
3	8.07E-6	3.07E-5	0.0	0.0	7.12E-6	1.45E-6	0.0	0.0
4	12.82	12.82	12.82	12.82	12.82	12.82	12.82	0.0
5	34.62	34.62	34.62	34.62	34.62	34.62	34.62	12.82
6					66.60	66.60	66.60	34.62
7								66.60

Figure 4-5 shows the output for the rigid-body modes computed for the SINV method when using no SUPORT. The rigid-body frequencies are denoted by computational zeroes on the order of 10^{-5} Hz or less. Note that the magnitude may be different when the same problem is run on a different computer type.

Figure 4-6 shows the output for the rigid-body modes computed for the SINV method when using the statically determinate SUPORT. The accuracy of the statically determinate SUPORT DOFs is verified by the computational zeroes for epsilon and the strain energy printed in UIM 3035. Note that the three rigid-body modes have frequencies of 0.0 Hz. The SUPORT entry provides cleaner mode shapes than those shown in Figure 4-5 as illustrated by the purely x-translation, y-translation, and z-rotation eigenvectors shown in Figure 4-6.

```
                                       R E A L   E I G E N V A L U E S
      MODE    EXTRACTION       EIGENVALUE           RADIANS              CYCLES            GENERALIZED          GENERALIZ
      NO.       ORDER                                                                         MASS              STIFFNES
       1          1         1.866121E-11         4.319862E-06         6.875275E-07        1.000000E+00         1.866121E
       2          2         1.819082E-09         4.265070E-05         6.788069E-06        1.000000E+00         1.819082E
       3          3         2.000299E-09         4.472470E-05         7.118156E-06        1.000000E+00         2.000299E
       4          4         6.483918E+03         8.052278E+01         1.281560E+01        1.000000E+00         6.483918E
       5          5         4.732272E+04         2.175379E+02         3.462222E+01        1.000000E+00         4.732272E
       6          6         1.751285E+05         4.184836E+02         6.660372E+01        1.000000E+00         1.751285E

      EIGENVALUE =  1.866121E-11
        CYCLES =  6.875275E-07      R E A L   E I G E N V E C T O R   N O .       1

      POINT ID.   TYPE       T1                T2               T3                R1                R2                R3
          1        G     4.273806E-34   -2.470432E-01    0.0              0.0              0.0              -1.067134E-01
          2        G     4.273806E-34   -2.790573E-01    0.0              0.0              0.0              -1.067134E-01
          3        G     4.273806E-34   -3.110713E-01    0.0              0.0              0.0              -1.067134E-01
          4        G     4.273806E-34   -3.430853E-01    0.0              0.0              0.0              -1.067134E-01
          5        G     4.273806E-34   -3.750993E-01    0.0              0.0              0.0              -1.067134E-01
          6        G     4.273806E-34   -4.071134E-01    0.0              0.0              0.0              -1.067134E-01
          7        G     4.273806E-34   -4.391274E-01    0.0              0.0              0.0              -1.067134E-01
          8        G     4.273806E-34   -4.711414E-01    0.0              0.0              0.0              -1.067134E-01
          9        G     4.273806E-34   -5.031555E-01    0.0              0.0              0.0              -1.067134E-01
         10        G     4.273806E-34   -5.351695E-01    0.0              0.0              0.0              -1.067134E-01
         11        G     4.273806E-34   -5.671835E-01    0.0              0.0              0.0              -1.067134E-01

      EIGENVALUE =  1.819082E-09
        CYCLES =  6.788069E-06      R E A L   E I G E N V E C T O R   N O .       2

      POINT ID.   TYPE       T1                T2               T3                R1                R2                R3
          1        G    -2.953214E-01   -5.597336E-01    0.0              0.0              0.0              3.291537E-01
          2        G    -2.953214E-01   -4.609875E-01    0.0              0.0              0.0              3.291537E-01
          3        G    -2.953214E-01   -3.622414E-01    0.0              0.0              0.0              3.291537E-01
          4        G    -2.953214E-01   -2.634953E-01    0.0              0.0              0.0              3.291537E-01
          5        G    -2.953214E-01   -1.647492E-01    0.0              0.0              0.0              3.291537E-01
          6        G    -2.953214E-01   -6.600305E-02    0.0              0.0              0.0              3.291537E-01
          7        G    -2.953214E-01    3.274305E-02    0.0              0.0              0.0              3.291537E-01
          8        G    -2.953214E-01    1.314892E-01    0.0              0.0              0.0              3.291538E-01
          9        G    -2.953214E-01    2.302354E-01    0.0              0.0              0.0              3.291538E-01
         10        G    -2.953214E-01    3.289815E-01    0.0              0.0              0.0              3.291538E-01
         11        G    -2.953214E-01    4.277276E-01    0.0              0.0              0.0              3.291538E-01

      EIGENVALUE =  2.000299E-09
        CYCLES =  7.118156E-06      R E A L   E I G E N V E C T O R   N O .       3

      POINT ID.   TYPE       T1                T2               T3                R1                R2                R3
          1        G    -2.953611E-01    5.596586E-01    0.0              0.0              0.0              -3.291096E-01
```

```
 2    G    -2.953611E-01    4.609257E-01   0.0           0.0           0.0          -3.291096E-01
 3    G    -2.953611E-01    3.621928E-01   0.0           0.0           0.0          -3.291096E-01
 4    G    -2.953611E-01    2.634599E-01   0.0           0.0           0.0          -3.291096E-01
 5    G    -2.953611E-01    1.647270E-01   0.0           0.0           0.0          -3.291096E-01
 6    G    -2.953611E-01    6.599414E-02   0.0           0.0           0.0          -3.291095E-01
 7    G    -2.953611E-01   -3.273870E-02   0.0           0.0           0.0          -3.291095E-01
 8    G    -2.953611E-01   -1.314715E-01   0.0           0.0           0.0          -3.291095E-01
 9    G    -2.953611E-01   -2.302044E-01   0.0           0.0           0.0          -3.291095E-01
10    G    -2.953611E-01   -3.289373E-01   0.0           0.0           0.0          -3.291095E-01
11    G    -2.953611E-01   -4.276701E-01   0.0           0.0           0.0          -3.291095E-01
```

Figure 4-5. Unconstrained Beam Modes Without SUPORT (SINV Method)

```
*** USER INFORMATION MESSAGE 3035 FOR DATA BLOCK   KLR
    SUPPORT PT.NO.            EPSILON            STRAIN   ENERGY    EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH AS
            1            7.7496606E-17        5.5879354E-09
            2            7.7496606E-17        0.0000000E+00
            3            7.7496606E-17        7.1622708E-11

                                 R E A L   E I G E N V A L U E S
    MODE    EXTRACTION      EIGENVALUE         RADIANS            CYCLES          GENERALIZED         GENERALIZE
    NO.       ORDER                                                                 MASS              STIFFNESS
     1          1          0.0               0.0               0.0               1.000000E+00        0.0
     2          2          0.0               0.0               0.0               1.000000E+00        0.0
     3          3          0.0               0.0               0.0               1.000000E+00        0.0
     4          4          6.483918E+03      8.052278E+01      1.281560E+01      1.000000E+00        6.483918E+
     5          5          4.732272E+04      2.175379E+02      3.462222E+01      1.000000E+00        4.732272E+
     6          6          1.751285E+05      4.184836E+02      6.660372E+01      1.000000E+00        1.751285E+
     7          7          4.616299E+05      6.794335E+02      1.081352E+02      1.000000E+00        4.616299E+

    EIGENVALUE =  0.000000E+00
       CYCLES =  0.000000E+00      R E A L   E I G E N V E C T O R   N O .         1

    POINT ID.   TYPE        T1              T2              T3              R1              R2              R3
         1      G       4.176756E-01    0.0             0.0             0.0             0.0             0.0
         2      G       4.176756E-01    0.0             0.0             0.0             0.0             0.0
         3      G       4.176756E-01    0.0             0.0             0.0             0.0             0.0
         4      G       4.176756E-01    0.0             0.0             0.0             0.0             0.0
         5      G       4.176756E-01    0.0             0.0             0.0             0.0             0.0
         6      G       4.176756E-01    0.0             0.0             0.0             0.0             0.0
         7      G       4.176756E-01    0.0             0.0             0.0             0.0             0.0
         8      G       4.176756E-01    0.0             0.0             0.0             0.0             0.0
         9      G       4.176756E-01    0.0             0.0             0.0             0.0             0.0
        10      G       4.176756E-01    0.0             0.0             0.0             0.0             0.0
        11      G       4.176756E-01    0.0             0.0             0.0             0.0             0.0

    EIGENVALUE =  0.000000E+00
       CYCLES =  0.000000E+00      R E A L   E I G E N V E C T O R   N O .         2

    POINT ID.   TYPE        T1              T2              T3              R1              R2              R3
         1      G       0.0           4.176756E-01   0.0             0.0             0.0             0.0
         2      G       0.0           4.176756E-01   0.0             0.0             0.0            -3.593777E-15
         3      G       0.0           4.176756E-01   0.0             0.0             0.0            -6.723840E-15
         4      G       0.0           4.176756E-01   0.0             0.0             0.0            -9.575675E-15
         5      G       0.0           4.176756E-01   0.0             0.0             0.0            -1.220145E-14
         6      G       0.0           4.176756E-01   0.0             0.0             0.0            -1.439249E-14
         7      G       0.0           4.176756E-01   0.0             0.0             0.0            -1.596912E-14
         8      G       0.0           4.176756E-01   0.0             0.0             0.0            -1.698929E-14
         9      G       0.0           4.176756E-01   0.0             0.0             0.0            -1.745155E-14
        10      G       0.0           4.176756E-01   0.0             0.0             0.0            -1.770949E-14
        11      G       0.0           4.176756E-01   0.0             0.0             0.0            -1.758923E-14

    EIGENVALUE =  0.000000E+00
       CYCLES =  0.000000E+00      R E A L   E I G E N V E C T O R   N O .         3

    POINT ID.   TYPE        T1              T2              T3              R1              R2              R3
         1      G       0.0          -7.163078E-01   0.0             0.0             0.0            4.775385E-01
         2      G       0.0          -5.730462E-01   0.0             0.0             0.0            4.775385E-01
         3      G       0.0          -4.297847E-01   0.0             0.0             0.0            4.775385E-01
         4      G       0.0          -2.865231E-01   0.0             0.0             0.0            4.775385E-01
         5      G       0.0          -1.432615E-01   0.0             0.0             0.0            4.775385E-01
         6      G       0.0           1.508516E-14   0.0             0.0             0.0            4.775385E-01
         7      G       0.0           1.432615E-01   0.0             0.0             0.0            4.775385E-01
         8      G       0.0           2.865231E-01   0.0             0.0             0.0            4.775385E-01
         9      G       0.0           4.297847E-01   0.0             0.0             0.0            4.775385E-01
        10      G       0.0           5.730463E-01   0.0             0.0             0.0            4.775385E-01
        11      G       0.0           7.163078E-01   0.0             0.0             0.0            4.775385E-01
```

Figure 4-6. Unconstrained Beam Modes With Statically Determinate SUPORT (SINV Method)

Table 4-7 shows the epsilon and strain energy printed in UIM 3035 for the three SUPORT cases (statically determinate, overdetermined, and underdetermined).

It can be seen from this example that aside from clean rigid-body vectors there is no advantage
to using a SUPORT entry to compute rigid-body modes.

```
Statically determinate SUPORT:

 *** USER INFORMATION MESSAGE 3035 FOR DATA BLOCK    KLR

     SUPPORT PT.NO.            EPSILON            STRAIN  ENERGY     EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTER
              1             7.7496606E-17          5.5879354E-09
              2             7.7496606E-17          0.0000000E+00
              3             7.7496606E-17          7.1622708E-11

Overdetermined SUPORT:

 *** USER INFORMATION MESSAGE 3035 FOR DATA BLOCK    KLR

     SUPPORT PT.NO.            EPSILON            STRAIN  ENERGY    EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTER
              1             1.9913979E-01          7.2869660E+06        ****
              2             1.9913979E-01          0.0000000E+00        ****
              3             1.9913979E-01          7.1850081E-11        ****
              4             1.9913979E-01          7.2869660E+06        ****

Underdetermined SUPORT:

*** USER INFORMATION MESSAGE 4158---STATISTICS FOR SYMMETRIC DECOMPOSITION OF DATA BLOCK KLL      FOLLOW

         MAXIMUM RATIO OF MATRIX DIAGONAL TO FACTOR DIAGONAL = 9.1E+13 AT ROW NUMBER    31
 *** USER WARNING MESSAGE 4698.  STATISTICS FOR DECOMPOSITION OF MATRIX KLL      .
THE FOLLOWING DEGREES OF FREEDOM HAVE FACTOR DIAGONAL RATIOS GREATER THAN
  1.00000E+05 OR HAVE NEGATIVE TERMS ON THE FACTOR DIAGONAL.

GRID POINT ID       DEGREE OF FREEDOM   MATRIX/FACTOR DIAGONAL RATIO        MATRIX DIAGONAL
11                      R3                  9.49483E+13                    2.84000E+04

 *** USER INFORMATION MESSAGE 3035 FOR DATA BLOCK    KLR

     SUPPORT PT.NO.            EPSILON            STRAIN  ENERGY     EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH AS
              1             7.6288287E-17          5.5879354E-09
              2             7.6288287E-17         -7.2759576E-12
```

Figure 4-7. UIM 3035 Results

Unconstrained Bracket Example

The constraints (SPCs) on the example bracket model from "Real Eigenvalue Analysis" are
removed to create an unconstrained model (see "Examples" for a description of the model).
Figure 4-8 shows the bracket model. The model is a three-dimensional model and therefore
produces six rigid-body modes. The NX Nastran results are shown in Figure 4-9 for the first
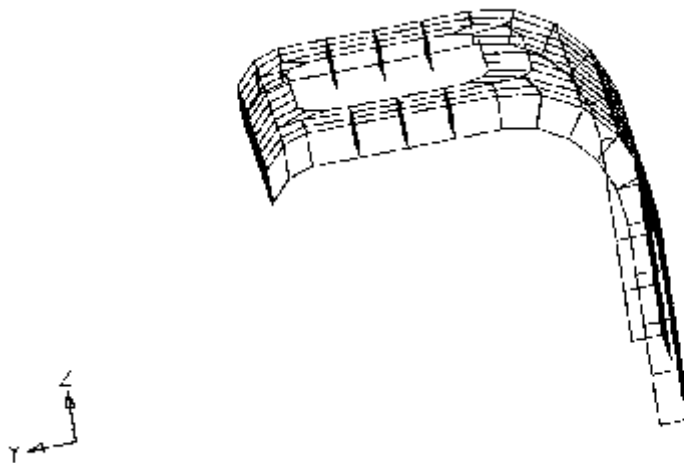eight modes. The Lanczos method is used.

Figure 4-8. Unconstrained Bracket Model

R E A L   E I G E N V A L U E S

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	-1.690642E-07	4.111741E-04	6.544039E-05	1.000000E+00	-1.690642E-07
2	2	-9.807991E-09	9.903530E-05	1.576196E-05	1.000000E+00	-9.807991E-09
3	3	-5.515176E-09	7.426423E-05	1.181952E-05	1.000000E+00	-5.515176E-09
4	4	-3.390596E-09	5.822883E-05	9.267405E-06	1.000000E+00	-3.390596E-09
5	5	1.266017E-08	1.125174E-04	1.790770E-05	1.000000E+00	1.266017E-08
6	6	1.726585E-08	1.313996E-04	2.091289E-05	1.000000E+00	1.726585E-08
7	7	2.649932E+06	1.627861E+03	2.590821E+02	1.000000E+00	2.649932E+06
8	8	4.279463E+06	2.068686E+03	3.292416E+02	1.000000E+00	4.279463E+06

Figure 4-9. Unconstrained Bracket Frequencies

The six rigid-body modes have computational zero frequencies on the order of 10^{-5} Hz. Note that the magnitudes of the rigid-body modes may be different when the same problem is run on a different computer type. Also note that the output is sorted by the value of the eigenvalue in ascending order.

Chapter

5   *Frequency Response Analysis*

- *Overview*

- *Direct Frequency Response Analysis*

- *Modal Frequency Response Analysis*

- *Modal Versus Direct Frequency Response*

- *Frequency-Dependent Excitation Definition*

- *Solution Frequencies*

- *Frequency Response Considerations*

- *Solution Control for Frequency Response Analysis*

- *Examples*

5.1   Overview

Frequency response analysis is a method used to compute structural response to steady-state oscillatory excitation. Examples of oscillatory excitation include rotating machinery, unbalanced tires, and helicopter blades. In frequency response analysis the excitation is explicitly defined in the frequency domain. All of the applied forces are known at each forcing frequency. Forces can be in the form of applied forces and/or enforced motions (displacements, velocities, or accelerations).

Phase Shift

Oscillatory loading is sinusoidal in nature. In its simplest case, this loading is defined as having an amplitude at a specific frequency. The steady-state oscillatory response occurs at the same frequency as the loading. The response may be shifted in time due to damping in the system. The shift in response is called a phase shift because the peak loading and peak response no longer occur at the same time. An example of phase shift is shown in Figure 5-1.
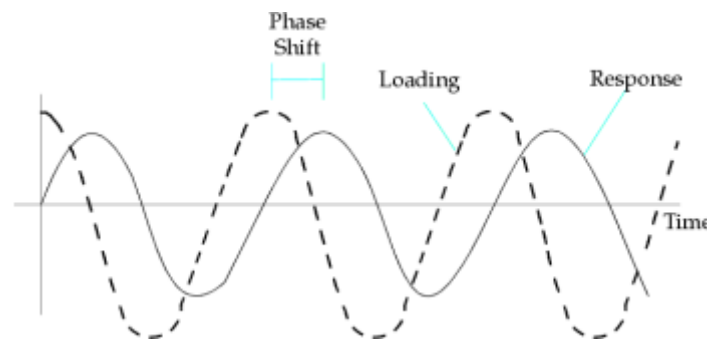

Figure 5-1. Phase Shift

Complex Numbers

The important results obtained from a frequency response analysis usually include the displacements, velocities, and accelerations of grid points as well as the forces and stresses of elements. The computed responses are complex numbers defined as magnitude and phase (with respect to the applied force) or as real and imaginary components, which are vector components of the response in the real/imaginary plane. These quantities are graphically presented in Figure 5-2.
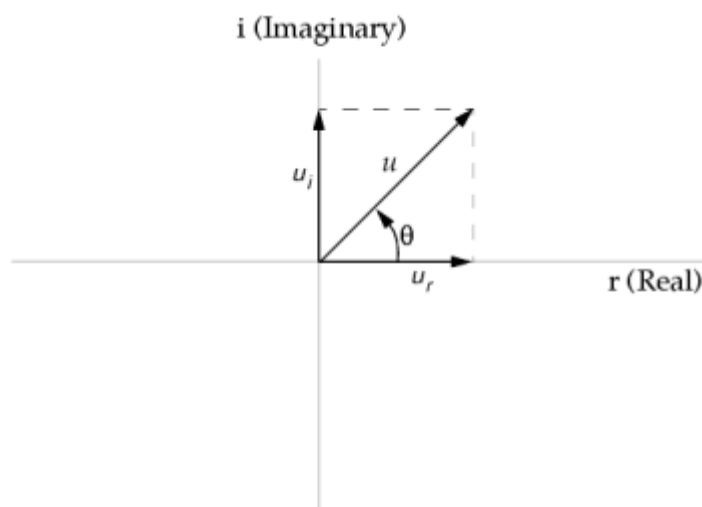
Figure 5-2. Complex Plane

where:

u	=	magnitude = $\sqrt{u_r^2 + u_i^2}$
q	=	phase angle = $\tan^{-1} (u_i/u_r$
u_r	=	real component = u cos q
u_i	=	imaginary component = $u$sin q

Two different numerical methods can be used in frequency response analysis. The direct method solves the coupled equations of motion in terms of forcing frequency. The modal method utilizes the mode shapes of the structure to reduce and uncouple the equations of motion (when modal or no damping is used); the solution for a particular forcing frequency is obtained through the summation of the individual modal responses. The choice of the method depends on the problem. The two methods are described in Direct Frequency Response Analysis and Modal Frequency Response Analysis.

5.2   Direct Frequency Response Analysis

In direct frequency response analysis, structural response is computed at discrete excitation frequencies by solving a set of coupled matrix equations using complex algebra. Begin with the damped forced vibration equation of motion with harmonic excitation

$$[M]\{\ddot{x}(t)\} + [B]\{\dot{x}(t)\} + [K]\{x(t)\} = \{P(\omega)\}e^{i\omega t}$$

Equation 5-1.

The load in Equation 5-1 is introduced as a complex vector, which is convenient for the mathematical solution of the problem. From a physical point of view, the load can be real or imaginary, or both. The same interpretation is used for response quantities.

For harmonic motion (which is the basis of a frequency response analysis), assume a harmonic solution of the form:

$$\{x\} = \{u(\omega)\}e^{i\omega t}$$

Equation 5-2.

where $\{u(w)\}$ is a complex displacement vector. Taking the first and second derivatives of Equation 5-2, the following is obtained:

$$\{\dot{x}\} = i\omega\{u(\omega)\}e^{i\omega t}$$

$$\{\ddot{x}\} = -\omega^2\{u(\omega)\}e^{i\omega t}$$

Equation 5-3.

When the above expressions are substituted into Equation 5-1, the following is obtained:

$$-\omega^2[M]\{u(\omega)\}e^{i\omega t} + i\omega[B]\{u(\omega)\}e^{i\omega t} + [K]\{u(\omega)\}e^{i\omega t} = \{P(\omega)\}e^{i\omega t}$$

Equation 5-4.

which after dividing by $e^{i w t}$ simplifies to

$$[-\omega^2 M + i\omega B + K]\{u(\omega)\} = \{P(\omega)\}$$

Equation 5-5.

The equation of motion is solved by inserting the forcing frequency w into the equation of motion. This expression represents a system of equations with complex coefficients if damping is included or the applied loads have phase angles. The equations of motion at each input frequency are then solved in a manner similar to a statics problem using complex arithmetic.

Damping in Direct Frequency Response

Damping simulates the energy dissipation characteristics of a structure. Damping in direct frequency response is represented by the damping matrix $[B]$ and additions to the stiffness matrix $[K]$.

The damping matrix is comprised of several matrices

$$[B] = [B^1] + [B^2]$$

Equation 5-6.

where:

$[B^1]$	=	damping elements (CVISC, CDAMPi) and B2GG
$[B^2]$	=	B2PP direct input matrix and transfer functions

In frequency response, PARAM,G and GE on the MATi entry do not form a damping matrix. Instead, they form the following complex stiffness matrix:

$$[K] = (1 + iG)[K] + i\sum G_E[K_E]$$

Equation 5-7.

where:

$[K]$	=	global stiffness matrix
G	=	overall structural damping coefficient (PARAM,G)
$[K_E]$	=	element stiffness matrix
G_E	=	element structural damping coefficient (GE on the MATi entry)

When the above parameters and/or coefficients are specified, they are automatically incorporated into the stiffness matrix and therefore into the equation of motion for the solution. All of the forms of damping can be used in the same analysis, and their effects are added together.

In frequency response analysis, it is not necessary to assume an equivalent viscous form for structural damping since the solution is complex. Therefore, a complex stiffness matrix is allowed.

5.3   Modal Frequency Response Analysis

Modal frequency response analysis is an alternate approach to computing the frequency response of a structure. This method uses the mode shapes of the structure to reduce the size, uncouple the equations of motion (when modal or no damping is used), and make the numerical solution more efficient. Since the mode shapes are typically computed as part of the characterization of the structure, modal frequency response is a natural extension of a normal modes analysis.

As a first step in the formulation, transform the variables from physical coordinates $\{u(\omega)\}$ to modal coordinates $\{x(\omega)\}$ by assuming

$$\{x\} = [\phi]\{\xi(\omega)\}e^{i\omega t}$$

Equation 5-8.

The mode shapes [f] are used to transform the problem in terms of the behavior of the modes as opposed to the behavior of the grid points. Equation 5-8 represents an equality if all modes are used; however, because all modes are rarely used, the equation usually represents an approximation.

To proceed, temporarily ignore all damping, which results in the undamped equation for harmonic motion

$$-\omega^2[M]\{x\} + [K]\{x\} = \{P(\omega)\}$$

Equation 5-9.

at forcing frequency ω.

Substituting the modal coordinates in Equation 5-8 for the physical coordinates in Equation5-9 and dividing by $e^{i\omega t}$, the following is obtained:

$$-\omega^2 [M][\phi]\{\xi(\omega)\} + [K][\phi]\{\xi(\omega)\} = \{P(\omega)\}$$

Equation 5-10.

Now this is the equation of motion in terms of the modal coordinates. At this point, however, the equations remain coupled.

To uncouple the equations, premultiply by $[\phi^T]$ to obtain

$$-\omega^2 [\phi]^T [M][\phi]\{\xi(\omega)\} + [\phi]^T [K][\phi]\{\xi(\omega)\} = [\phi]^T \{P(\omega)\}$$

Equation 5-11.

where:

$[\phi]^T [M][\phi]$	=	modal (generalized) mass matrix
$[\phi]^T [K][\phi]$	=	modal (generalized) stiffness matrix
$[\phi]^T \{P\}$	=	modal force vector

The final step uses the orthogonality property of the mode shapes to formulate the equation of motion in terms of the generalized mass and stiffness matrices, which are diagonal matrices. These diagonal matrices do not have the off-diagonal terms that couple the equations of motion. Therefore, in this form the modal equations of motion are uncoupled. In this uncoupled form, the equations of motion are written as a set of uncoupled single degree-of-freedom systems as

$$-\omega^2 m_i \xi_i(\omega) + k_i \xi_i(\omega) = p_i(\omega)$$

Equation 5-12.

where:

m_i	=	i-th modal mass
k_i	=	i-th modal stiffness
p_i	=	i-th modal force

The modal form of the frequency response equation of motion is much faster to solve than the direct method because it is a series of uncoupled single degree-of-freedom systems.

Once the individual modal responses $x_i(w)$ are computed, physical responses are recovered as the summation of the modal responses using

$$\{x\} = [\phi]\{\xi(\omega)\}e^{i\omega t}$$

Equation 5-13.

These responses are in complex form (magnitude/phase or real/imaginary) and are used to recover additional output quantities requested in the Case Control Section.

Damping in Modal Frequency Response

If a damping matrix [B] exists, the orthogonality property (see "Overview of Normal Modes Analysis") of the modes does not, in general, diagonalize the generalized damping matrix

$$[\phi]^T [B][\phi] \neq diagonal$$

Equation 5-14.

If structural damping is used, the orthogonality property does not, in general, diagonalize the generalized stiffness matrix

$$[\phi]^T [K][\phi] \neq diagonal$$

Equation 5-15.

where:

$$K = (1 + iG)[K] + i\sum G_E[K_E]$$

In the presence of a [B] matrix or a complex stiffness matrix, the modal frequency approach solves the coupled problem in terms of modal coordinates using the direct frequency approach described in Direct Frequency Response Analysis.

$$[-\omega^2 [\phi]^T [M][\phi] + i\omega[\phi]^T [B][\phi] + [\phi]^T [K][\phi]]\{\xi(\omega)\} = [\phi]^T \{P(\omega)\}$$

Equation 5-16.

Equation 5-16 is similar to Equation 5-5 for the direct frequency response analysis method except that Equation 5-16 is expressed in terms of modal coordinates x . Since the number of modes used in a solution is typically much less than the number of physical variables, using the coupled solution of the modal equations is less costly than using physical variables.

If damping is applied to each mode separately, the uncoupled equations of motion can be maintained. When modal damping is used, each mode has damping b_i where $b_i = 2m_i w_i z_i$. The equations of motion remain uncoupled and have the form

$$-\omega^2 m_i \xi_i(\omega) + i\omega b_i \xi_i(\omega) + k_i \xi_i(\omega) = p_i(\omega)$$

Equation 5-17.

for each mode.

Each of the modal responses is computed using

$$\xi_i(\omega) = \frac{p_i(\omega)}{-m_i\omega^2 + ib_i\omega + k_i}$$

Equation 5-18.

The TABDMP1 Bulk Data entry defines the modal damping ratios. A table is created by the frequency/damping pairs specified on the TABDMP1 entry. The solution refers to this table for the damping value to be applied at a particular frequency. The TABDMP1 Bulk Data entry has a Table ID. A particular TABDMP1 table is activated by selecting the Table ID with the SDAMPING Case Control command.

1	2	3	4	5	6	7	8	9	10
TABDMP1	TID	TYPE							
	f1	g1	f2	g2	f3	g3	-etc.-	ENDT	

Field	**Contents**
TID	Table identification number.
TYPE	Type of damping units:
	G (default)
	CRIT
	Q
fi	Frequency value (cycles per unit time).
gi	Damping value in the units specified.

At resonance, the three types of damping are related by the following equations:

$$\zeta_i = \frac{b_i}{b_{cr}} = \frac{G_i}{2}$$

$$b_{cr} = 2m_i\omega_i$$

$$Q_i = \frac{1}{(2\zeta_i)} = \frac{1}{G_i}$$

Equation 5-19.

Note that the i subscript is for the i-th mode, and not the i-th excitation frequency.

The values of fi and gi define pairs of frequencies and dampings. Note that gi can be entered as one of the following: structural damping (default), critical damping, or quality factor. The entered damping is converted to structural damping internally using Equation 5-19. Straight-line interpolation is used for modal frequencies between consecutive fi values. Linear extrapolation is used at the ends of the table. ENDT ends the table input.

For example, if modal damping is entered using Table 5-1 and modes exist at 1.0, 2.5, 3.6, and 5.5 Hz, NX Nastran interpolates and extrapolates as shown in Figure 5-3 and in the table. Note that there is no table entry at 1.0 Hz; NX Nastran uses the first two table entries at $f = 2.0$ and $f = 3.0$ to extrapolate the value for $f = 1.0$.
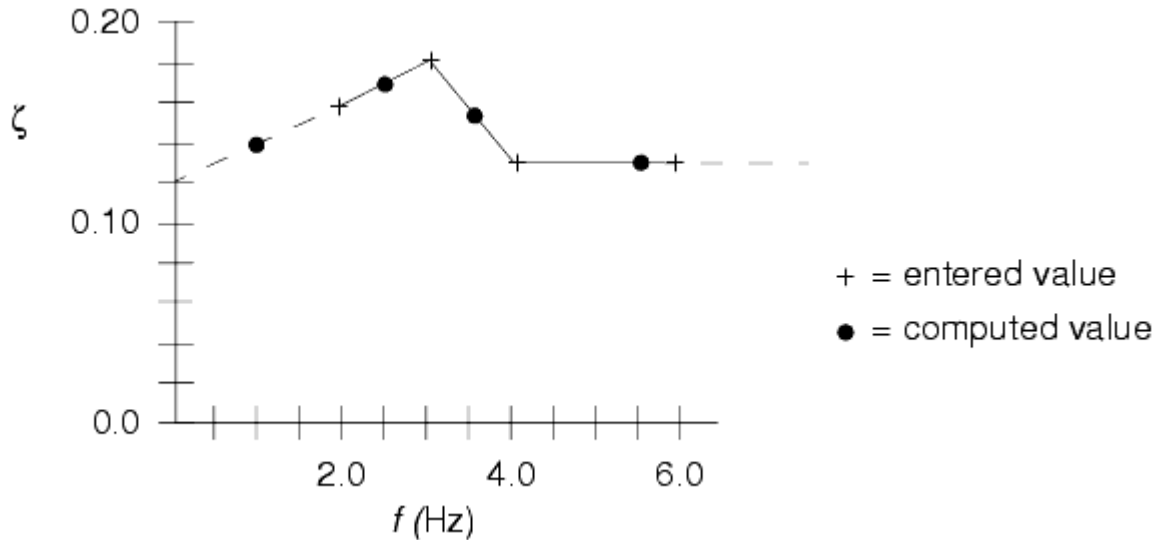

Figure 5-3. Example TABDMP1

Table 5-1. Example TABDMP1 Interpolation/Extrapolation			
Entered		**Computed**	
f	**z**	f	**z**
2.0	0.16	1.0	0.14
3.0	0.18	2.5	0.17
4.0	0.13	3.6	0.15
6.0	0.13	5.5	0.13

1	2	3	4	5	6	7	8	9	10
TABDMP1	10	CRIT							+TAB1
+TAB1	2.0	0.16	3.0	0.18	4.0	0.13	6.0	0.13	+TAB2
+TAB2	ENDT								

Modal damping is processed as a complex stiffness when PARAM,KDAMP is entered as -1. The uncoupled equation of motion becomes

$$-\omega^2 m_i \xi_i(\omega) + (1 + iG(\omega))k_i \xi_i(\omega) = p_i(\omega)$$

Equation 5-20.

The default for PARAM,KDAMP is 1, which processes modal damping as a damping matrix as shown in Equation 5-17.

The decoupled solution procedure used in modal frequency response can be used only if either no damping is present or modal damping alone (via TABDMP1) is used. Otherwise, the modal method uses the coupled solution method on the smaller modal coordinate matrices if nonmodal damping (i.e., CVISC, CDAMPi, GE on the MATi entry, or PARAM,G) is present.

Mode Truncation in Modal Frequency Response Analysis

It is possible that not all of the computed modes are required in the frequency response solution. You need to retain, at a minimum, all the modes whose resonant frequencies lie within the range of forcing frequencies. For example, if the frequency response analysis must be between 200 and 2000 Hz, all modes whose resonant frequencies are in this range should be retained. This guideline is only a minimum requirement, however. For better accuracy, all modes up to at least two to three times the highest forcing frequency should be retained. In the example where a structure is excited to between 200 and 2000 Hz, all modes from 0 to at least 4000 Hz should be retained.

The frequency range selected on the eigenvalue entry (EIGRL or EIGR) is one means to control the modes used in the modal frequency response solution. Also, three parameters are available to limit the number of modes included in the solution. PARAM,LFREQ gives the lower limit on the frequency range of retained modes, and PARAM,HFREQ gives the upper limit on the frequency range of retained modes. PARAM,LMODES gives the number of lowest modes to be retained. These parameters can be used to include the proper set of modes. Note that the default is for all computed modes to be retained.

Dynamic Data Recovery in Modal Frequency Response Analysis

In modal frequency response analysis, two options are available for recovering displacements and stresses: the mode displacement method and the matrix method. Both methods give the same answers, although with differences in cost.

The mode displacement method computes the total physical displacements for each excitation frequency from the modal displacements, and then computes element stresses from the total physical displacements. The number of operations is proportional to the number of excitation frequencies.

The matrix method computes displacements per mode and element stresses per mode, and then computes physical displacements and element stresses as the summation of modal displacements and element stresses. Costly operations are proportional to the number of modes.

Since the number of modes is usually much less that the number of excitation frequencies, the matrix method is usually more efficient and is the default. The mode displacement method can be selected by using PARAM,DDRMM,-1 in the Bulk Data. The mode displacement method is required when "frequency-frozen" structural plots are requested (see "Plotted Output").

The mode acceleration method ("Advanced Dynamic Analysis Capabilities") is another data recovery method for modal frequency response analysis. This method can provide better accuracy since detailed local stresses and forces are subject to mode truncation and may not be as accurate as the results computed with the direct method.

5.4   Modal Versus Direct Frequency Response

Some general guidelines can be used when selecting modal frequency response analysis versus direct frequency response analysis. These guidelines are summarized in Table 5-2.

Table 5-2. Modal Versus Direct Frequency Response

	Modal	**Direct**
Small Model		X
Large Model	X	
Few Excitation Frequencies		X
Many Excitation Frequencies	X	
High Frequency Excitation		X
Nonmodal Damping		X
Higher Accuracy		X

In general, larger models may be solved more efficiently in modal frequency response because the numerical solution is a solution of a smaller system of uncoupled equations. The modal method is particularly advantageous if the natural frequencies and mode shapes were computed during a previous stage of the analysis. In that case, you simply perform a restart (see "Restarts In Dynamic Analysis"). Using the modal approach to solve the uncoupled equations is very efficient, even for very large numbers of excitation frequencies. On the other hand, the major portion of the effort in a modal frequency response analysis is the calculation of the modes. For large systems with a large number of modes, this operation can be as costly as a direct solution. This result is especially true for high-frequency excitation. To capture high frequency response in a modal solution, less accurate, high-frequency modes must be computed. For small models with a few excitation frequencies, the direct method may be the most efficient because it solves the equations without first computing the modes. The direct method is more accurate than the modal method because the direct method is not concerned with mode truncation.

Table 5-2 provides an overview of which method to use. Many additional factors may be involved in the choice of a method, such as contractual obligations or local standards of practice.

5.5 Frequency-Dependent Excitation Definition

An important aspect of a frequency response analysis is the definition of the loading function. In a frequency response analysis, the force must be defined as a function of frequency. Forces are defined in the same manner regardless of whether the direct or modal method is used.

The following Bulk Data entries are used for the frequency-dependent load definition:

RLOAD1	Tabular input-real and imaginary.
RLOAD2	Tabular input-magnitude and phase.
DAREA	Spatial distribution of dynamic load.
LSEQ	Generates the spatial distribution of dynamic loads from static load entries.
DLOAD	Combines dynamic load sets.
TABLEDi	Tabular values versus frequency.
DELAY	Time delay.
DPHASE	Phase lead.

The particular entry chosen for defining the dynamic loading is largely a function of user convenience for concentrated loads. Pressure and distributed loads, however, require a more complicated format.

There are two important aspects of dynamic load definition. First, the location of the loading on the structure must be defined. Since this characteristic locates the loading in space, it is called the spatial distribution of the dynamic loading. Secondly, the frequency variation in the loading is the characteristic that differentiates a dynamic load from a static load. This frequency variation is called the temporal distribution of the load. A complete dynamic loading is a product of spatial and temporal distributions.

Using Table IDs and Set IDs in NX Nastran makes it possible to apply many complicated and temporally similar loadings with a minimum of input. Combining simple loadings to create complicated loading distributions that vary in position as well as frequency is also a straightforward task.

The remainder of this section describes the Bulk Data entries for frequency-dependent excitation. The description is given in terms of the coefficients that define the dynamic load. See the NX Nastran Quick Reference Guide for more complete Bulk Data descriptions.

Frequency-Dependent Loads – RLOAD1 Entry

The RLOAD1 entry is a general form in which to define a frequency-dependent load. It defines a dynamic loading of the form

$$\{P(f)\} = \{A[C(f) + iD(f)]e^{i\{\theta - 2\pi f\tau\}}\}$$

The values of the coefficients are defined in tabular format on a TABLEDi entry. You need not explicitly define a force at every excitation frequency. Only those values that describe the character of the loading are required. NX Nastran will interpolate for intermediate values.

1	2	3	4	5	6	7	8	9	10
RLOAD1	SID	DAREA	DELAY	DPHASE	TC	TD	TYPE		

Field	Contents
SID	Set ID defined by a DLOAD Case Control command or a DLOAD Bulk Data entry.
DAREA	Identification number of the DAREA entry set that defines A. (Integer > 0)
DELAY	Identification number of the DELAY entry set that defines τ.
DPHASE	Identification number of the DPHASE entry set that defines q.
TC	TABLEDi entry that defines C(f).
TD	TABLEDi entry that defines D(f).
TYPE	Defines the type of the dynamic excitation. (Integer, character or blank; Defaults=0)

Note that f is the frequency in cycles per unit time and that $e^{iwt} = (\cot wt + i\sin wt)$.

Frequency-Dependent Loads – RLOAD2 Entry

The RLOAD2 entry is a variation of the RLOAD1 entry used for defining a frequency-dependent load. Whereas the RLOAD1 entry defines the real and imaginary parts of the complex load, the RLOAD2 entry defines the magnitude and phase.

The RLOAD2 entry defines dynamic excitation in the form

$$\{P(f)\} = \{AB(f)e^{i\{\phi(f) + \theta - 2\pi f\tau\}}\}$$

Equation 5-21.

The RLOAD2 definition may be related to the RLOAD1 definition by

$$\underbrace{C(f) + iD(f)}_{\begin{array}{c}\text{RLOAD1}\\\text{Definition}\end{array}} = \underbrace{B(f)e^{i\phi(f)}}_{\begin{array}{c}\text{RLOAD2}\\\text{Definition}\end{array}}$$

Equation 5-22.

1	2	3	4	5	6	7	8	9	10
RLOAD2	SID	DAREA	DELAY	DPHASE	TB	TP	TYPE		

Field	Contents
SID	Set ID defined by a DLOAD Case Control command.
DAREA	Identification number of the DAREA entry set that defines A. (Integer > 0)
DELAY	Identification number of the DELAY entry set that defines t. (Integer > 0)
DPHASE	Identification number of the DPHASE entry set that defines q in degrees. (Integer > 0)
TB	TABLEDi entry defining amplitude versus frequency pairs for $B(f)$. (Integer > 0)
TP	TABLEDi entry defining phase angle versus frequency pairs for f (f) in degrees. (Integer > 0)
TYPE	Defines the type of the dynamic excitation. (Integer, character or blank; Defaults=0)

Note that *f* is the frequency in cycles per unit time.

Spatial Distribution of Loading – DAREA Entry

The DAREA entry defines the degrees-of-freedom where the dynamic load is to be applied and the scale factor to be applied to the loading. The DAREA entry provides the basic spatial distribution of the dynamic loading.

1	2	3	4	5	6	7	8	9	10
DAREA	SID	P1	C1	A1	P2	C2	A2		

Field	Contents
SID	Set ID specified by RLOADi entries.
Pi	Grid, extra, or scalar point ID.
Ci	Component number.
Ai	Scale factor.

A DAREA entry is selected by RLOAD1 or RLOAD2 entries. Any number of DAREA entries may be used; all those with the same SID are combined.

Time Delay – DELAY Entry

The DELAY entry defines the time delay t in an applied load.

1	2	3	4	5	6	7	8	9	10
DELAY	SID	P1	C1		P2	C2	t_2		

Field	Contents
SID	Set ID specified by an RLOADi entry.
Pi	Grid, extra, or scalar point ID.
Ci	Component number.
t_i	Time delay for Pi, Ci. (Default = 0.0)

A DAREA entry must be defined for the same point and component. Any number of DELAY entries may be used; all those with the same SID are combined.

Phase Lead – DPHASE Entry

The DPHASE entry defines the phase lead q .

1	2	3	4	5	6	7	8	9	10
DPHASE	SID	P1	C1	q_1	P2	C2	q_2		

Field	Contents
SID	Set ID specified by an RLOADi entry.
Pi	Grid, extra, or scalar point ID.
Ci	Component number.
q_i	Phase lead (in degrees) for Pi, Ci. (Default = 0.0)

A DAREA entry must be defined for the same point and component. Any number of DPHASE entries may be used; all those with the same SID are combined.

Dynamic Load Tabular Function – TABLEDi Entries

The TABLEDi entries (i = 1 through 4) each define a tabular function for use in generating frequency-dependent dynamic loads. The form of each TABLEDi entry varies slightly, depending on the value of i, as does the algorithm for y(x). The x values need not be evenly spaced.

The TABLED1, TABLED2, and TABLED3 entries linearly interpolate between the end points and linearly extrapolate outside of the endpoints, as shown in Figure 5-4. The TABLED1 entry also performs logarithmic interpolation between points. The TABLED4 entry assigns the endpoint values to any value beyond the endpoints.
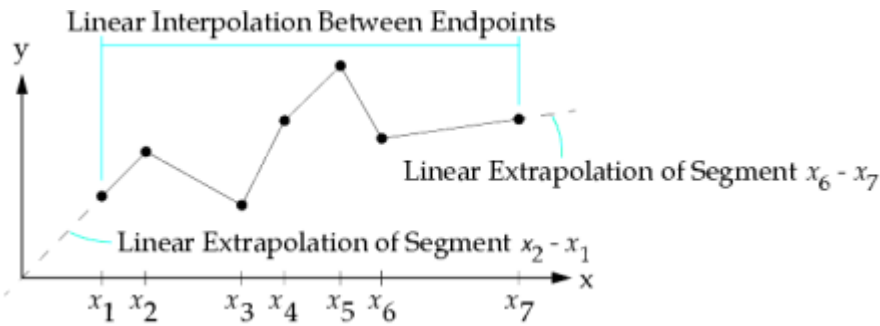
Figure 5-4. Interpolation and Extrapolation for TABLED1, TABLED2, and TABLED3 Entries

The TABLED1 entry has the following format:

1	2	3	4	5	6	7	8	9	10
TABLED1	TID	XAXIS	YAXIS						
	x1	y1	x2	y2	x3	y3	-etc.-	ENDT	

Field	Contents
TID	Table identification number.
XAXIS	Specifies a linear or logarithmic interpolation for the x-axis. (Character: "LINEAR" or "LOG"; default = "LINEAR")
YAXIS	Specifies a linear or logarithmic interpolation for the y-axis. (Character: "LINEAR" or "LOG"; default = "LINEAR")
xi, yi	Tabular values. Values of x are frequency in cycles per unit time.
ENDT	Ends the table input.

The TABLED1 entry uses the algorithm

$$y = y_T(x)$$

Equation 5-23.

The algorithms used for interpolation and extrapolation are as follows:

XAXIS	YAXIS	$y_T(x)$
LINEAR	LINEAR	$\dfrac{xj - x}{xj - xi} yi + \dfrac{x - xi}{xj - xi} yj$
LOG	LINEAR	$\dfrac{\ln(xj / x)}{\ln(xj / xi)} yi + \dfrac{\ln(x / xi)}{\ln(xj / xi)} yj$
LINEAR	LOG	$\exp\left[\dfrac{xj - x}{xj - xi} \ln yi + \dfrac{x - xi}{xj - xi} \ln yj\right]$

XAXIS	YAXIS	$y_T(x)$
LOG	LOG	$\exp\left[\dfrac{\ln(xj/x)}{\ln(xj/xi)}\ln yi + \dfrac{\ln(x/xi)}{\ln(xj/xi)}yj\right]$

The TABLED2 entry has the following format:

1	2	3	4	5	6	7	8	9	10
TABLED2	TID	X1							
	x1	y1	x2	y2	x3	y3	-etc.-	ENDT	

Field	Contents
TID	Table identification number.
X1	Table parameter.
xi, yi	Tabular values. Values of x are frequency in cycles per unit time.

The TABLED2 entry uses the algorithm

$$y = y_T(x - X1)$$

Equation 5-24.

ENDT ends the table input.

The TABLED3 entry has the following format:

1	2	3	4	5	6	7	8	9	10
TABLED3	TID	X1	X2						
	x1	y1	x2	y2	x3	y3	-etc.-	ENDT	

Field	Contents
TID	Table identification number.
X1, X2	Table parameters (X2≠ 0.0)
xi, yi	Tabular values. Values of x are frequency in cycles per unit time.

The TABLED3 entry uses the algorithm

$$y = y_T\left(\frac{x - X1}{X2}\right)$$

Equation 5-25.

ENDT ends the table input.

The TABLED4 entry has the following format:

1	2	3	4	5	6	7	8	9	10
TABLED4	TID	X1	X2	X3	X4				

1	2	3	4	5	6	7	8	9	10
	A0	A1	A2	A3	A4	A5	-etc.-	ENDT	

Field	Contents
TID	Table identification number.
Xi	Table parameters (X2 ≠0.0; X3 < X4).
Ai	Coefficients.

The TABLED4 entry uses the algorithm

$$y = \sum_{i=0}^{N} A i \left(\frac{x - X1}{X2} \right)^{i}$$

Equation 5-26.

N is the degree of the power series. When x < X3, X3 is used for x; when x > X4, X4 is used for x. This condition has the effect of placing bounds on the table; there is no extrapolation outside of the table boundaries.

ENDT ends the table input.

DAREA Example

Suppose the following command is in the Case Control Section:

DLOAD = 35

in addition to the following entries in the Bulk Data Section:

1	2	3	4	5	6	7	8	9	10
$RLOAD1	SID	DAREA	DELAY	DPHASE	TC	TD			
RLOAD1	35	29	31		40				
$DAREA	SID	POINT	COMPONENT	SCALE					
DAREA	29	30	1	5.2					
$DELAY	SID	POINT	COMPONENT	LAG					
DELAY	31	30	1	0.2					
$TABLED1	ID	XAXIS	YAXIS						
$	x1	y1	x2	y2	x3	y3	x4	y4	
TABLED1	40	LINEAR	LINEAR						
	0.0	4.0	2.0	8.0	6.0	8.0	ENDT		

The DLOAD Set ID 35 in Case Control selects the RLOAD1 entry in the Bulk Data having a Set ID 35. On the RLOAD1 entry is a reference to DAREA Set ID 29, DELAY Set ID 31, and TABLED1 Set ID 40. The DAREA entry with Set ID 29 positions the loading on grid point 30 in the 1 direction with a scale factor of 5.2 applied to the load. The DELAY entry with Set ID 31 delays the loading on grid point 30 in the 1 direction by 0.2 units of time. The TABLED1 entry with Set ID 40 defines the load in tabular form. This table is shown graphically in Figure 5-5.

The result of these entries is a dynamic load applied to grid point 30, component T1, scaled by 5.2 and delayed by 0.2 units of time.
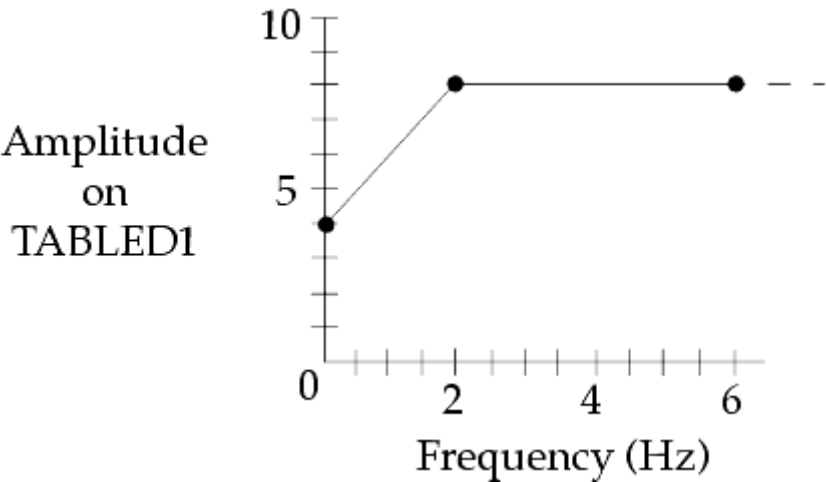

Figure 5-5. TABLED1 - Amplitude Versus Frequency

Static Load Sets – LSEQ Entry

NX Nastran does not have specific data entries for many types of dynamic loads. Only concentrated forces and moments can be specified directly using DAREA entries. To accommodate more complicated loadings conveniently, the LSEQ entry is used to define static load entries that define the spatial distribution of dynamic loads.

1	2	3	4	5	6	7	8	9	10
LSEQ	SID	DAREA ID	LOAD ID	TEMP ID					

The LSEQ Bulk Data entry contains a reference to a DAREA Set ID and a static Load Set ID. The static loads are combined with any DAREA entries in the referenced set. The DAREA Set ID does not need to be defined with a DAREA Bulk Data entry. The DAREA Set ID is referenced by an RLOADi entry. This reference defines the temporal distribution of the dynamic loading. The Load Set ID may refer to one or more static load entries (FORCE, PLOADi, GRAV, etc.). All static loads with the Set ID referenced on the LSEQ entry define the spatial distribution of the dynamic loading. NX Nastran converts this information to equivalent dynamic loading.

Figure 5-6 demonstrates the relationships of these entries. To activate a load set defined in this manner, the DLOAD Case Control command refers to the Set ID of the selected DLOAD or RLOADi entry, and the LOADSET Case Control command refers to the Set ID of the selected LSEQ entries. The LSEQ entries point to the static loading entries that are used to define dynamic loadings with DAREA Set ID references. Together this relationship defines a complete dynamic loading. To apply dynamic loadings in this manner, the DLOAD and LOADSET Case Control commands and the RLOADi and LSEQ Bulk Data entries must be defined. A DAREA Bulk Data entry does not need to be defined since the RLOADi and LSEQ entries reference a common DAREA ID. The LSEQ entry can also be interpreted as an internal DAREA entry generator for static load entries.
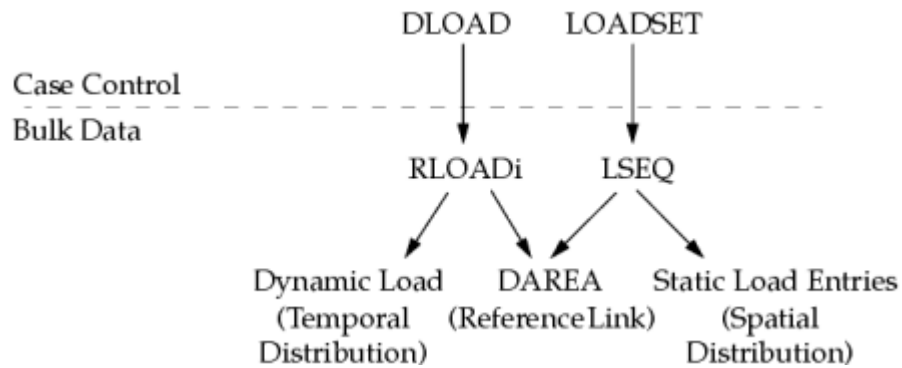
Figure 5-6. Relationship of Dynamic and Static Load Entries

LSEQ Example

Suppose the following commands are in the Case Control Section:

LOADSET = 27

DLOAD = 25

and the following entries are in the Bulk Data Section:

1	2	3	4	5	6	7	8	9	10
$LSEQ	SID	DAREA	LID						
LSEQ	27	28	26						

$RLOAD1	SID	DAREA	DELAY	DPHASE	TC	TD			
RLOAD1	25	28			29				

$STATIC LOAD	SID								
PLOAD1	26	etc.							
FORCE	26	etc.							

$TABLED1	TID								
TABLED1	29	etc.							

In the above, the LOADSET request in Case Control selects the LSEQ Set ID 27 entry. The DLOAD request in Case Control selects the RLOAD1 Set ID 25 entry. This RLOAD1 entry refers to a TABLED1 ID 29, which is used to define the frequency-dependent variation in the loading. DAREA Set ID 28 links the LSEQ and RLOAD1 entries. In addition, the LSEQ entry refers to static Load Set ID 26, which is defined by FORCE and PLOAD1 entries. The FORCE and PLOAD1 entries define the spatial distribution of the dynamic loading and through the DAREA link refer to the RLOAD1/TABLED1 combination for the frequency-varying characteristics of the load. Note that there is no DAREA entry.

Dynamic Load Set Combination – DLOAD

One of the requirements of frequency-dependent loads is that RLOAD1s and RLOAD2s must have unique SIDs. If they are to be applied in the same analysis, they must be combined using

the DLOAD Bulk Data entry. The total applied load is constructed from a linear combination of component load sets as follows:

$$\{P\} \;=\; S\sum_{i}S_{i}\{P_{i}\}$$

Equation 5-27.

where:

S	=	overall scale factor
S_i	=	scale factor for the i-th load set
P_i	=	i-th load set
P	=	total applied load

The DLOAD Bulk Data entry has the following format:

1	2	3	4	5	6	7	8	9	10
DLOAD	SID	S	S1	L1	S2	L2			

	Field	**Contents**
	SID	Load Set ID.
	S	Overall scale factor.
	Si	Individual scale factors.
	Li	Load Set ID numbers for RLOAD1 and RLOAD2 entries.

As an example, in the following DLOAD entry

$DLOAD	SID	S	S1	L1	S2	L2	-etc.-		
DLOAD	33	3.25	0.5	14	2.0	27			

a dynamic Load Set ID of 33 is created by taking 0.5 times the loads in the Load Set ID of 14, adding to it 2.0 times the loads in the Load Set ID of 27, and multiplying that sum by an overall scale factor of 3.25.

As with other frequency-dependent loads, a dynamic load combination defined by the DLOAD Bulk Data entry is selected by the DLOAD Case Control command.

5.6   Solution Frequencies

A major consideration when you conduct a frequency response analysis is selecting the frequency at which the solution is to be performed. There are six Bulk Data entries that you can use to select the solution frequencies. It is important to remember that each specified frequency results in an independent solution at the specified excitation frequency.

To select the loading frequencies, use the FREQ, FREQ1, FREQ2, FREQ3, FREQ4 and FREQ5 Bulk Data entries.

FREQ   Defines discrete excitation frequencies.

FREQ1
: Defines a starting frequency F_{start}, a frequency incitement Δf, and the number of frequency increments to solve NDF.

FREQ2
: Defines a starting frequency F_{start}, and ending frequency F_{end}, and the number of logarithmic intervals, NF, to be used in the frequency range.

FREQ3[1]
: Defines the number of excitation frequencies used between modal pairs in a given range.

FREQ4[1]
: Defines excitation frequencies using a spread about each normal mode within a range.

FREQ5[1]
: Defines excitation frequencies as all frequencies in a given range as a defined fraction of the normal modes.

[1]Used for modal solution only.

The FREQUENCY Case Control command selects FREQi Bulk Data entries. All FREQi entries with the same selected Set ID are applied in the analysis; therefore, you can use any combination of FREQ, FREQ1, FREQ2, FREQ3, FREQ4 and FREQ5 entries.

The examples that follow show the formats of the FREQi entries. Notice that the six sets of excitation frequencies shown in the examples will be combined in a single analysis if the Set IDs are identical.

FREQ

This FREQ entry specifies ten specific (unequally spaced loading frequencies to be analyzed.

1	2	3	4	5	6	7	8	9	10
$FREQ	SID	F	F	F	F	F	F	F	
$	F	F	F	F	F	F	F	F	
FREQ	3	2.98	3.05	17.9	21.3	25.6	28.8	31.2	
	29.2	22.4	19.3						

Field	Contents
SID	Set ID specified by a FREQUENCY Case Control command.
F	Frequency value (cycles per unit time).

FREQ1

The FREQ1 example specifies 14 frequencies between 2.9 Hz and 9.4 Hz in increments of 0.5 Hz.

$FREQ1	SID	F_{start}	Δf	NDF					
FREQ1	6	2.9	0.5	13					

Field	Contents
SID	Set ID specified by a FREQUENCY Case Control command.
F_{start}	Starting frequency in set (cycles per unit time).
Δf	Frequency increment (cycles per unit time).
NDF	Number of frequency increments.

FREQ2

The FREQ2 example specifies six logarithmic frequency intervals between 1.0 and 8.0 Hz, resulting in frequencies at 1.0, 1.4142, 2.0, 2.8284, 4.0, 5.6569, and 8.0 Hz being used for the analysis.

1	2	3	4	5	6	7	8	9	10
$FREQ2	SID	F_{start}	F_{end}	NF					
FREQ2	9	1.0	8.0	6					

Field	Contents
SID	Set ID specified by a FREQUENCY Case Control command.
F_{start}	Starting frequency (cycles per unit time).
F_{end}	Ending frequency (cycles per unit time).
NF	Number of logarithmic intervals.

FREQ3

The FREQ3 example requests 10 frequencies between each set of modes within the range 20 and 2000, plus ten frequencies between 20 and the lowest mode in the range, plus 10 frequencies between the highest mode in the range and 2000.

1	2	3	4	5	6	7	8	9	10
$FREQ3	SID	F1	F2	TYPE	NEF	CLUSTER			
FREQ3	6	20.0	2000.0	LINEAR	10	2.0			

Field	Contents
SID	Set ID specified by a FREQUENCY Case Control command.
F1	Lower bound of modal frequency range in cycles per unit time. (Real > 0.0)
F2	Upper bound of modal frequency range in cycles per unit time. (Real > 0.0, F2 ≥F1, Default = F1)
TYPE	LINEAR or LOG. Specifies linear or logarithmic interpolation between frequencies. (Character; Default = "LINEAR")
NEF	Number of excitation frequencies within each subrange including the end points. (Integer > 1, Default = 10)
CLUSTER	A CLUSTER value greater than 1 provides closer spacing of excitation frequencies near the modal frequencies, where greater resolution is needed. (Real > 0.0; Default = 1.0)

FREQ4

The example FREQ4 chooses 21 equally spaced frequencies across a frequency band of 0.7 $\cdot f_N$ to 1.3 $\cdot f_N$ for each natural frequency between 20 and 2000.

1	2	3	4	5	6	7	8	9	10
$FREQ4	SID	F1	F2	FSPD	NFM				
FREQ4	6	20.0	2000.0	0.30	21				

Field	Contents
SID	Set ID specified by a FREQUENCY Case Control command.
F1	Lower bound of modal frequency range in cycles per unit time. (Real > 0.0)
F2	Upper bound of modal frequency range in cycles per unit time. (Real > 0.0, F2 ≥F1, Default = F1)
FSPD	Frequency spread, +/– the fractional amount specified for each mode which occurs in the frequency range F1 to F2. (1.0 > Real > 0.0, Default = 0.10)
NFM	Number of evenly spaced frequencies per "spread" mode. (Integer > 0; Default = 3; If NFM is even, NFM + 1 will be used.)

FREQ5

The example FREQ5 will compute excitation frequencies which are 0.6, 0.8, 0.9, 0.95, 1.0, 1.05, 1.1, and 1.2 times the natural frequencies for all natural frequencies, but use only the computed frequencies that fall within the range 20 and 2000.

1	2	3	4	5	6	7	8	9	10
$FREQ5	SID	F1	F2	FR1	FR2	FR3	FR4	FR5	
	FR6	FR7	-etc.-						
FREQ5	6	20.0	2000.0	1.0	0.6	0.8	0.9	0.95	
	1.05	1.1	1.2						

Field	Contents
SID	Set ID specified by a FREQUENCY Case Control command.
F1	Lower bound of modal frequency range in cycles per unit time. (Real > 0.0)
F2	Upper bound of modal frequency range in cycles per unit time. (Real > 0.0, F2 ≥ F1, Default = F1)
FRi	Fractions of the natural frequencies in the range F1 to F2. (Real > 0.0)

5.7   Frequency Response Considerations

Exciting an undamped (or modal or viscous damped) system at 0.0 Hz using direct frequency response analysis gives the same results as a static analysis and also gives almost the same results when using modal frequency response (depending on the number of retained modes). Therefore, if the maximum excitation frequency is much less than the lowest resonant frequency of the system, a static analysis is probably sufficient.

Undamped or very lightly damped structures exhibit large dynamic responses for excitation frequencies near resonant frequencies. A small change in the model (or running it on another computer) may result in large changes in such responses.

Use a fine enough frequency step size (Δf) to adequately predict peak response. Use at least five points across the half-power bandwidth (which is approximately $2xf_n$ for an SDOF system) as shown in .
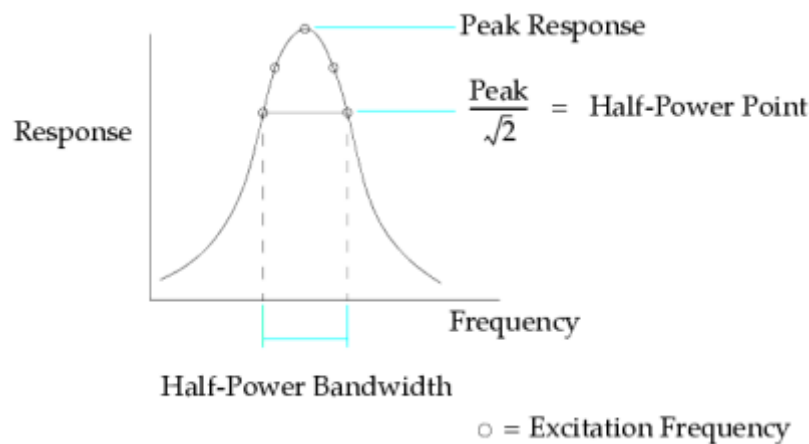
Figure 5-7. Half-Power Bandwidth

For maximum efficiency, an uneven frequency step size should be used. Smaller frequency spacing should be used in regions near resonant frequencies, and larger frequency step sizes should be used in regions away from resonant frequencies.

5.8   Solution Control for Frequency Response Analysis

The following tables summarize the data entries that can be used to control a frequency response analysis. Certain data entries are required, some data entries are optional, and others are user selectable.

In the Executive Control Section of the NX Nastran input file, a solution must be selected using the SOL i statement where i is an integer value chosen from Table 5-3.

Table 5-3. Frequency Response Solutions in NX Nastran

	Rigid Formats	**Structured Solution Sequences**
Direct	8	108
Modal	11	111

Solutions 108 and 111 are the preferred SOLs; these are the ones used in the examples that follow.

In the Case Control Section of the NX Nastran input file, you must select the solution parameters associated with the current analysis (i.e., frequencies, loads, and boundary conditions), and also the output quantities required from the analysis. The Case Control commands directly related to frequency response analysis are listed in Table 5-4. They can be combined in the standard fashion with the more generic entries, such as SPC, MPC, etc.

Table 5-4. Case Control Commands for Frequency Response Solution Control

Case Control Command	**Direct or Modal**	**Description**	**Required/Optional**
DLOAD	Both	Select the dynamic load set from Bulk Data	Required

Table 5-4. Case Control Commands for Frequency Response Solution Control

Case Control Command	Direct or Modal	Description	Required/Optional
FREQUENCY	Both	Select FREQi entries from Bulk Data	Required
METHOD	Modal	Select the eigenvalue extraction parameters	Required
LOADSET	Both	Select the LSEQ set from Bulk Data	Optional
SDAMPING	Modal	Select the modal damping table from Bulk Data	Optional
OFREQUENCY	Both	Select the frequencies for output (default = all)	Optional

The types of results available from a frequency response analysis are similar to those for a typical static analysis except that the results are a complex function of the applied loading frequency. Additional quantities (characteristic of dynamic problems) are also available. The output quantities are summarized in Table 5-5 and Table 5-6.

Table 5-5. Grid Output from a Frequency Response Analysis

Case Control Command	Description
ACCELERATION	Grid point acceleration results for a set of grid points.
DISPLACEMENT (or VECTOR)	Grid point displacement results for a set of grid points.
OLOAD	Requests the applied load table to be output for a set of grid points.
SACCELERATION	Requests the solution set acceleration output: d-set in direct solutions and modal variables in modal solutions.
SDISPLACEMENT	Requests the solution set displacement output: d-set in direct solutions and modal variables in modal solutions.
SVECTOR	Requests the real eigenvector output for the a-set in modal solutions.
SVELOCITY	Requests the solution set velocity output: d-set in direct solutions and modal variables in modal solutions.
SPCFORCES	Requests the forces of a single-point constraint for a set of grid points.
VELOCITY	Grid point velocity results for a set of grid points.

Frequency response output is in real/imaginary format (the default) or magnitude/phase format (the phase angle is in degrees). Frequency response output is also in SORT1 or SORT2 format. In SORT1 format, the results are listed by frequency; in SORT2 format, the results are listed by grid point or element number. SORT1 is the default for direct frequency response analysis (SOL 108), and SORT2 is the default for modal frequency response analysis (SOL 111). PARAM,CURVPLOT,1 and PARAM,DDRMM,-1 are necessary to obtain SORT1 output in SOL 111. These output formats are specified with the Case Control commands. The command

DISPLACEMENT(PHASE, SORT2) = ALL

prints displacements in magnitude/phase and SORT2 formats. The output formats are illustrated in the first example in "Examples" .

Table 5-6. Element Output from a Frequency Response Analysis

Case Control Command	Description
ELSTRESS (or STRESS)	Element stress results for a set of elements.
ELFORCE (or FORCE)	Element force results for a set of elements.
STRAIN	Element strain results for a set of elements.

A number of Bulk Data entries are unique to frequency response analysis. These entries can be combined with other generic entries in the Bulk Data. Bulk Data entries directly related to frequency response analysis are summarized in Table 5-7.

Table 5-7. Bulk Data Entries for Frequency Response Analysis

Bulk Data Entry	Direct or Modal	Description	Required/ Optional
FREQ, FREQi	Both	Excitation frequencies	Required
RLOADi	Both	Dynamic loading	Required
EIGR or EIGRL	Modal	Eigenvalue analysis parameters	Required
LSEQ	Both	Dynamic loading from static loads	Optional
TABLEDi	Both	Frequency-dependent tables	Both[1]
DAREA	Both	Load component and scale factor	Required[2]
DELAY	Both	Time delay on dynamic load	Optional
DPHASE	Both	Phase angle on dynamic load	Optional
DLOAD	Both	Dynamic load combination, required if RLOAD1 and RLOAD2 are used	Optional
TABDMP1	Modal	Modal damping table	Optional

[1]Required for RLOAD1; optional for RLOAD2.

[2]The DAREA ID is required; the DAREA Bulk Data entry is not required if an LSEQ entry is used.

5.9   Examples

This section provides several examples showing the input and output. These examples are:

Model	Frequency Response Bulk Data Entries	Output
bd05two	EIGRL, FREQ1, TABDMP1, RLOAD1, DAREA, TABLED1	X-Y plots (linear), printed results (SORT1, SORT2)
bd05bar	EIGRL, FREQ1, TABDMP1, DLOAD, RLOAD2, DAREA, DPHASE, TABLED1	X-Y plots (log)
bd05bkt	EIGRL, FREQ1, TABDMP1, RLOAD1, LSEQ, TABLED1, PLOAD4	X-Y plot (log)

These examples are described in the sections that follow.

Two-DOF Model

Consider the two-DOF system shown in Figure 5-8. Modal frequency response (SOL 111) is run with a 20 N load applied to the primary mass (grid point 2) across a frequency range of 2 to 10 Hz with an excitation frequency increment of 0.05 Hz. Uniform modal damping of 5% critical damping is used. Part of the input file is shown below.
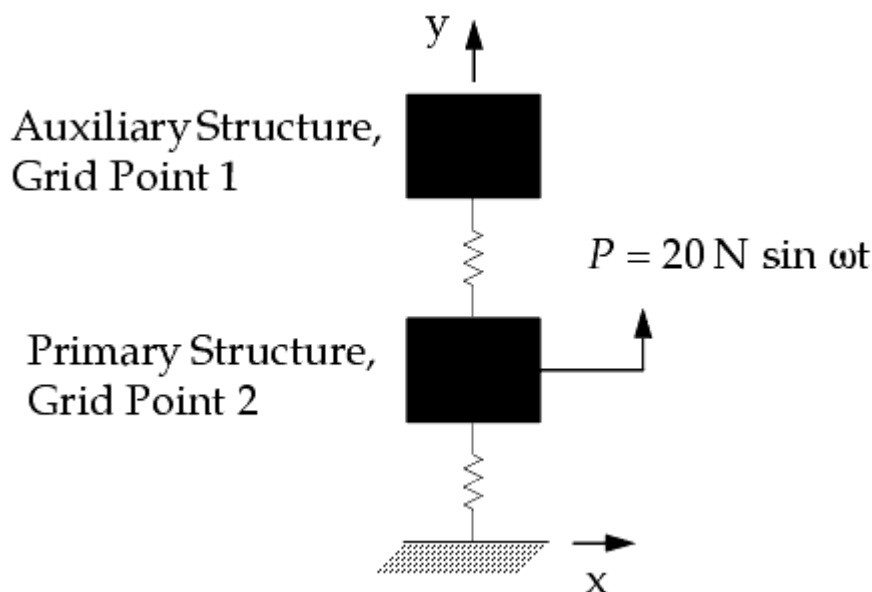

Figure 5-8. Two-DOF Model

```
$   FILE bd05two.dat
$
$   TWO-DOF SYSTEM
$   CHAPTER 5, FREQUENCY RESPONSE
$
TIME 5
SOL 111      $  MODAL FREQUENCY RESPONSE
CEND
TITLE = TWO-DOF SYSTEM
SUBTITLE = MODAL FREQUENCY RESPONSE
LABEL =   20 N FORCE APPLIED TO PRIMARY MASS
$
$   SPECIFY SPC
SPC = 996
$
$   SPECIFY MODAL EXTRACTION
METHOD = 10
$
$   SPECIFY DYNAMIC INPUT
DLOAD = 999
FREQ = 888
SDAMPING = 777
$
$   SELECT OUTPUT
SET 11 = 1,2
DISPLACEMENT(PHASE,PLOT) = 11
$
$   XYPLOTS
$
... X-Y plot commands ...
$
BEGIN BULK
$
$.......2.......3.......4.......5.......6.......7.......8.......9.......10.....$
$
$   ENTRIES FOR FREQUENCY RESPONSE
$
$   LOAD DEFINITION
$RLOAD1 SID     DAREA                   TC
```

```
RLOAD1  999     997                             901
$DAREA  SID     P1      C1      A1
DAREA   997     2       2       20.0
$TABLED1 TID                                                            +TABL1
$+TABL1 X1      Y1      X2      Y3      ETC.
TABLED1 901                                                             +TAB901
+TAB901 0.0     1.0     10.0    1.0     ENDT
$
$  ALTERNATE LOAD DEFINITION USING DLOAD
$DLOAD   SID     S       S1      RLOAD1
$DLOAD   999     1.0     1.0     998
$RLOAD1 SID     DAREA                   TC
$RLOAD1 998     997                     901
$
$  FREQUENCY RANGE 2-10 HZ
$FREQ1   SID     F1      DF      NDF
FREQ1    888     2.      0.05    160
$
$  MODAL DAMPING OF 5% CRITICAL
$TABDMP1 TID    TYPE                                                    +TABD1
$+TABD1 F1      G1      F2      G2      ETC.
TABDMP1 777     CRIT                                                    +TABD7
+TABD7  0.      0.05    100.    0.05    ENDT
$
$ MODAL EXTRACTION
$EIGRL   SID     V1      V2      ND      MSGLVL
EIGRL    10      -0.1    20.             0
$
... basic model ...
$
ENDDATA
```

Figure 5-9. Input File (Abridged) for the Two-DOF Example

Table 5-8 shows the relationship between the Case Control commands and the Bulk Data entries. Note that the RLOAD1 entry references the DAREA and TABLED1 entries. The input file also shows an alternate way to specify the dynamic load, by using a DLOAD Bulk Data entry. Because there is only a single RLOAD1 entry, the DLOAD Bulk Data entry is not required.

Table 5-8. Relationship Between the Case Control Commands and Bulk Data Entries for the Two-DOF Model

Case Control	Bulk Data
METHOD	EIGRL
FREQUENCY	FREQ1
SDAMPING	TABDMP1
DLOAD	RLOAD1 → DAREA, TABLED1

The RLOAD1 entry describes a sinusoidal load in the form

$$\{P(f)\} = \{A[C(f) + iD(f)]e^{i\{\theta - 2\pi f \tau\}}\}$$

Equation 5-28.

where:

A = 20.0 (entered on the DAREA entry)

C = 1.0 for all frequencies entered on the TABLED1 entry

D = 0.0 (field 7 of the RLOAD1 entry is blank)

q = 0.0 (field 5 of the RLOAD1 entry is blank)

t = 0.0 (field 4 of the RLOAD1 entry is blank)

Output can be printed in either real/imaginary or magnitude/phase format and in either SORT1 or SORT2 format. These formats are illustrated in Figure 5-10, Figure 5-11, and Figure 5-12 showing a portion of their printed output.

```
    POINT-ID =       1
                                  C O M P L E X   D I S P L A C E M E N T   V E C T O R
                                              (REAL/IMAGINARY)

     FREQUENCY    TYPE          T1                 T2                T3                R1                R2                R3
  2.000000E+00     G      0.0            2.813052E-03      0.0               0.0               0.0               0.0
                          0.0           -2.107985E-04      0.0               0.0               0.0               0.0
  2.050000E+00     G      0.0            2.866642E-03      0.0               0.0               0.0               0.0
                          0.0           -2.229164E-04      0.0               0.0               0.0               0.0
  2.100000E+00     G      0.0            2.923141E-03      0.0               0.0               0.0               0.0
                          0.0           -2.358382E-04      0.0               0.0               0.0               0.0
  2.150000E+00     G      0.0            2.982732E-03      0.0               0.0               0.0               0.0
                          0.0           -2.496362E-04      0.0               0.0               0.0               0.0
  2.200000E+00     G      0.0            3.045609E-03      0.0               0.0               0.0               0.0
                          0.0           -2.643908E-04      0.0               0.0               0.0               0.0

    POINT-ID =       2
                                  C O M P L E X   D I S P L A C E M E N T   V E C T O R
                                              (REAL/IMAGINARY)

     FREQUENCY    TYPE          T1                 T2                T3                R1                R2                R3
  2.000000E+00     G      0.0            2.374954E-03      0.0               0.0               0.0               0.0
                          0.0           -1.129933E-04      0.0               0.0               0.0               0.0
  2.050000E+00     G      0.0            2.397706E-03      0.0               0.0               0.0               0.0
                          0.0           -1.180853E-04      0.0               0.0               0.0               0.0
  2.100000E+00     G      0.0            2.421475E-03      0.0               0.0               0.0               0.0
                          0.0           -1.234173E-04      0.0               0.0               0.0               0.0
  2.150000E+00     G      0.0            2.446311E-03      0.0               0.0               0.0               0.0
                          0.0           -1.290072E-04      0.0               0.0               0.0               0.0
  2.200000E+00     G      0.0            2.472262E-03      0.0               0.0               0.0               0.0
                          0.0           -1.348744E-04      0.0               0.0               0.0               0.0
```

Figure 5-10. Real/Imaginary Output in SORT2 Format

```
    POINT-ID =       1
                                  C O M P L E X   D I S P L A C E M E N T   V E C T O R
                                              (MAGNITUDE/PHASE)

     FREQUENCY    TYPE          T1                 T2                T3                R1                R2                R3
  2.000000E+00     G      0.0            2.820939E-03      0.0               0.0               0.0               0.0
                          0.0           355.7145           0.0               0.0               0.0               0.0
  2.050000E+00     G      0.0            2.875296E-03      0.0               0.0               0.0               0.0
                          0.0           355.5535           0.0               0.0               0.0               0.0
  2.100000E+00     G      0.0            2.932640E-03      0.0               0.0               0.0               0.0
                          0.0           355.3874           0.0               0.0               0.0               0.0
  2.150000E+00     G      0.0            2.993161E-03      0.0               0.0               0.0               0.0
                          0.0           355.2159           0.0               0.0               0.0               0.0
  2.200000E+00     G      0.0            3.057064E-03      0.0               0.0               0.0               0.0
                          0.0           355.0386           0.0               0.0               0.0               0.0

    POINT-ID =       2
                                  C O M P L E X   D I S P L A C E M E N T   V E C T O R
                                              (MAGNITUDE/PHASE)

     FREQUENCY    TYPE          T1                 T2                T3                R1                R2                R3
  2.000000E+00     G      0.0            2.377640E-03      0.0               0.0               0.0               0.0
                          0.0           357.2761           0.0               0.0               0.0               0.0
  2.050000E+00     G      0.0            2.400612E-03      0.0               0.0               0.0               0.0
                          0.0           357.1805           0.0               0.0               0.0               0.0
  2.100000E+00     G      0.0            2.424619E-03      0.0               0.0               0.0               0.0
                          0.0           357.0823           0.0               0.0               0.0               0.0
  2.150000E+00     G      0.0            2.449710E-03      0.0               0.0               0.0               0.0
                          0.0           356.9813           0.0               0.0               0.0               0.0
  2.200000E+00     G      0.0            2.475939E-03      0.0               0.0               0.0               0.0
                          0.0           356.8773           0.0               0.0               0.0               0.0
```

Figure 5-11. Magnitude/Phase Output in SORT2 Format

```
  FREQUENCY =  2.000000E+00
                                  C O M P L E X   D I S P L A C E M E N T   V E C T O R
                                              (REAL/IMAGINARY)

     POINT ID.   TYPE          T1                 T2                T3                R1                R2                R3
         1         G      0.0            2.813051E-03      0.0               0.0               0.0               0.0
                          0.0           -2.107985E-04      0.0               0.0               0.0               0.0
         2         G      0.0            2.374954E-03      0.0               0.0               0.0               0.0
                          0.0           -1.129933E-04      0.0               0.0               0.0               0.00

  FREQUENCY =  2.050000E+00
```

```
                              C O M P L E X   D I S P L A C E M E N T   V E C T O R
                                                 (REAL/IMAGINARY)
    POINT ID.   TYPE          T1           T2              T3            R1            R2            R3
        1        G     0.0          2.866640E-03   0.0           0.0           0.0           0.0
                       0.0         -2.229163E-04   0.0           0.0           0.0           0.0
        2        G     0.0          2.397706E-03   0.0           0.0           0.0           0.0
                       0.0         -1.180853E-04   0.0           0.0           0.0           0.00
    FREQUENCY =  2.100000E+00
                              C O M P L E X   D I S P L A C E M E N T   V E C T O R
                                                 (REAL/IMAGINARY)
    POINT ID.   TYPE          T1           T2              T3            R1            R2            R3
        1        G     0.0          2.923141E-03   0.0           0.0           0.0           0.0
                       0.0         -2.358381E-04   0.0           0.0           0.0           0.0
        2        G     0.0          2.421475E-03   0.0           0.0           0.0           0.0
                       0.0         -1.234173E-04   0.0           0.0           0.0           0.00
    FREQUENCY =  2.150000E+00
                              C O M P L E X   D I S P L A C E M E N T   V E C T O R
                                                 (REAL/IMAGINARY)
    POINT ID.   TYPE          T1           T2              T3            R1            R2            R3
        1        G     0.0          2.982731E-03   0.0           0.0           0.0           0.0
                       0.0         -2.496362E-04   0.0           0.0           0.0           0.0
        2        G     0.0          2.446311E-03   0.0           0.0           0.0           0.0
                       0.0         -1.290072E-04   0.0           0.0           0.0           0.00
    FREQUENCY =  2.200000E+00
                              C O M P L E X   D I S P L A C E M E N T   V E C T O R
                                                 (REAL/IMAGINARY)
    POINT ID.   TYPE          T1           T2              T3            R1            R2            R3
        1        G     0.0          3.045608E-03   0.0           0.0           0.0           0.0
                       0.0         -2.643907E-04   0.0           0.0           0.0           0.0
        2        G     0.0          2.472263E-03   0.0           0.0           0.0           0.0
                       0.0         -1.348744E-04   0.0           0.0           0.0           0.0
```

Figure 5-12. Real/Imaginary Output in SORT1 Format

Figure 5-13 shows the plots of the resulting displacement magnitudes for grid points 1 and 2. Note that the response for grid point 1 is nearly an order of magnitude larger than that of grid point 2. This large difference in response magnitudes is characteristic of dynamic absorbers (also called tuned mass dampers), in which an auxiliary structure (i.e., the small mass and stiffness) is attached to the primary structure in order to decrease the dynamic response of the primary structure. If this same model is rerun without the auxiliary structure, the response of the primary structure (grid point 2) at 5.03 Hz is twice what it was with the auxiliary structure attached, as shown in Figure 5-14.
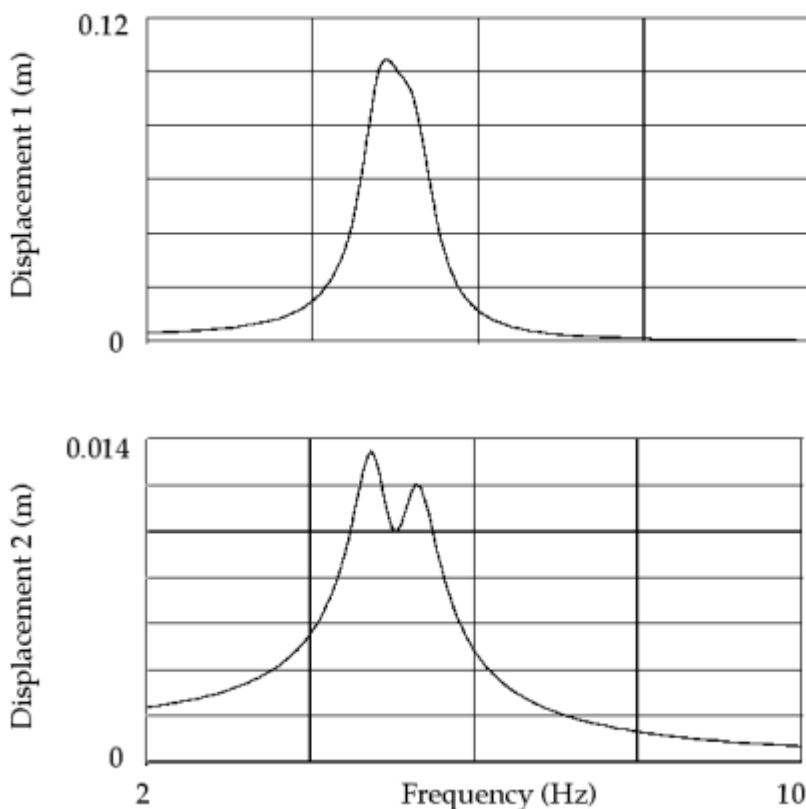
Figure 5-13. Displacement Response Magnitudes With the Auxiliary Structure
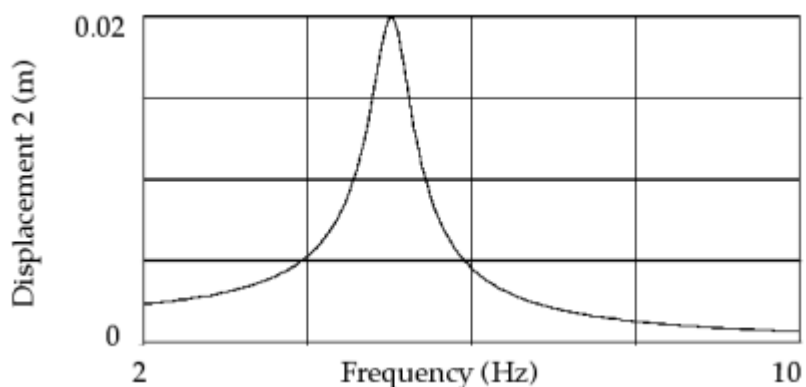
Figure 5-14. Displacement Response Magnitude Without the Auxiliary Structure

Cantilever Beam Model

Consider the cantilever beam shown in Figure 5-15. This model is a planar model of the cantilever beam introduced in "Real Eigenvalue Analysis" with unrestrained DOFs in the T2 and R3 directions. Two loads are applied: one at grid point 6 and the other at grid point 11. The loads have the frequency variation shown in Figure 5-16. The loads in the figure are indicated with a heavy line in order to emphasize their values. The load at grid point 6 has a 45-degree phase lead, and the load at grid point 11 is scaled to be twice that of the load at grid point 6. Modal frequency response is run across a frequency range of 0 to 20 Hz. Modal damping is used with 2% critical damping between 0 and 10 Hz and 5% critical damping above 10 Hz. Modes to 500 Hz are computed using the Lanczos method.
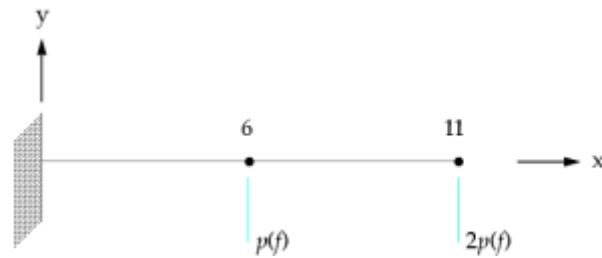
Figure 5-15. Cantilever Beam Model with Applied Loads
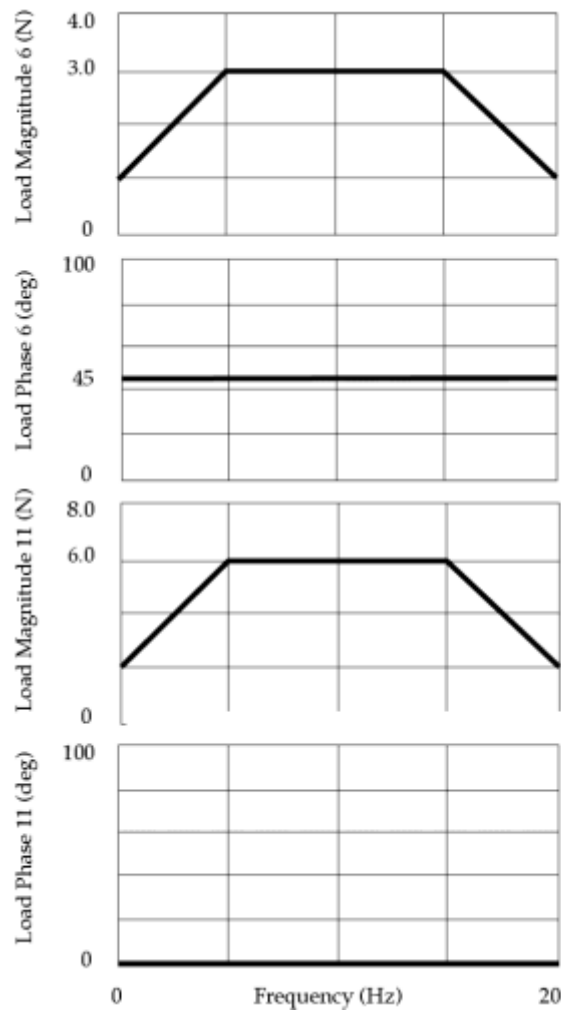
Figure 5-16. Applied Loads

The abridged input file is shown below. The output quantities, as defined in the Case Control Section, are the applied loads (OLOAD) for grid points 6 and 11, physical displacements (DISPLACEMENT) for grid points 6 and 11, solution set displacements (SDISPLACEMENT) for modes 1 and 2, and element forces (ELFORCE) for element 6. These output quantities are plotted rather than printed.

```
$    FILE bd05bar.dat
$
$    CANTILEVER BEAM MODEL
$    CHAPTER 5, FREQUENCY RESPONSE
$
```

```
SOL 111     $  MODAL FREQUENCY RESPONSE
TIME 10
CEND
TITLE =  CANTILEVER BEAM
SUBTITLE =  MODAL FREQUENCY RESPONSE
$
SPC = 21
$
DLOAD = 22
FREQ = 27
SDAMPING = 20
$
METHOD = 10
$
SET 15 = 6,11
OLOAD(PHASE,PLOT) = 15
$
$  PHYSICAL OUTPUT REQUEST
SET 11 = 6,11
DISPLACEMENT(PHASE,PLOT) = 11
$
$  MODAL SOLUTION SET OUTPUT
SET 12 = 1,2
SDISP(PHASE,PLOT) = 12
$
$  ELEMENT FORCE OUTPUT
SET 13 = 6
ELFORCE(PHASE,PLOT) = 13
$
$  XYPLOTS
$
... X-Y plot commands ...
$
BEGIN BULK
$
$.......2.......3.......4.......5.......6.......7.......8.......9.......10.....$
$
EIGRL   10      -0.1    500.            0
$
FREQ1   27      0.0     0.05    400
$
TABDMP1 20      CRIT                                            +TABD1
+TABD1  0.0     0.02    10.0    0.02    10.01   0.05    25.0    0.05    +TABD2
+TABD2  ENDT
$
$ DYNAMIC LOADING
$DLOAD  SID     S       S1      L1      S2      L2
DLOAD   22      1.0     1.0     231     1.0     232
$RLOAD2 SID     DAREA   DELAY   DPHASE  TB      TP
RLOAD2  231     241             261     25
RLOAD2  232     242             25
$DAREA  SID     P1      C1      A1
DAREA   241     6       2       1.0
DAREA   242     11      2       2.0
$DPHASE SID     P1      C1      TH1
DPHASE  261     6       2       45.
$TABLED1 TID                                                    +TABL1
$+TABL1 X1      Y1      X2      Y2      ETC.
TABLED1 25                                                      +TABL1
+TABL1  0.      1.      5.0     3.      15.0    3.0     20.0    1.      +TABL2
+TABL2  25.0    1.      ENDT
$
... basic model ...
$
ENDDATA
```

Figure 5-17. Input File (Abridged) for the Beam Example

Table 5-9 shows the relationship between the Case Control commands and the Bulk Data entries. Note that the DLOAD Bulk Data entry references two RLOAD2 entries, each of which references a separate DAREA entry and a common TABLED1 entry. The RLOAD2 entry for grid point 6 also references a DPHASE entry that defines the 45-degree phase lead.

Table 5-9. Relationship Between Case Control Commands and Bulk Data Entries for the Beam Model

Case Control	Bulk Data
METHOD	EIGRL
FREQUENCY	FREQ1
SDAMPING	TABDMP1

Table 5-9. Relationship Between Case Control Commands and Bulk Data Entries for the Beam Model

Case Control	Bulk Data

DLOAD	DLOAD
	  → RLOAD2 231
	      → DAREA
	      → DPHASE
	      → TABLED1
	  → RLOAD2 232
	      → DAREA

The RLOAD2 entry describes a sinusoidal load in the form

$$\{P(f)\} \;=\; \{AB(f)e^{i\{\phi(f) + \theta - 2\pi f\tau\}}\}$$

Equation 5-29.

where:

A	=	1.0 for grid point 6 and 2.0 for grid point 11 (entered on the DAREA entry)
B	=	function defined on the TABLED1 entry
f	=	0.0 (field 7 of the RLOAD2 entry is blank)
q	=	phase lead of 45 degrees for grid point 6 (entered on the DPHASE entry)
t	=	0.0 (field 4 of the RLOAD2 entry is blank)

Logarithmic plots of the output are shown in the following figures. Figure 5-18 shows the magnitude of the displacements for grid points 6 and 11. Figure 5-19 shows the magnitude of the modal displacements for modes 1 and 2. Figure 5-20 shows the magnitude of the bending moment at end A in plane 1 for element 6. Logarithmic plots are especially useful for displaying frequency response results since there can be several orders of magnitude between the maximum and minimum response values.
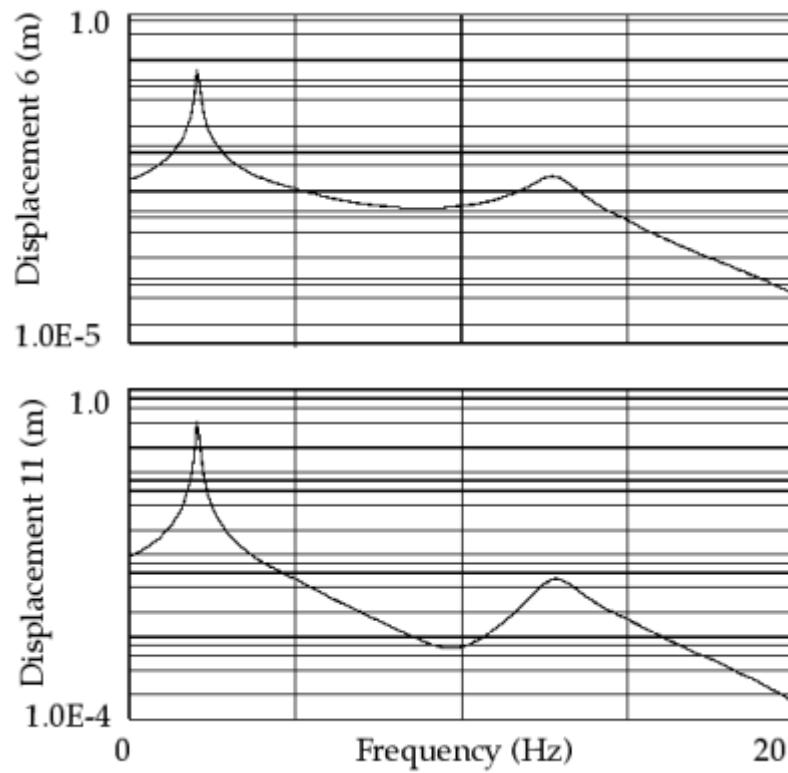
Figure 5-18. Displacement Magnitude (Log)
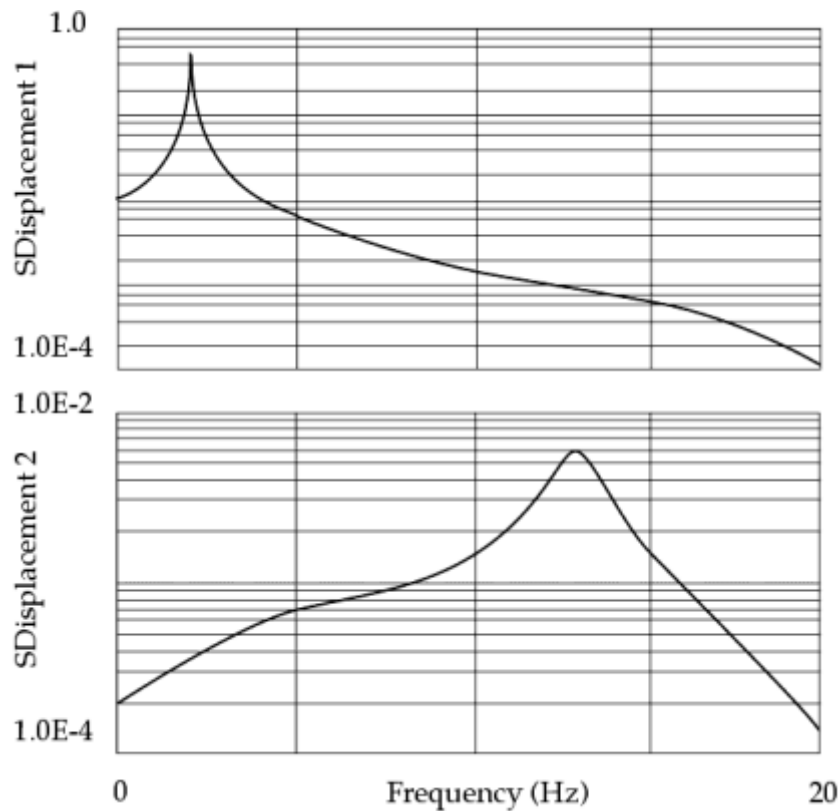
Figure 5-19. Modal Displacement Magnitude (Log)
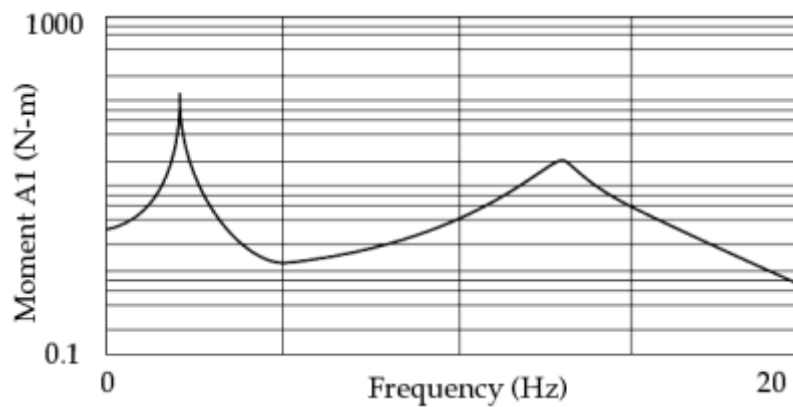
Figure 5-20. Bending Moment Magnitude at End A, Plane 1 (Log)

Bracket Model

Consider the bracket model shown in Figure 5-21. An oscillating pressure load of 3 psi is applied to the elements on the top face in the z-direction. The model is constrained at its base. Modal frequency response is run from 0 to 100 Hz with a frequency step size of 0.2 Hz. Eigenvalues to 1000 Hz are computed using the Lanczos method. Modal damping is applied as 2% critical damping for all modes.

Figure 5-21. Bracket Model

Figure 5-22 shows the abridged input file. The LSEQ entry is used to apply the pressure loads (PLOAD4 entries). Note that the LSEQ and RLOAD1 entries reference a common DAREA ID (999) and that there is no explicit DAREA entry. Table 5-10 shows the relationship between the Case Control commands and the Bulk Data entries.

```
$   FILE  bd05bkt.dat
$
$   BRACKET MODEL
$   CHAPTER 5, FREQUENCY RESPONSE
$
SOL    111     $  MODAL FREQUENCY RESPONSE
TIME   100
CEND
TITLE =  BRACKET MODEL
SUBTITLE = MODAL FREQUENCY RESPONSE ANALYSIS
$
SPC   = 1
$
METHOD = 777
DLOAD = 2
LOADSET = 3
SDAMPING = 4
FREQUENCY = 5
$
$   OUTPUT REQUEST
SET 123 = 999
DISPLACEMENT(PHASE,PLOT)=123
$
$   XYPLOTS
$
... X-Y plot commands ...
$
BEGIN BULK
$
$.......2.......3.......4.......5.......6.......7.......8.......9.......10......
$
$   NORMAL MODES TO 1000 HZ
$EIGRL  SID     V1      V2
EIGRL   777     -0.1    1000.

$
$   EXCITATION FREQUENCY DEFINITION  0 TO 100 HZ
$FREQ1  SID     F1      DF      NDF
FREQ1   5       0.0     0.2     500
$
$   MODAL DAMPING OF 2% CRITICAL FOR ALL MODES
$TABDMP1 TID    TYPE                                            +TABD1
$+TABD1 F1      G1      F2      G2      ETC.
TABDMP1 4       CRIT                                            +TABD1
+TABD1  0.0     0.02    1000.0  0.02    ENDT
$
$   LOAD DEFINITION
$
$RLOAD1 SID     DAREA   DELAY   DPHASE  TC      TD
RLOAD1  2       999                     22
$
$LSEQ   SID     DAREA   LID     TID
LSEQ    3       999     1
$
$TABLED1 TID                                                   +TABL1
$+TABL1 X1      Y1      X2      Y2      ETC.
TABLED1 22                                                     +TABL1
+TABL1  0.0     1.0     1000.0  1.0     ENDT
$
$   PRESURE LOAD OF 3 PSI PER ELEMENT
$PLOAD4 SID     EID     P1
```

```
PLOAD4  1       171     -3.
PLOAD4  1       172     -3.
PLOAD4  1       160     -3.
etc.
$
... basic model ...
$
ENDDATA
```

Figure 5-22. Abridged Input File for the Bracket Model

Table 5-10. Relationship Between Case Control Commands and Bulk Data Entries for the Bracket Model

Case Control	Bulk Data
METHOD	EIGRL
FREQUENCY	FREQ1
SDAMPING	TABDMP1
LOADSET	LSEQ ┌→ PLOAD4 └→ (DAREA ID)
DLOAD	RLOAD1 ┌→ (DAREA ID) └→ TABLED1

Figure 5-23 shows a logarithmic plot of the z-displacement magnitude of grid point 999, which is the concentrated mass at the center of the cutout.
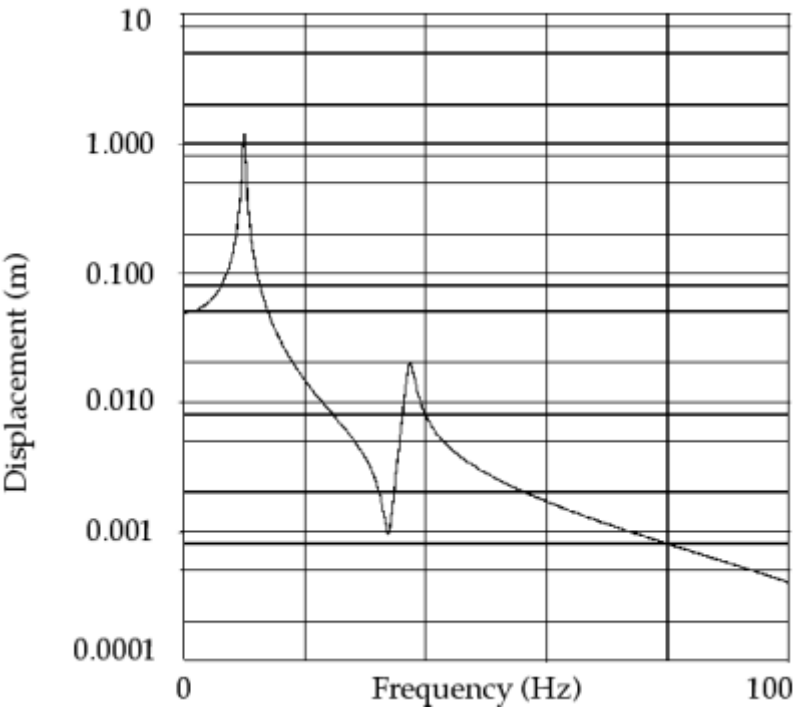
Figure 5-23. Displacement Magnitude (Log)

Chapter

6 *Transient Response Analysis*

- *Overview*

- *Direct Transient Response Analysis*

- *Modal Transient Response Analysis*

- *Modal Versus Direct Transient Response*

- *Transient Excitation Definition*

- *Integration Time Step*

- *Transient Excitation Considerations*

- *Solution Control for Transient Response Analysis*

- *Examples*

6.1   Overview

Transient response analysis is the most general method for computing forced dynamic response. The purpose of a transient response analysis is to compute the behavior of a structure subjected to time-varying excitation. The transient excitation is explicitly defined in the time domain. All of the forces applied to the structure are known at each instant in time. Forces can be in the form of applied forces and/or enforced motions (see "Enforced Motion").

The important results obtained from a transient analysis are typically displacements, velocities, and accelerations of grid points, and forces and stresses in elements.

Depending upon the structure and the nature of the loading, two different numerical methods can be used for a transient response analysis: direct and modal. The direct method performs a numerical integration on the complete coupled equations of motion. The modal method utilizes the mode shapes of the structure to reduce and uncouple the equations of motion (when modal or no damping is used); the solution is then obtained through the summation of the individual modal responses. The choice of the approach is problem dependent. The two methods are described in "Direct Transient Response Analysis" and "Modal Transient Response Analysis".

6.2   Direct Transient Response Analysis

In direct transient response, structural response is computed by solving a set of coupled equations using direct numerical integration. Begin with the dynamic equation of motion in matrix form

$$[M]\{\ddot{u}(t)\} + [B]\{\dot{u}(t)\} + [K]\{u(t)\} = \{P(t)\}$$

Equation 6-1.

The fundamental structural response (displacement) is solved at discrete times, typically with a fixed integration time step Δt.

By using a central finite difference representation for the velocity $\{\dot{u}(t)\}$ and the acceleration $\{\ddot{u}(t)\}$ at discrete times,

$$\{\dot{u}_n\} = \frac{1}{2\Delta t}\{u_{n+1} - u_{n-1}\}$$

$$\{\ddot{u}_n\} = \frac{1}{\Delta t^2}\{u_{n+1} - 2u_n + u_{n-1}\}$$

Equation 6-2.

and averaging the applied force over three adjacent time points, the equation of motion can be rewritten as:

$$\left[\frac{M}{\Delta t^2}\right](u_{n+1} - 2u_n + u_{n-1}) + \left[\frac{B}{2\Delta t}\right](u_{n+1} - u_{n-1})$$

$$+ \left[\frac{K}{3}\right](u_{n+1} + u_n + u_{n-1}) = \frac{1}{3}(P_{n+1} + P_n + P_{n-1})$$

Equation 6-3.

Collecting terms, the equation of motion can be rewritten as:

$$[A_1]\{u_{n+1}\} = [A_2] + [A_3]\{u_n\} + [A_4]\{u_{n-1}\}$$

Equation 6-4.

where:

$[A_1]$ $\quad = \quad$ $\left[\dfrac{M}{\Delta t^2} + \dfrac{B}{2\Delta t} + \dfrac{K}{3}\right]$

$[A_2]$ $\quad = \quad$ $\dfrac{1}{3}\{P_{n+1} + P_n + P_{n-1}\}$

$[A_3]$ $\quad = \quad$ $\left[\dfrac{2M}{\Delta t^2} - \dfrac{K}{3}\right]$

$[A_4]$ $\quad = \quad$ $\left[-\dfrac{M}{\Delta t^2} + \dfrac{B}{2\Delta t} - \dfrac{K}{3}\right]$

Matrix $[A_1]$ is termed the dynamic matrix, and $[A_2]$ is the applied force (averaged over three adjacent time points). This approach is similar to the classical Newmark-Beta direct integration method except that $\{P(t)\}$ is averaged over three time points and $[K]$ is modified such that the dynamic equation of motion reduces to a static solution $[K]\{u_n\} = \{P_n\}$ if no $[M]$ or $[B]$ exists.

The transient solution is obtained by decomposing $[A_1]$ and applying it to the right-hand side of the above equation. In this form, the solution behaves like a succession of static solutions with each time step performing a forward-backward substitution (FBS) on a new load vector. Note that the transient nature of the solution is carried through by modifying the applied force matrix $[A_2]$ with the $[A_3]$ and $[A_4]$ terms.

In its simplest form, the $[M]$, $[B]$, and $[K]$ matrices are assumed to be constant throughout the analysis and do not change with time. Special solution methods are available in NX Nastran for variations in these matrices (see the *NX Nastran Advanced Dynamic Analysis User's Guide*).

A significant benefit presents itself if Δt remains constant during the analysis. With a constant Δt, the $[A_1]$ matrix needs to be decomposed only once. Each progressive step in the analysis is only an FBS of a new load vector. If Δt is changed, $[A_1]$ must be redecomposed, which can be a costly operation in large problems.

Another efficiency in the direct transient solution is that the output time interval may be greater than the solution time interval. In many cases it is not necessary to sample output response at

each solution time. For example, if the solution is performed every 0.001 second the results can be output every fifth time step or every 0.005 second. This efficiency reduces the amount of output.

Damping in Direct Transient Response

The damping matrix [B] is used to represent the energy dissipation characteristics of a structure. In the general case, the damping matrix is comprised of several matrices

$$[B] = [B^1] + [B^2] + \frac{G}{W_3}[K] + \frac{1}{W_4}\sum G_E[K_E]$$

Equation 6-5.

where:

$[B^1]$	=	damping elements (CVISC, CDAMPi) + B2GG
$[B^2]$	=	B2PP direct input matrix + transfer functions
G	=	overall structural damping coefficient (PARAM,G)
W_3	=	frequency of interest in radians per unit time (PARAM,W3) for the conversion of overall structural damping into equivalent viscous damping
$[K]$	=	global stiffness matrix
G_E	=	element structural damping coefficient (GE on the MATi entry)
W_4	=	frequency of interest in radians per unit time (PARAM,W4) for conversion of element structural damping into equivalent viscous damping
$[K_E]$	=	element stiffness matrix

Transient response analysis does not permit the use of complex coefficients. Therefore, structural damping is included by means of equivalent viscous damping. To appreciate the impact of this on the solution, a relation between structural damping and equivalent viscous damping must be defined.

The viscous damping force is a damping force that is a function of a damping coefficient b and the velocity. It is an induced force that is represented in the equation of motion using the [B] matrix and velocity vector.

$$[M]\{\ddot{u}(t)\} + [B]\{\dot{u}(t)\} + [K]\{u(t)\} = \{P(t)\}$$

Equation 6-6.

The structural damping force is a displacement-dependent damping. The structural damping force is a function of a damping coefficient G and a complex component of the structural stiffness matrix.

$$[M]\{\ddot{u}(t)\} + (1 + iG)[K]\{u(t)\} = \{P(t)\}$$

Equation 6-7.

Assuming constant amplitude oscillatory response for an SDOF system, the two damping forces are identical if

$$Gk \; = \; b\omega$$

Equation 6-8.

or

$$b \; = \; \frac{Gk}{\omega}$$

Equation 6-9.

Therefore, if structural damping G is to be modeled using equivalent viscous damping b, then the equality Equation 6-9 holds at only one frequency (see Figure 6-1).

Two parameters are used to convert structural damping to equivalent viscous damping. An overall structural damping coefficient can be applied to the entire system stiffness matrix using PARAM,W3,r where r is the circular frequency at which damping is to be made equivalent. This parameter is used in conjunction with PARAM,G. The default value for W3 is 0.0, which causes the damping related to this source to be ignored in transient analysis.

PARAM,W4 is an alternate parameter used to convert element structural damping to equivalent viscous damping. PARAM,W4,r is used where r is the circular frequency at which damping is to be made equivalent. PARAM,W4 is used in conjunction with the GE field on the MATi entry. The default value for W4 is 0.0 which causes the related damping terms to be ignored in transient analysis.

Units for PARAM,W3 and PARAM,W4 are radians per unit time. The choice of W3 or W4 is typically the dominant frequency at which the damping is active. Often, the first natural frequency is chosen, but isolated individual element damping can occur at different frequencies and can be handled by the appropriate data entries.
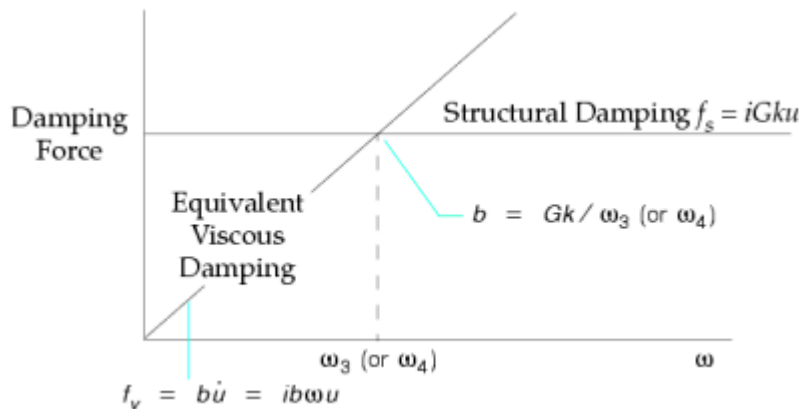
Figure 6-1. Structural Damping Versus Viscous Damping (Constant Oscillatory Displacement)

Initial Conditions in Direct Transient Response

You may impose initial displacements and/or velocities in direct transient response. The TIC Bulk Data entry is used to define initial conditions on the components of grid points. The IC Case Control command is used to select TIC entries from the Bulk Data.

If initial conditions are used, initial conditions should be specified for all DOFs having nonzero values. Initial conditions for any unspecified DOFs are set to zero.

Initial conditions $\{u_0\}$ and $\{\dot{u}_0\}$ are used to determine the values of $\{u_{-1}\}$, $\{P_0\}$, and $\{P_{-1}\}$ used in Equation 6-4 to calculate $\{u_1\}$.

$$\{u_{-1}\} = \{u_0\} - \{\dot{u}_0\}\Delta t$$

Equation 6-10.

$$\{P_{-1}\} = [K]\{u_{-1}\} + [B]\{\dot{u}_0\}$$

Equation 6-11.

In the presence of initial conditions, the applied load specified at $t = 0$ is replaced by

$$\{P_0\} = [K]\{u_0\} + [B]\{\dot{u}_0\}$$

Equation 6-12.

Regardless of the initial conditions specified, the initial acceleration for all points in the structure is assumed to be zero (constant initial velocity).

The format for the TIC entry is

1	2	3	4	5	6	7	8	9	10
TIC	SID	G	C	U0	V0				

Field	Contents
SID	Set ID specified by the IC Case Control command.
G	Grid, scalar, or extra point.
C	Component number.
U0	Initial displacement.
V0	Initial velocity.

Initial conditions may be specified only in direct transient response. In modal transient response all initial conditions are set to zero. Initial conditions may be specified only in the a-set (see "Advanced Dynamic Analysis Capabilities").

6.3   Modal Transient Response Analysis

Modal transient response is an alternate approach to computing the transient response of a structure. This method uses the mode shapes of the structure to reduce the size, uncouple the equations of motion (when modal or no damping is used), and make the numerical integration more efficient. Since the mode shapes are typically computed as part of the characterization of the structure, modal transient response is a natural extension of a normal modes analysis.

As a first step in the formulation, transform the variables from physical coordinates {u} to modal coordinates {x} by

$$\{u(t)\} = [\phi]\{\xi(t)\}$$

Equation 6-13.

The mode shapes [f] are used to transform the problem in terms of the behavior of the modes as opposed to the behavior of the grid points. Equation 6-13 represents an equality if all modes are used; however, because all modes are rarely used, the equation usually represents an approximation.

To proceed, temporarily ignore the damping, resulting in the equation of motion

$$[M]\{\ddot{u}(t)\} + [K]\{u(t)\} = \{P(t)\}$$

Equation 6-14.

If the physical coordinates in terms of the modal coordinates (Equation 6-13 is substituted into Equation 6-14), the following equation is obtained:

$$[M][\phi]\{\ddot{\xi}(t)\} + [K]\{\phi\}\{\xi(t)\} = \{P(t)\}$$

Equation 6-15.

This is now the equation of motion in terms of the modal coordinates. At this point, however, the equations remain coupled.

To uncouple the equations, premultiply by [f ^{T}] to obtain

$$[\phi]^T[M][\phi]\{\ddot{\xi}\} + [\phi]^T[K][\phi]\{\xi\} = [\phi]^T\{P(t)\}$$

Equation 6-16.

where:

$[\phi]^T[M][\phi]$	=	modal (generalized) mass matrix
$[\phi]^T[K][\phi]$	=	modal (generalized) stiffness matrix
$[\phi]^T\{P\}$	=	modal force vector

The final step uses the orthogonality property of the mode shapes to formulate the equation of motion in terms of the generalized mass and stiffness matrices that are diagonal matrices. These matrices do not have off-diagonal terms that couple the equations of motion. Therefore, in this form, the modal equations of motion are uncoupled. In this uncoupled form, the equations of motion are written as a set on uncoupled SDOF systems as

$$m_i \ddot{\xi}_i(t) + k_i \xi_i(t) = p_i(t)$$

Equation 6-17.

where:

m_i	=	i-th modal mass
k_i	=	i-th modal stiffness
p_i	=	i-th modal force

Note that there is no damping in the resulting equation. The next subsection describes how to include damping in modal transient response.

Once the individual modal responses $x_i(t)$ are computed, physical responses are recovered as the summation of the modal responses

$$\{u(t)\} = [\phi]\{\xi(t)\}$$

Equation 6-18.

Since numerical integration is applied to the relatively small number of uncoupled equations, there is not as large a computational penalty for changing Δt as there is in direct transient response analysis. However, a constant Δt is still recommended.

Another efficiency option in the modal transient solution is that the output time interval may be greater than the solution time interval. In many cases, it is not necessary to sample output response at each solution time. For example, if the solution is performed every 0.001 second, the results can be output every fifth time step or every 0.005 second. This efficiency reduces the amount of output.

Damping in Modal Transient Response Analysis

If the damping matrix [B] exists, the orthogonality property (see "Overview of Normal Modes Analysis") of the modes does not, in general, diagonalize the generalized damping matrix

$$[\phi]^T[B][\phi] \neq \text{diagonal}$$

Equation 6-19.

In the presence of a [B] matrix, the modal transient approach solves the coupled problem in terms of modal coordinates using the direct transient numerical integration approach described in Section 4.2 as follows:

$$[A_1]\{\xi_{n+1}\} = [A_2] + [A_3]\{\xi_n\} + [A_4]\{\xi_{n-1}\}$$

Equation 6-20.

where:

$$[A_1] \quad = \quad [\phi^T]\left[\frac{M}{\Delta t^2} + \frac{B}{2\Delta t} + \frac{K}{3}\right][\phi]$$

$$[A_2] \quad = \quad \frac{1}{3}[\phi]^T\{P_{n+1} + P_n + P_{n-1}\}$$

$$[A_3] \quad = \quad [\phi^T]\left[\frac{2M}{\Delta t^2} - \frac{K}{3}\right][\phi]$$

$$[A_4] \quad = \quad [\phi]^T\left[-\frac{M}{\Delta t^2} + \frac{B}{2\Delta t} - \frac{K}{3}\right][\phi]$$

These equations are similar to the direct transient method except that they are in terms of modal coordinates. Since the number of modes used in a solution is typically much less than the number of physical variables, the direct integration of the modal equations is not as costly as with physical variables.

If damping is applied to each mode separately, the decoupled equations of motion can be maintained. When modal damping is used, each mode has damping b_i. The equations of motion remain uncoupled and have the following form for each mode:

$$m_i\ddot{\xi}_i(t) + b_i\dot{\xi}_i(t) + k_i\xi_i(t) \;=\; p_i(t)$$

Equation 6-21.

or

$$\ddot{\xi}_i(t) + 2\zeta_i\omega_i\dot{\xi}_i(t) + \omega_i^2\xi_i(t) \;=\; \frac{1}{m_i}p_i(t)$$

Equation 6-22.

where:

z_i	=	$b_i(2m_iw_i)$ º modal damping ratio
$w^2{}_i$	=	k_i/m_iº modal frequency (eigenvalue)

The TABDMP1 Bulk Data entry defines the modal damping ratios. A table is created by the frequency-damping pairs specified on a TABDMP1 entry. The solution refers to this table for the damping value to be applied at a particular frequency. The TABDMP1 Bulk Data entry has a Set ID. A particular TABDMP1 table is activated by selecting the Set ID with SDAMPING = Set ID Case Control command.

1	2	3	4	5	6	7	8	9	10
TABDMP1	ID	TYPE							
	f1	g1	f2	g2	f3	g3	-etc.-	ENDT	

Field	**Contents**
TID	Table identification number.
	Type of damping units:
TYPE	G (default)
	CRIT
	Q
fi	Frequency value (cycles per unit time).
gi	Damping value in the units specified.

At resonance, the three types of damping are related by the following equations:

$$\zeta_i = \frac{b_i}{b_{cr}} = \frac{G_i}{2}$$

$$b_{cr} = 2m_i\omega_i$$

$$Q_i = \frac{1}{2\zeta_i} = \frac{1}{G_i}$$

Equation 6-23.

The values of fi (units = cycles per unit time) and gi define pairs of frequencies and dampings. Note that gi can be entered as structural damping (default), critical damping, or quality factor. The entered damping is internally converted to structural damping using Equation 6-23. Straight-line interpolation is used for modal frequencies between consecutive fi values. Linear extrapolation is used at the ends of the table. ENDT ends the table input.

For example, if modal damping is entered using Table 6-1 and if modes exist at 1.0, 2.5, 3.6, and 5.5 Hz, NX Nastran interpolates and extrapolates as shown in Figure 6-2 and the table. Note that there is no table entry at 1.0 Hz; NX Nastran uses the first two table entries at $f = 2.0$ and $f = 3.0$ to extrapolate the value for $f = 1.0$.
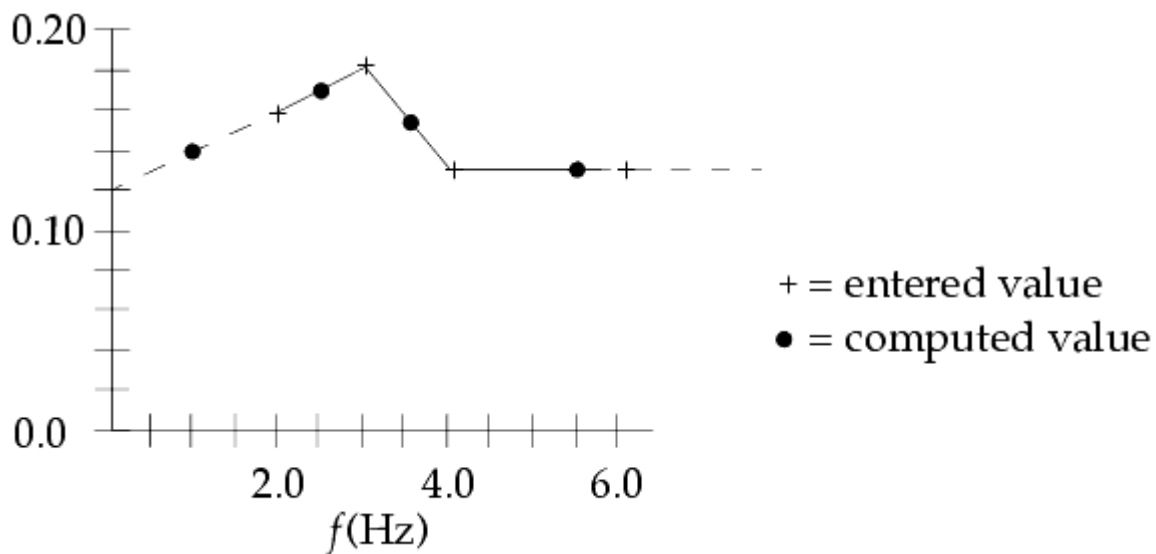
Figure 6-2. Example TABDMP1

Table 6-1. Example TABDMP1 Interpolation/Extrapolation

Entered		Computed	
f	z	f	z
2.0	0.16	1.0	0.14
3.0	0.18	2.5	0.17
4.0	0.13	3.6	0.15
6.0	0.13	5.5	0.13

1	2	3	4	5	6	7	8	9	10
TABDMP1	10	CRIT							+TAB1
+TAB1	2.0	0.16	3.0	0.18	4.0	0.13	6.0	0.13	+TAB2
+TAB2	ENDT								

With the modal equations in the form of Equation 6-22, an efficient uncoupled analytical integration algorithm is used to solve for modal response as decoupled SDOF systems. Each of the modal responses is computed using

$$0, \text{No initial conditions for modal transient response.}$$

$$\xi(t) = e^{-bt/2m}\left(\xi_o \cos\omega_d t + \frac{\xi_o + b/(2m)\xi_o}{\omega_d}\sin\omega_d t\right)$$

$$+ e^{-bt/2m}\frac{1}{m\omega_d}\int_0^t e^{b\tau/2m}p(\tau)\sin\omega_d(t-\tau)d\tau$$

Equation 6-24.

In a modal transient analysis, you may add nonmodal damping (CVISC, CDAMPi, GE on the MATi entry, or PARAM,G). With nonmodal damping, there is a computational penalty due to the coupled [*B*] matrix, causing the coupled solution algorithm to be used. In modal transient response analysis, it is recommended that you use only modal damping (TABDMP1). If discrete damping is desired, direct transient response analysis is recommended.

Note that there are no nonzero initial conditions for modal transient response analysis.

Mode Truncation in Modal Transient Response Analysis

It is possible that not all of the computed modes are required in the transient response solution. Often, only the lowest few suffice for dynamic response calculation. It is quite common to evaluate the frequency content of transient loads and determine a frequency above which no modes are noticeably excited. This frequency is called the cutoff frequency. The act of specifically not using all of the modes of a system in the solution is termed mode truncation. Mode truncation assumes that an accurate solution can be obtained using a reduced set of modes. The number of modes used in a solution is controlled in a modal transient response analysis through a number of methods.

The frequency range selected on the eigenvalue entry (EIGRL or EIGR) is one means to control the frequency range used in the transient response solution. Also, three parameters are available to limit the number of modes included in the solution. PARAM,LFREQ gives the lower limit on the frequency range of retained modes, and PARAM,HFREQ gives the upper limit on the frequency range of retained modes. PARAM,LMODES gives the number of the lowest modes to be retained. These parameters can be used to include the desired set of modes. Note that the default is for all computed modes to be retained.

It is very important to remember that truncating modes in a particular frequency range may truncate a significant portion of the behavior in that frequency range. Typically, high-frequency modes are truncated because they are more costly to compute. So, truncating high-frequency modes truncates high frequency response. In most cases, high-frequency mode truncation is not of concern. You should evaluate the truncation in terms of the loading frequency and the important characteristic frequencies of the structure.

Dynamic Data Recovery in Modal Transient Response Analysis

In modal transient response analysis, two options are available for recovering displacements and stresses: mode displacement method and matrix method. Both methods give the same answers, although with cost differences.

The mode displacement method computes the total physical displacements for each time step from the modal displacements and then computes element stresses from the total physical displacements. The number of operations is proportional to the number of time steps.

The matrix method computes displacements per mode and element stresses per mode, and then computes physical displacements and element stresses as the summation of modal displacements and element stresses. Costly operations are proportional to the number of modes.

Since the number of modes is usually much less that the number of time steps, the matrix method is usually more efficient and is the default. The mode displacement method can be selected by using PARAM,DDRMM,-1 in the Bulk Data. The mode displacement method is required when "time frozen" deformed structure plots are requested (see "Plotted Output").

The mode acceleration method ("Advanced Dynamic Analysis Capabilities") is another form of data recovery for modal transient response analysis. This method can provide better accuracy since detailed local stresses and forces are subject to mode truncation and may not be as accurate as the results computed with the direct method.

6.4   Modal Versus Direct Transient Response

Some general guidelines can be used in selecting modal transient response analysis versus direct transient response analysis. These guidelines are summarized in 6-2.

Table 6-2. Modal Versus Direct Transient Response		
	Modal	**Direct**
Small Model		X
Large Model	X	
Few Time Steps		X
Many Time Steps	X	
High Frequency Excitation		X
Normal Damping		X
Higher Accuracy		X
Initial Conditions		X

In general, larger models may be solved more efficiently in modal transient response because the numerical solution is a solution of a smaller system of uncoupled equations. This result is certainly true if the natural frequencies and mode shape were computed during a previous stage of the analysis. Using Duhamel's integral to solve the uncoupled equations is very efficient even for very long duration transients. On the other hand, the major portion of the effort in a modal transient response analysis is the calculation of the modes. For large systems with a large number of modes, this operation can be as costly as direct integration. This is especially true for high-frequency excitation. To capture high frequency response in a modal solution, less accurate high-frequency modes must be computed. For small models with a few time steps, the direct method may be the most efficient because it solves the equations without first computing the modes. The direct method is more accurate than the modal method because the direct method

is not concerned with mode truncation. For systems with initial conditions, direct transient response is the only choice.

Table 6-2 provides a starting place for evaluating which method to use. Many additional factors may be involved in the choice of a method, such as contractual obligations or local standards of practice.

6.5   Transient Excitation Definition

An important aspect of a transient response analysis is the definition of the loading function. In a transient response analysis, the force must be defined as a function of time. Forces are defined in the same manner whether the direct or modal method is used.

You can use the following Bulk Data entries to define transient loads:

TLOAD1   Tabular input
TLOAD2   Analytical function
DAREA   Spatial distribution of dynamic load
TABLEDi   Tabular values versus time
LSEQ   Generates the spatial distribution of dynamic loads from static load entries
DLOAD   Combines dynamic load sets
DELAY   Time delay

The particular entry chosen for defining the dynamic loading is largely a function of user convenience for concentrated loads. Pressure and distributed loads, however, require a more complicated format.

There are two important aspects of dynamic load definition. First, the location of the loading on the structure must be defined. Since this characteristic locates the loading in space, it is called the spatial distribution of the dynamic loading. Secondly, the time variation in the loading is the characteristic that differentiates a dynamic load from a static load. This time variation is called the temporal distribution of the load. A complete dynamic loading is a product of spatial and temporal distributions.

Using Table IDs and Set IDs in NX Nastran makes it possible to apply many complicated and temporally similar loadings with a minimum of input. Combining simple loadings to create complicated loading distributions that vary in position as well as time is also a straightforward task.

The remainder of this section describes the Bulk Data entries for transient excitation. The description is given in terms of the coefficients that define the dynamic load. See the NX Nastran Quick Reference Guide for more complete Bulk Data descriptions.

Time-Dependent Loads – TLOAD1 Entry

The TLOAD1 entry is the most general form in which to define a time-dependent load. It defines a dynamic loading of the form

$$\{P(t)\} \;=\; \{A \cdot F(t - \tau)\}$$

Equation 6-25.

The coefficients of the force are defined in tabular format. You need not explicitly define a force at every instant in time for which the transient solution is evaluated. Only those values which describe the character of the loading are required. NX Nastran interpolates linearly for intermediate values.

1	2	3	4	5	6	7	8	9	10
TLOAD1	SID	DAREA	DELAY	TYPE	TID				

Field	Contents
SID	Set ID defined by a DLOAD Case Control command or a DLOAD Bulk Data entry.
DAREA	Identification number of DAREA entry that defines A. (Integer > 0)
DELAY	Identification number of DELAY entry that defines t. (Integer > 0)
	Excitation function as defined below. Additional information is in "Enforced Motion" .

TYPE		
	Integer	Excitation Function
	0 or blank	Force or Moment
	1	Enforced Displacement
	2	Enforced Velocity
	3	Enforced Acceleration

	Values 1, 2, and 3 apply only to the large mass method for enforced motion.
TID	TABLEDi entry that defines F(t).

Time-Dependent Loads – TLOAD2 Entry

The TLOAD2 entry is a general analytical form with which to define a time-dependent load. The value of the force at a particular instant in time is determined by evaluating the analytic function at the specific time. You enter the appropriate constants in the function.

The TLOAD2 entry defines dynamic excitation in the form:

$$\{P(t)\} = \begin{cases} 0 & , \ t < (T1 + \tau) \ \text{or} \ t > (T2 + \tau) \\ A\tilde{t}^B e^{C\tilde{t}} \cos(2\pi F\tilde{t} + P) & , \ (T1 + \tau) \leq t \leq (T2 + \tau) \end{cases}$$

Equation 6-26.

where:

$$\tilde{t} = t - (T1 - \tau)$$

1	2	3	4	5	6	7	8	9	10
TLOAD2	SID	DAREA	DELAY	TYPE	T1	T2	F	P	
	C	B							

Field	Contents
SID	Set ID defined by a DLOAD Case Control command.
DAREA	Identification number of DAREA entry that defines A. (Integer > 0)
DELAY	Identification number of DAREA entry that defines t. (Integer > 0)
TYPE	Defined as on the TLOAD1 entry.
T1, T2	Time constants.
F	Frequency (cycles per unit time).
P	Phase angle (degrees).
C	Exponential coefficient.
B	Growth coefficient.

Spatial Distribution of Loading – DAREA Entry

The DAREA entry defines the degrees-of-freedom where the dynamic load is to be applied and a scale factor to be applied to the loading. The DAREA entry provides the basic spatial distribution of the dynamic loading.

1	2	3	4	5	6	7	8	9	10
DAREA	SID	P1	C1	A1	P2	C2	A2		

Field	Contents
SID	Set ID specified by TLOADi entries.
Pi	Grid, extra, or scalar point ID.
Ci	Component number.
Ai	Scale factor.

A DAREA entry is selected by the TLOAD1 or TLOAD2 entry. Any number of DAREA entries may be used; all those with the same SID are combined.

Time Delay – DELAY Entry

The DELAY entry defines the time delay t in an applied load.

1	2	3	4	5	6	7	8	9	10
DELAY	SID	P1	C1	t_1	P2	C2	t_2		

Field	Contents
SID	Set ID specified by TLOADi entry.
Pi	Grid, extra, or scalar point ID.
Ci	Component number.
t_1	Time delay for Pi, Ci.

A DAREA entry must be defined for the same point and component.

Any number of DELAY entries may be used; all those with the same SID are combined.

Dynamic Load Tabular Function – TABLEDi Entries

The TABLEDi entries (i = 1 through 4) each define a tabular function for use in generating frequency-dependent dynamic loads. The form of each TABLEDi entry varies slightly, depending on the value of i, as does the algorithm for y(x). The x values need not be evenly spaced.

The TABLED1, TABLED2, and TABLED3 entries linearly interpolate between the end points and linearly extrapolate outside of the endpoints as shown in Figure 6-3. The TABLED1 entry gives you the option to perform logarithmic interpolation between points, also. The TABLED4 entry uses the endpoint values for values beyond the endpoints.
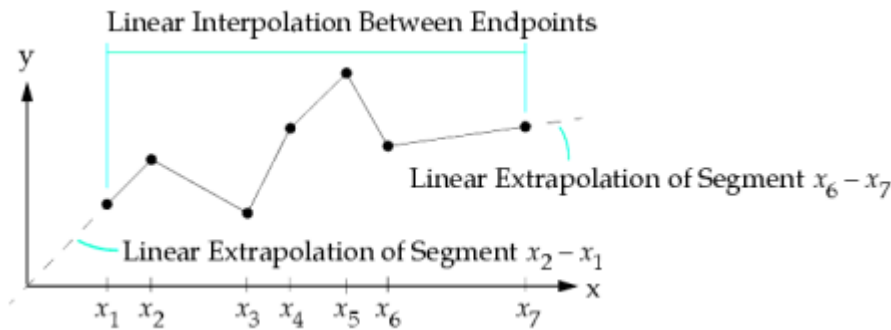
Figure 6-3. Interpolation and Extrapolation for TABLED1, TABLED2, and TABLED3 Entries

The TABLED1 entry has the following format:

1	2	3	4	5	6	7	8	9	10
TABLED1	TID	XAXIS	YAXIS						
	x1	y1	x2	y2	x3	y3	-etc.-	ENDT	

Field	Contents
TID	Table identification number.
XAXIS	Specifies a linear or logarithmic interpolation for the x-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR")
YAXIS	Specifies a linear or logarithmic interpolation for the y-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR")
xi, yi	Tabular values. Values of x are frequency in cycles per unit time.
ENDT	Ends the table input.

The TABLED1 entry uses the algorithm

$$y = y_T(x)$$

Equation 6-27.

The algorithms used for interpolation and extrapolation are as follows:

XAXIS	YAXIS	$y_T(x)$
LINEAR	LINEAR	$\dfrac{xj - x}{xj - xi} yi + \dfrac{x - xi}{xj - xi} yj$

XAXIS	YAXIS	$y_T(x)$
LOG	LINEAR	$\dfrac{\ln(xj/x)}{\ln(xj/xi)}yi + \dfrac{\ln(x/xi)}{\ln(xj/xi)}yj$
LINEAR	LOG	$\exp\left[\dfrac{xj-x}{xj-xi}\ln yi + \dfrac{x-xi}{xj-xi}\ln yj\right]$
LOG	LOG	$\exp\left[\dfrac{\ln(xj/x)}{\ln(xj/xi)}\ln yi + \dfrac{\ln(x/xi)}{\ln(xj/xi)}\ln yj\right]$

The TABLED2 entry has the following format:

1	2	3	4	5	6	7	8	9	10
TABLED2	TID	X1							
	x1	y1	x2	y2	x3	y3	-etc.-	ENDT	

Field	**Contents**
TID	Table identification number.
X1	Table parameter.
xi, yi	Tabular values.
ENDT	Ends the table input.

The TABLED2 entry uses the algorithm

$$y = y_T(x - X1)$$

Equation 6-28.

The TABLED3 entry has the following format:

1	2	3	4	5	6	7	8	9	10
TABLED3	TID	X1	X2						
	x1	y1	x2	y2	x3	y3	-etc.-	ENDT	

Field	**Contents**
TID	Table identification number.
X1, X2	Table parameters.
xi, yi	Tabular values.
ENDT	Ends the table input.

The TABLED3 entry uses the algorithm

$$y = y_T\left(\frac{x - X1}{X2}\right)$$

Equation 6-29.

The TABLED4 entry has the following format:

1	2	3	4	5	6	7	8	9	10
TABLED4	TID	X1	X2	X3	X4				
	A0	A1	A2	A3	A4	A5	-etc.-	ENDT	

Field	**Contents**
TID	Table identification number.
Xi	Table parameters. (X2 ≠ 0.0; X3 < X4)..
Ai	Coefficients.

The TABLED4 entry uses the algorithm

$$y = \sum_{i=0}^{N} A_i\left(\frac{x - X1}{X2}\right)^i$$

Equation 6-30.

N is the degree of the power series. When $x < X3$,*X3* is used for *x*; when $x > X3$, *X4* is used for *x*. This condition has the effect of placing bounds on the table; there is no extrapolation outside of the table boundaries.

ENDT ends the table input.

DAREA Example

Suppose the following command is in the Case Control Section:

DLOAD = 35

in addition to the following entries in the Bulk Data Section:

1	2	3	4	5	6	7	8	9	10
$TLOAD1	SID	DAREA	DELAY	TYPE	TID				
TLOAD1	35	29	31		40				
$DAREA	SID	POINT	COMPONENT	SCALE					
DAREA	29	30	1	4.0					
$DELAY	SID	POINT	COMPONENT	LAG					
DELAY	31	30	1	0.2					
$TABLED1	ID								
$	X1	y1	x2	y2	x3	y3	x4	y4	
TABLED1	40	LINEAR	LINEAR						
	0.0	0.0	0.3	1.0	2.0	1.0	ENDT		

The DLOAD Set ID 35 in the Case Control selects the TLOAD1 entry in the Bulk Data having a Set ID 35. On the TLOAD1 entry is a reference to DAREA Set ID 29, DELAY Set ID 31, and TABLED1 Set ID 40. The DAREA entry with Set ID 29 positions the loading on grid point 30 in the 1 direction with a scale factor of 4.0 applied to the load. The DELAY entry with Set ID 31 delays the loading on grid point 30 in the 1 direction by 0.2 units of time. The TABLED1 entry with Set ID 40 defines the load time history in tabular form. The result of these entries is a dynamic load applied to grid point 30, component T1, scaled by 4.0 and delayed by 0.2 units of time.

Figure 6-4 shows the TABLED1 time history and the applied load (scaled by the DAREA entry and time shifted by the DELAY entry).
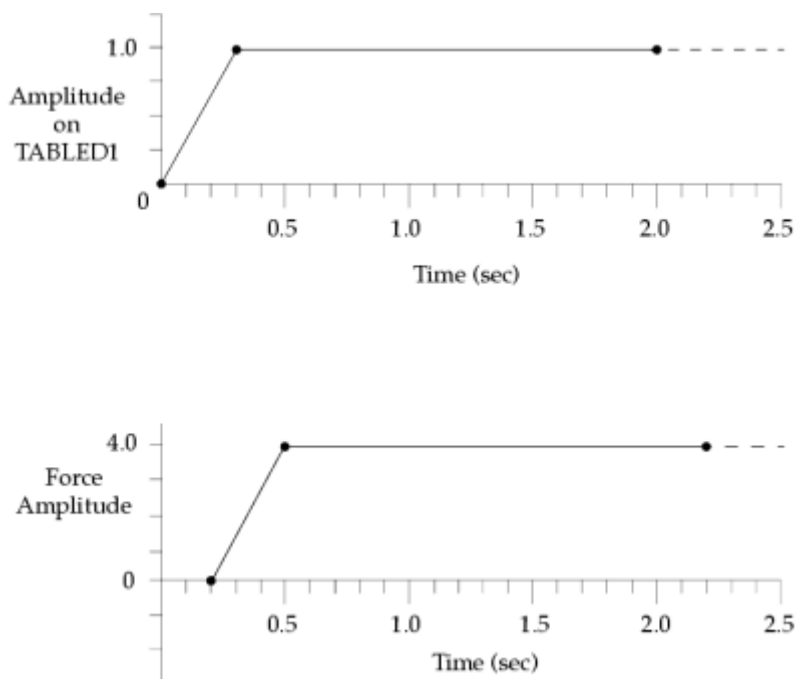
Figure 6-4. Time History from the TABLED1 Entry (Top) and Applied Load (Bottom)

Static Load Sets – LSEQ Entry

NX Nastran does not have specific data entries for many types of dynamic loads. Only concentrated forces and moments can be specified directly using DAREA entries. To accommodate more complicated loadings conveniently, the LSEQ entry is used to define static load entries that define the spatial distribution of dynamic loads.

1	2	3	4	5	6	7	8	9	10
LSEQ	SID	DAREA ID	LOAD ID	TEMP ID					

The LSEQ Bulk Data entry contains a reference to a DAREA Set ID and a static Load Set ID. The static loads are combined with any DAREA entry in the referenced set. The DAREA Set ID does not need to be defined with a DAREA Bulk Data entry. The DAREA Set ID is referenced by a TLOADi entry. This reference defines the temporal distribution of the dynamic loading. The Load Set ID may refer to one or more static load entries (FORCE, PLOADi, GRAV, etc.). All static loads with the Set ID referenced on the LSEQ entry define the spatial distribution of the dynamic loading. NX Nastran converts this information to equivalent dynamic loading.

Figure 6-5 demonstrates the relationships of these entries. To activate a load set defined in this manner, the DLOAD Case Control command refers to the Set ID of the selected DLOAD or TLOADi entry, and the LOADSET Case Control command refers to the Set ID of the selected LSEQ entries. The LSEQ entries point to the static loading entries that are used to define dynamic loadings with DAREA Set ID references. Together this relationship defines a complete dynamic loading. To apply dynamic loadings in this manner, the DLOAD and LOADSET Case Control commands and the TLOADi and LSEQ Bulk Data entries must be defined. A DAREA Bulk Data entry does not need to be defined since the TLOADi and LSEQ entries reference a common DAREA ID. The LSEQ entry can also be interpreted as an internal DAREA entry generator for static load entries.
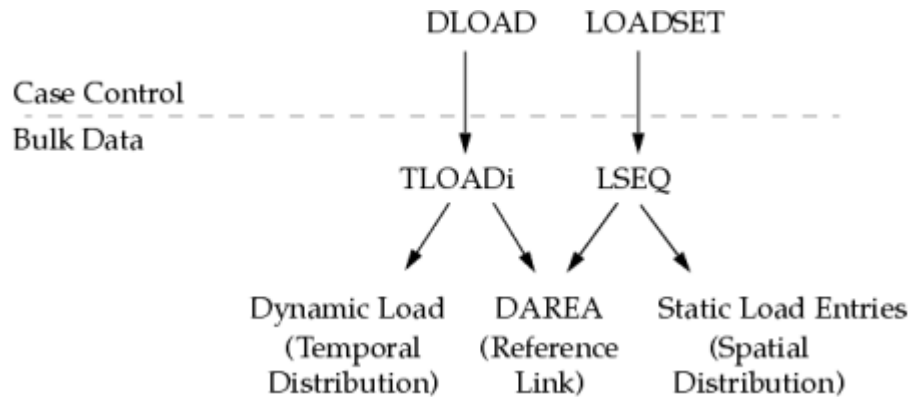
Figure 6-5. Relationship of Dynamic and Static Load Entries

LSEQ Example

Suppose the following commands are in the Case Control Section:

LOADSET = 27

DLOAD = 25

in addition to the following entries in the Bulk Data Section:

1	2	3	4	5	6	7	8	9	10
$LSEQ	SID	DAREA	LID						
LSEQ	27	28	26						

$TLOAD1	SID	DAREA	DELAY	TYPE	TID				
TLOAD1	25	28			29				

$STATIC LOAD	SID								
PLOAD1	26	-etc.-							
FORCE	26	-etc.-							

$TABLED1	TID								
TABLED1	29	-etc.-							

In the above, the LOADSET request in Case Control selects the LSEQ Set ID 27 entry. The DLOAD request in Case Control selects the TLOAD1 Set ID 25 entry. This TLOAD1 entry refers

to a TABLED1 ID 29, which is used to define the temporal variation in the loading. DAREA Set ID 28 links the LSEQ and TLOAD1 entries. In addition, the LSEQ entry refers to static Load Set ID 26, which is defined by FORCE and PLOAD1 entries. The FORCE and PLOAD1 entries define the spatial distribution of the dynamic loading and through the DAREA link refer to the TLOAD1/TABLED1 combination for the time-varying characteristics of the load.

Dynamic Load Set Combination – DLOAD

One of the requirements of transient loads is that all TLOAD1s and TLOAD2s must have unique SIDs. If they are to be applied in the same analysis, they must be combined using the DLOAD Bulk Data entry. The total applied load is constructed from a combination of component load sets as follows:

$$\{P\} \;=\; S\sum_i S_i\{P_i\}$$

Equation 6-31.

where:

S	=	overall scale factor
S_i	=	scale factor for i-th load set
P_i	=	i-th set of loads
P	=	total applied load

The DLOAD Bulk Data entry has the following format:

1	2	3	4	5	6	7	8	9	10
DLOAD	SID	S	S1	L1	S2	L2	-etc.-		

Field	Contents
SID	Load set ID.
S	Overall scale factor.
Si	Individual scale factors.
Li	Load set ID number for TLOAD1 and TLOAD2 entries.

As an example, in the following DLOAD entry:

1	2	3	4	5	6	7	8	9	10
$DLOAD	SID	S	S1	L1	S2	L2	-etc.-		
DLOAD	33	3.25	0.5	14	2.0	27			

a dynamic Load Set ID of 33 is created by taking 0.5 times the loads in Load Set ID of 14, adding to it 2.0 times the loads in Load Set ID of 27, and multiplying that sum by an overall scale factor of 3.25.

As with other transient loads, a dynamic load combination defined by the DLOAD Bulk Data entry is selected by the DLOAD Case Control command.

6.6   Integration Time Step

The TSTEP Bulk Data entry is used to select the integration time step for direct and modal transient response analysis. This entry also controls the duration of the solution and which time steps are to be output. The TSTEP Bulk Data entry is selected by the Set ID referenced on the TSTEP Case Control command.

The integration time step must be small enough to represent accurately the variation in the loading. The integration time step must also be small enough to represent the maximum frequency of interest. The maximum frequency of interest is often called the cutoff frequency. It is recommended to use at least ten solution time steps per period of response for the cutoff frequency. For a given integration time step, integration errors increase with increasing natural frequency because there is an upper limit to the frequency that can be represented by a given time step. Also, integration errors accumulate with total time.

In both direct and modal transient analysis, the cost of integration is directly proportional to the number of time steps. For example, doubling the load duration doubles the integration effort.

In specifying the duration of the analysis on the TSTEP entry, it is important to use an adequate length of time to properly capture long period (low frequency) response. In many cases, the peak dynamic response does not occur at the peak value of load nor necessarily during the duration of the loading function. A good rule is: always solve for at least one cycle of response for the lowest frequency mode after the peak excitation.

You may change Δt during a run but doing so causes the dynamic matrix to be redecomposed, which can be costly in direct transient response analysis.

The TSTEP Bulk Data entry has the following format:

1	2	3	4	5	6	7	8	9	10
TSTEP	SID	N1	Δt_1	NO1					
		N2	Δt_2	NO2					
		-etc.-							

Field	Contents
SID	Set ID specified by a TSTEP Case Control command.
Ni	Number of time steps of value Δt_i .
Δt_i	Integration time step.
NOi	Output every NOi-th time step.

6.7   Transient Excitation Considerations

A number of important considerations must be remembered when applying transient loads. The averaging of applied loads (6-3) in the integration smooths the force and decreases the apparent frequency content. Very sharp spikes in a loading function induce a high-frequency transient response. If the high-frequency transient response is of primary importance in an analysis, a very small integration time step must be used.

It is also important to avoid defining discontinuous forcing functions when describing applied loads. The numerical integration of discontinuous forcing functions may cause different results for the same analysis run on different computers because of slight numerical differences on different computer types. If the analysis calls for loadings with sharp impulses, it is best to smooth the impulse over at least one integration time increment.

The loading function must accurately describe the spatial and temporal distribution of the dynamic load. Simplifying assumptions must not change the character of the load in magnitude, location, or frequency content.

6.8   Solution Control for Transient Response Analysis

The following tables summarize the data entries that can be used to control a transient response analysis. Certain data entries are required, some data entries are optional, while others are user selectable.

In the Executive Control Section of the NX Nastran input file, you must select a solution using the SOL i statement where i is an integer value chosen from 6-3.

Table 6-3. Transient Response Solutions in NX Nastran

	Rigid Formats	**Structured Solution Sequences**
Direct	9	109
Modal	12	112

We have applied these solutions in the examples that follow.

In the Case Control Section of the NX Nastran input file, you must select the solution parameters associated with the current analysis (i.e., time steps, loads, and boundary conditions) and also the output quantities required from the analysis. The Case Control commands directly related to transient response analysis are listed in Table 6-4. They can be combined in the standard fashion with the more generic commands, such as SPC, MPC, etc.

Table 6-4. Transient Response Case Control Commands

Case Control Commands	Direct or Modal	Description	Required/ Optional
DLOAD	Both	Select the dynamic load set from the Bulk Data	Required[1]
TSTEP	Both	Select the TSTEP entry from the Bulk Data	Required
METHOD	Modal	Select the eigenvalue extraction parameters	Required
LOADSET	Both	Select the LSEQ set from the Bulk Data	Optional
SDAMPING	Modal	Select the modal damping table from the Bulk Data	Optional
IC	Direct	Select TIC entries for initial conditions from the Bulk Data	Optional
OTIME	Both	Select the times for output (default = all)	Optional

[1]Not required when using initial conditions.

The types of results available from a transient response analysis are similar to those for a static analysis except that the results vary with time. Additional quantities are also available, which is characteristic of dynamics problems. The output quantities are summarized in Table 6-5 and Table 6-6.

Table 6-5. Grid Point Output from a Transient Response Analysis

Case Control Command	Description
ACCELERATION	Grid point acceleration time history for a set of grid points
DISPLACEMENT (or VECTOR)	Grid point displacement time history for a set of grid points
GPSTRESS	Grid point stress time history (requires SURFACE/VOLUME definition in the OUTPUT(POST) section of the Case Control)
OLOAD	Requests applied load table to be output for a set of grid points
SACCELERATION	Requests solution set acceleration output: d-set in direct solutions and modal variables in modal solutions
SDISPLACEMENT	Requests solution set displacement output: d-set in direct solutions and modal variables in modal solutions
SVECTOR	Requests real eigenvector output for the a-set in modal solutions
SVELOCITY	Requests solution set velocity output: d-set in direct solutions and modal variables in modal solutions
SPCFORCES	Requests forces of single-point constraint for a set of grid points
VELOCITY	Grid point velocity time history for a set of grid points

Table 6-6. Element Output from a Transient Analysis

Case Control Command	Description
ELSTRESS (or STRESS)	Element stress time history for a set of elements
ELFORCE (or FORCE)	Element force time history for a set of elements
STRAIN	Element strain time history for a set of elements

Because the results may be output for many time steps, the volume of output can be very large. Prudent selection of the output quantities is recommended.

A number of Bulk Data entries are unique to transient response analysis. They can be combined with other generic entries in the Bulk Data. Bulk Data entries directly related to transient analysis are summarized in Table 6-7.

Table 6-7. Bulk Data Entries for Transient Response Analysis

Bulk Data Entry	Direct or Modal	Description	Required/ Optional
TSTEP	Both	Integration time step and solution control	Required
TLOADi	Both	Dynamic loading	Required[1]
EIGR or EIGRL	Modal	Eigenvalue analysis parameters	Required

Table 6-7. Bulk Data Entries for Transient Response Analysis

Bulk Data Entry	Direct or Modal	Description	Required/ Optional
LSEQ	Both	Dynamic loading from static loads	Optional
TABLEDi	Both	Time-dependent tables for TLOADi	Optional[1]
TIC	Direct	Initial conditions on grid, scalar, and extra points	Optional
DAREA	Both	Load component and scale factor	Required[2]
DELAY	Both	Time delay on dynamic load	Optional
DLOAD	Both	Dynamic load combination, required if TLOAD1 and TLOAD2 are used	Optional
TABDMP1	Modal	Modal damping table	Optional

[1]Not required for initial conditions.

[2]The DAREA ID is required; the DAREA Bulk Data entry is not required if an LSEQ entry is used.

6.9 Examples

This section provides several examples showing the input and output. These examples are

Model	Transient Response Bulk Data Entries	Output
bd06two	TSTEP, TIC	X-Y plots
bd06bar	EIGRL, TSTEP, TABDMP1, DLOAD, TLOAD2, DAREA, DELAY	X-Y plots
bd06bkt	EIGRL, TSTEP, TABDMP1, TLOAD1, LSEQ, TABLED1, PLOAD4	X-Y plot

These examples are described in the sections that follow.

Two-DOF Model

Consider the two-DOF system shown in Figure 6-6. Direct transient response (SOL 109) is run with an initial displacement of 0.1 meter at grid point 2. The analysis is run for a duration of 10 seconds with a Δt of 0.01 second. Damping is neglected in the analysis. Part of the input file is shown below.
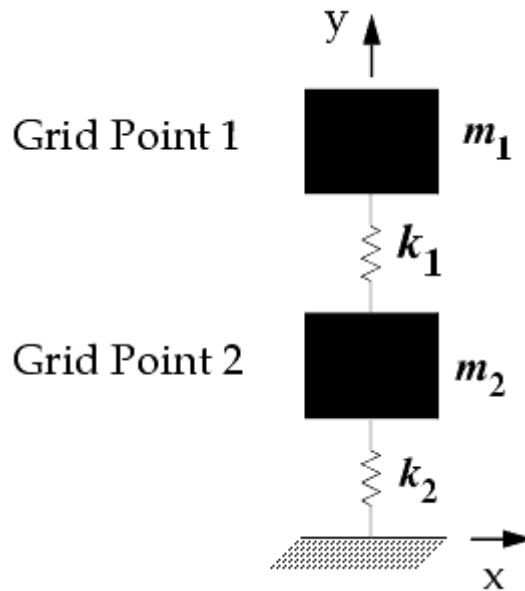
Figure 6-6. Two-DOF Model

```
$   FILE bd06two.dat
$
$   TWO-DOF SYSTEM
$   CHAPTER 6, TRANSIENT RESPONSE
$
TIME 5
SOL 109  $ DIRECT TRANSIENT RESPONSE
CEND
TITLE = TWO-DOF SYSTEM
SUBTITLE = DIRECT FREQUENCY RESPONSE
LABEL =  INITIAL DISPL. AT GRID 2
$
$   SPECIFY SPC
SPC = 996
$
$   SPECIFY DYNAMIC INPUT
TSTEP = 888
IC = 777
$
$   SELECT OUTPUT
SET 11 = 1,2
DISPLACEMENT(PLOT) = 11
$
$   XYPLOTS
$
... X-Y plot commands ...
$
BEGIN BULK
$
$.......2.......3.......4.......5.......6.......7.......8.......9.......10.....$
$
$   ENTRIES FOR TRANSIENT RESPONSE
$
$   INITIAL CONDITION
$TIC     SID     G       C       U0      V0
TIC     777     2       2       0.1
$
$   TIME STEP
$TSTEP   SID     N1      DT1     NO1
TSTEP   888     1000    0.01    1
$
... basic model ...
$
ENDDATA
```

Figure 6-7. Input File (Abridged) for the Two-DOF Example

Table 6-8 shows the relationship between the Case Control commands and the Bulk Data entries. This example represents the simplest form of dynamic response input. The only required entries are those that define the time step and the initial conditions. Note that the unspecified initial

conditions are assumed to be zero. Note, too, that the initial conditions are available only for direct transient response analysis.

Table 6-8. Relationship Between Case Control Commands and Bulk Data Entries for the Two-DOF Model	
Case Control	**Bulk Data**
TSTEP	TSTEP
IC	TIC

Figure 6-8 shows the plots of the resulting displacements for grid points 1 and 2. Note that there are two frequencies of response: a higher frequency of about 5 Hz, and a lower frequency of about 0.25 Hz. The energy (and hence response) appears to be transferred repetitively between grid points 1 and 2 as represented by the lower frequency response. This energy transfer is called beating. Beating occurs when there are closely-spaced modes (in this case, 4.79 Hz and 5.29 Hz) in which energy transfer can readily occur. The response is comprised of two frequencies as given below:

$$f_{higher} = \frac{f1 + f2}{2}$$

$$f_{lower} = \frac{f2 - f1}{2}$$

Equation 6-32.

where:

f1	=	lower of the closely-spaced mode frequencies
f2	=	higher of the closely-spaced mode frequencies

In this example, f_{higher} is 5.04 Hz and f_{lower} is 0.25 Hz. The lower frequency is called the beat frequency and is the frequency at which energy transfer occurs.
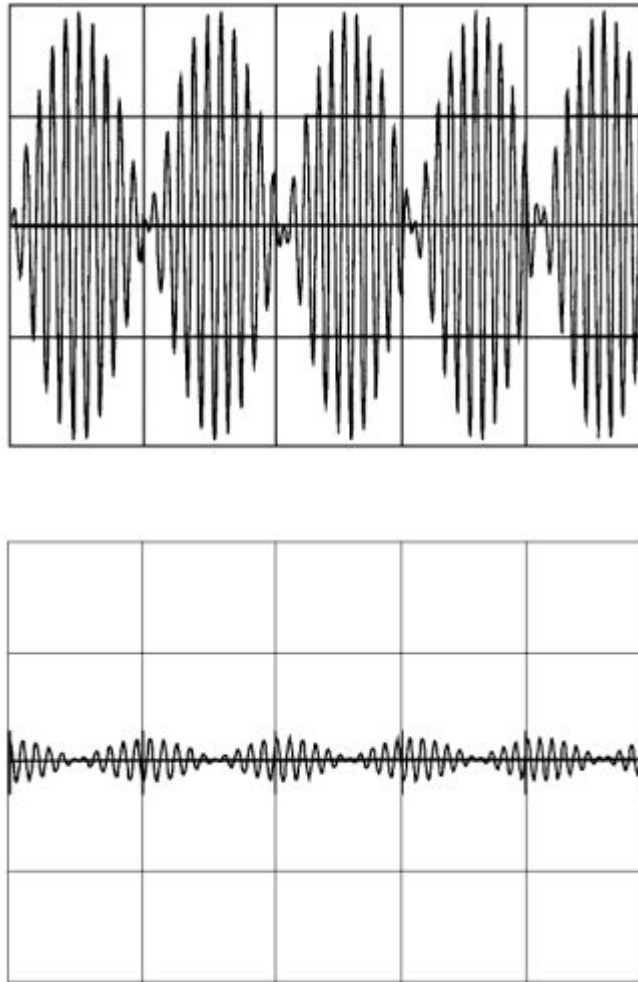
Figure 6-8. Displacements of Grid Points 1 and 2

Cantilever Beam Model

Consider the cantilever beam shown below. This beam model is the same as in "Examples" in Chapter 5. Modal transient response (SOL 112) is run with loads applied to grid points 6 and 11 as shown in Figure 6-10. The analysis is run for a duration of 2 seconds with a Δt of 0.001 second. Modal damping of 5% critical damping is used for all modes. Modes up to 3000 Hz are computed using the Lanczos method. Figure 6-11 shows part of the input file.
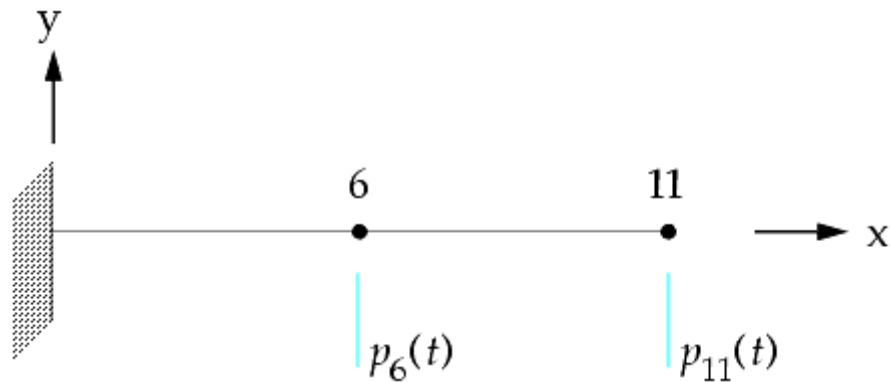
Figure 6-9. Cantilever Beam Model with Applied Loads
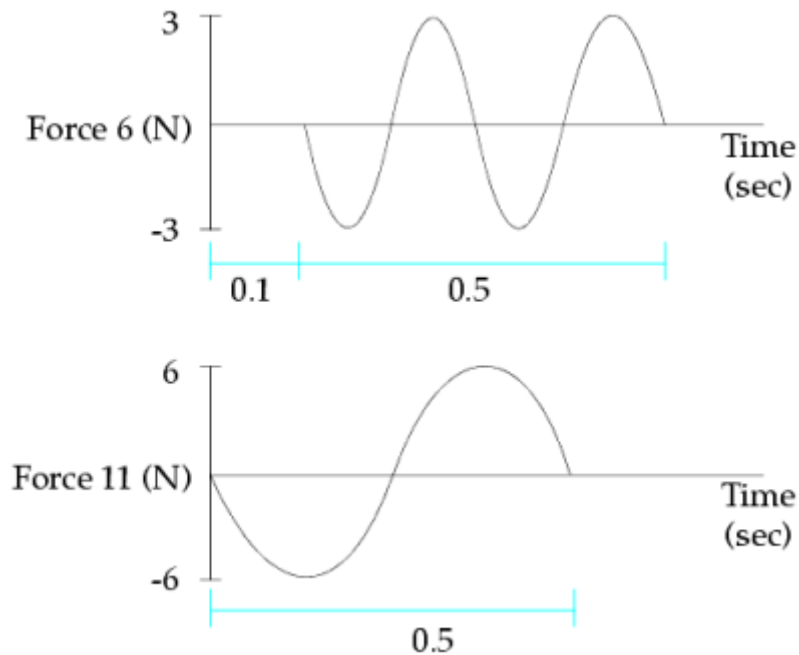
Figure 6-10. Applied Loads for the Beam Model

```
$   FILE bd06bar.dat
$
$   CANTILEVER BEAM MODEL
$   CHAPTER 6, TRANSIENT RESPONSE
$
SOL 112      $  MODAL TRANSIENT RESPONSE
TIME 10
CEND
TITLE =  CANTILEVER BEAM
SUBTITLE =  MODAL TRANSIENT RESPONSE
$
SPC = 21
DLOAD = 22
TSTEP = 27
SDAMPING = 25
$
METHOD = 10
$
$  PHYSICAL OUTPUT REQUEST
SET 11 = 6,11
DISPLACEMENT(PLOT) = 11
ACCELERATION(PLOT) = 11
$
$  MODAL SOLUTION SET OUTPUT
SET 12 = 1,2
SDISP(PLOT)  = 12
```

```
$
$  ELEMENT FORCE OUTPUT
SET 13 = 6
ELFORCE(PLOT) = 13
$
$  APPLIED LOAD OUTPUT
SET 15 = 6,11
OLOAD(PLOT) = 15
$
$  XYPLOTS
$
... X-Y plot commands ...
$
BEGIN BULK
$
$.......2.......3.......4.......5.......6.......7.......8.......9.......10.....$
$
$EIGRL   SID    V1      V2              MSGLVL
EIGRL   10     -0.1    3000.           0
$
$TSTEP   SID    N1      DT1     NO1
TSTEP   27     2000    0.001   1
$
$  MODAL DAMPING OF 5% IN ALL MODES
$TABDMP1 TID    TYPE                                            +TABD
$+TABD   F1     G1      F2      G2      ETC.
TABDMP1 25     CRIT                                             +TABD
+TABD   0.     0.05    1000.   0.05    ENDT
$
$ DYNAMIC LOADING
$DLOAD   SID    S       S1      L1      S2      L2
DLOAD   22     1.0     1.0     231     1.0     232
$TLOAD2  SID    DAREA   DELAY   TYPE    T1      T2      F       P       +TL1
$+TL1    C      B
TLOAD2  231    241             0       0.0     0.5     2.0     90.
TLOAD2  232    242     262     0       0.0     0.5     4.0     90.
$DAREA   SID    P1      C1      A1
DAREA   241    11      2       6.0
DAREA   242    6       2       3.0
$DELAY   SID    P1      C1      T1
DELAY   262    6       2       0.1
$
... basic model ...
$
ENDDATA
```

Figure 6-11. Input File (Abridged) for the Beam Example

Table 6-9 shows the relationship between the Case Control commands and the Bulk Data entries. The DLOAD Bulk Data entry references two TLOAD2 entries, each of which references separate DAREA entries. A TLOAD2 entry also references a DELAY entry to apply the time delay to the load at grid point 6.

Table 6-9. Relationship Between Case Control Commands and Bulk Data Entries for the Bar Model

Case Control	Bulk Data
METHOD	EIGRL
TSTEP	TSTEP
SDAMPING	TABDMP1

Table 6-9. Relationship Between Case Control Commands and Bulk Data Entries for the Bar Model

Case Control	Bulk Data

DLOAD	DLOAD
	
	→ TLOAD2 231
	
	    → DAREA
	
	→ TLOAD2 232
	
	    → DAREA
	    → DELAY

Plotted output is shown in the following figures. Figure 6-12 shows the applied loads at grid points 6 and 11. Figure 6-13 shows the plots of the displacements for grid points 6 and 11. Figure 6-14 shows the accelerations for grid points 6 and 11. Figure 6-15 shows the bending moment at end A in plane 1 for element 6. Figure 6-16 shows the modal displacements for modes 1 and 2.
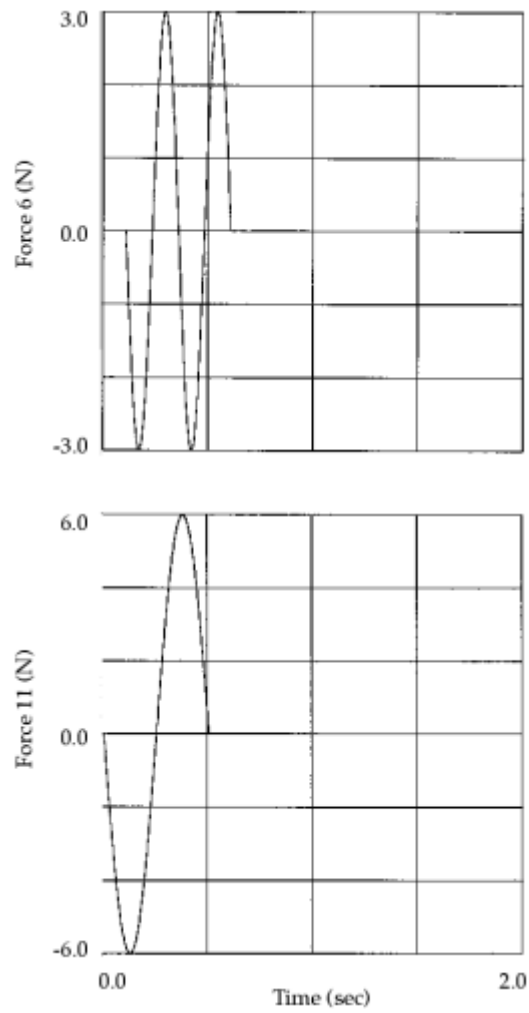
Figure 6-12. Applied Loads at Grid Points 6 and 11
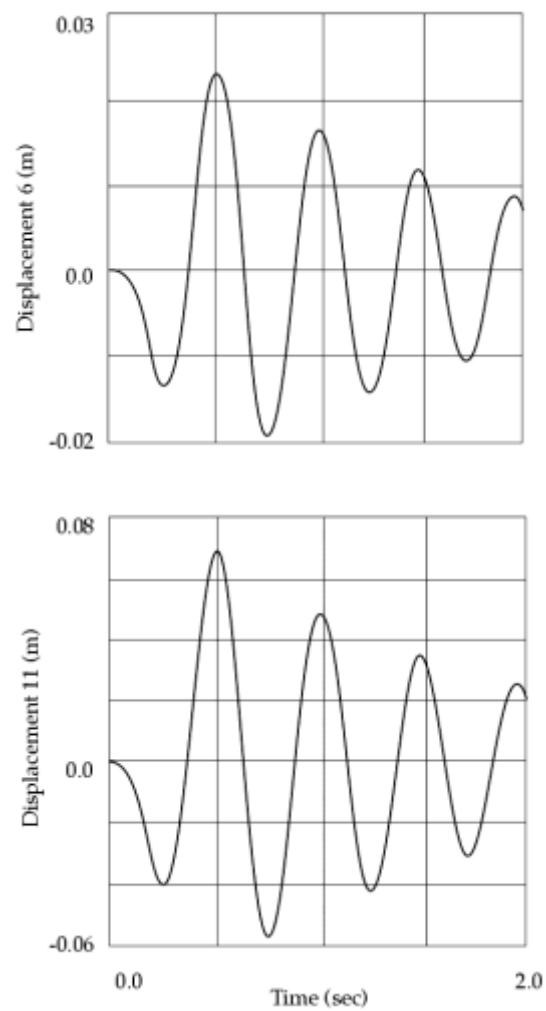
Figure 6-13. Displacements at Grid Points 6 and 11
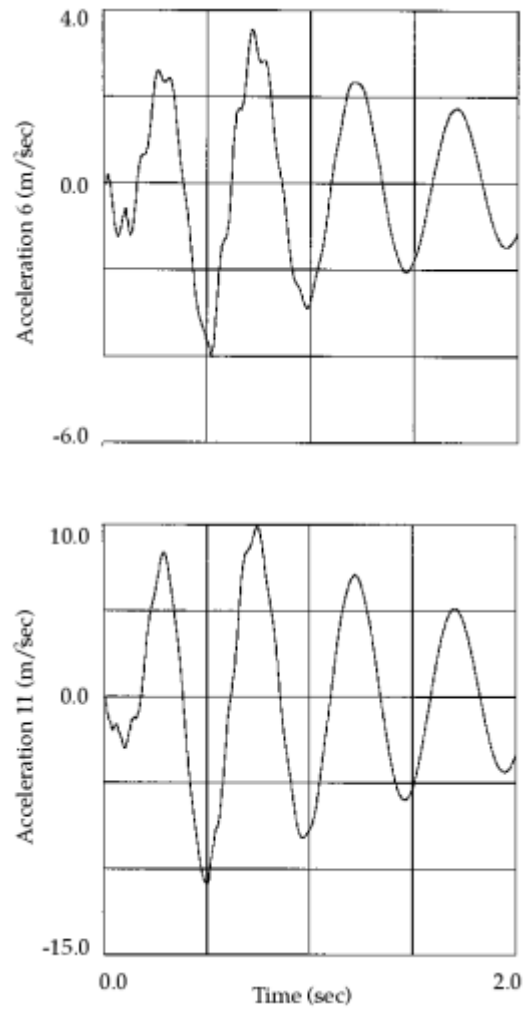
Figure 6-14. Accelerations at Grid Points 6 and 11
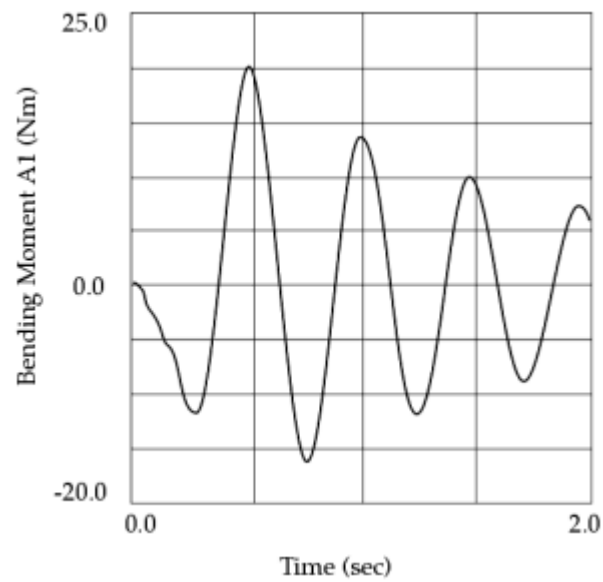
Figure 6-15. Bending Moment A1 for Element 6
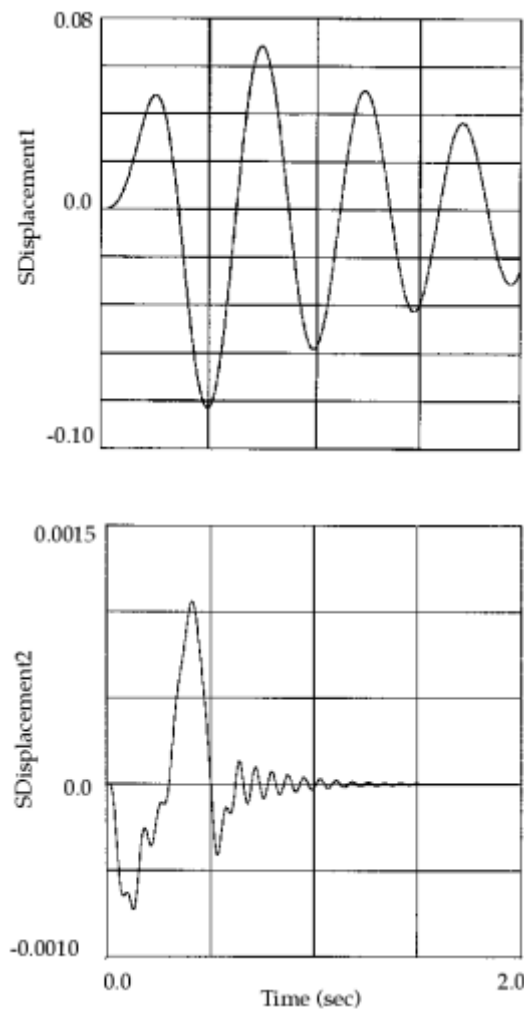
Figure 6-16. Modal Displacements for Modes 1 and 2

Bracket Model

Consider the bracket model shown in Figure 6-17. A pressure load of 3 psi is applied to the elements in the top face in the z-direction with the time history shown in Figure 6-18. The modal transient analysis is run for 4 seconds with a time step size of 0.005 second. Modal damping of 2% critical damping is used for all modes. Modes up to 3000 Hz are computed with the Lanczos method. The model is constrained near the base.
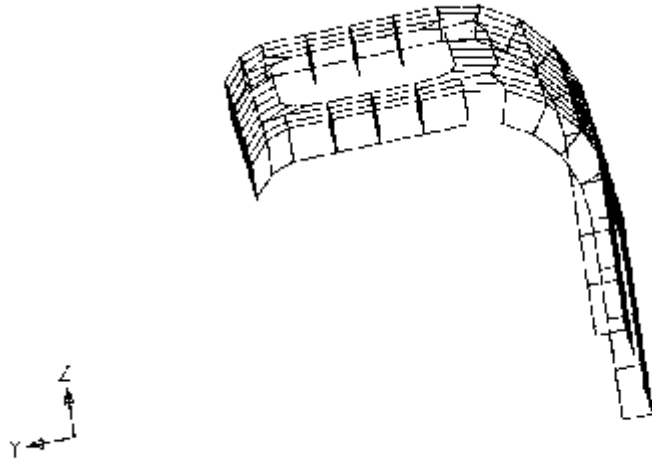
Figure 6-17. Bracket Model
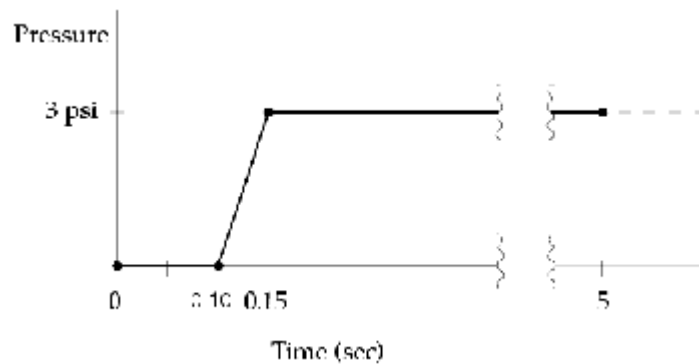
Figure 6-18. Time Variation for Applied Load

Figure 6-19 shows the abridged input file. The LSEQ entry is used to apply the pressure loads (PLOAD4 entries). Note that the LSEQ and TLOAD1 entries reference a common DAREA ID (999) and that there is no explicit DAREA entry. Table 6-10 shows the relationship between the Case Control commands and the Bulk Data entries.

```
$   FILE  bd06bkt.dat
$
$   BRACKET MODEL
$   CHAPTER 6, TRANSIENT RESPONSE
$
SOL    112   $ MODAL TRANSIENT RESPONSE
TIME   100
CEND
TITLE =  BRACKET MODEL
SUBTITLE = MODAL TRANSIENT RESPONSE ANALYSIS
$
SPC   = 1
$
METHOD = 777
$
DLOAD = 2
LOADSET = 3
SDAMPING = 4
TSTEP = 5
$
$  OUTPUT REQUEST
SET 123 = 999
DISPLACEMENT(PLOT)=123
```

```
$
$  XYPLOTS
$
... X-Y plot commands ...
$
BEGIN BULK
$
$......2.......3.......4.......5.......6.......7.......8.......9.......10......
$
$  NORMAL MODES TO 3000 HZ
$EIGRL  SID     V1      V2
EIGRL   777     -0.1    3000.
$
$  4 SECONDS OF RESPONSE
$TSTEP  SID     N1      DT1     NO1
TSTEP   5       800     0.005   1
$
$  MODAL DAMPING OF 2% CRITICAL
$TABDMP1 TID    TYPE                                            +TABD1
$+TABD1 F1      G1      F2      G2      ETC.
TABDMP1 4       CRIT                                            +TABD1
+TABD1  0.0     0.02    3000.0  0.02    ENDT
$
$  LOAD DEFINITION
$
$TLOAD1 SID     DAREA   DELAY   TYPE    TID
TLOAD1  2       999                     22
$
$LSEQ   SID     DAREA   LID     TID
LSEQ    3       999     1
$
$ TIME HISTORY
$TABLED1 TID                                                   +TABL1
$+TABL1 X1      Y1      X2      Y2      ETC.
TABLED1 22                                                      +TABL1
+TABL1  0.0     0.0     0.1     0.0     0.15    1.0     5.0     1.0     +TABL2
+TABL2  ENDT
$
$  PRESSURE LOAD OF 3 PSI PER ELEMENT
$PLOAD4 SID     EID     P1
PLOAD4  1       171     -3.
PLOAD4  1       172     -3.
PLOAD4  1       160     -3.
etc.
$
... basic model ...
$
ENDDATA
```

Figure 6-19. Input File (Abridged) for the Bracket Model

Table 6-10. Relationship Between Case Control Commands and Bulk Data Entries for the Bracket Model

Case Control		Bulk Data
METHOD	EIGRL	
FREQUENCY	FREQ1	
SDAMPING	TABDMP1	

Figure 6-20 shows a plot of the z-displacement of grid point 999, which is the concentrated mass at the center of the cutout.
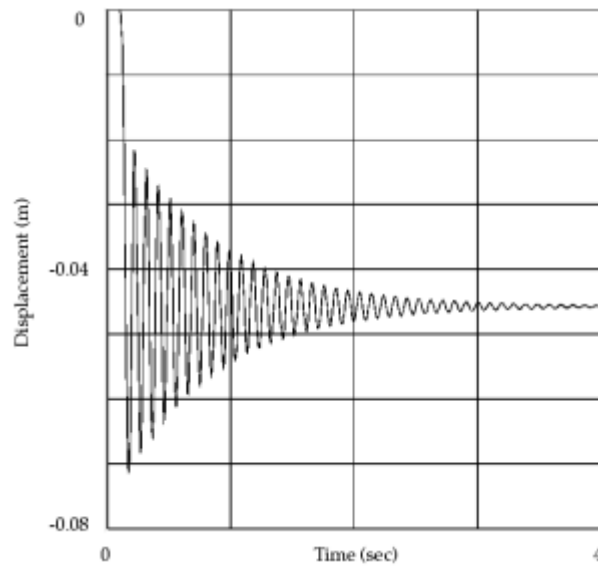
Figure 6-20. Displacement Time History for Grid Point 999

Chapter

7   *Enforced Motion*

- *Overview*

- *The Large Mass Method in Direct Transient and Direct Frequency Response*

- *The Large Mass Method in Modal Transient and Modal Frequency Response*

- *User Interface for the Large Mass Method*

- *Examples*

7.1   Overview

Enforced motion specifies the displacement, velocity, and/or acceleration at a set of grid points for frequency and transient response. Enforced motion is used when base motion is specified instead of or in conjunction with applied loads. A common application is an earthquake excitation applied to a building. In this case there are no applied loads, instead the base of the building undergoes an enforced displacement or acceleration time history.

NX Nastran does not include a completely automatic method for prescribing enforced motion in dynamics. Instead, the procedures described in 5 and "Transient Response Analysis" for specifying applied forces are used in conjunction with techniques that convert applied forces into enforced motion. One such method that is applicable to both transient response and frequency response is described in this chapter—the large mass method. Another method for enforced motion, described in the *NX Nastran Advanced Dynamic Analysis User's Guide*, is the Lagrange multiplier technique.

7.2   The Large Mass Method in Direct Transient and Direct Frequency Response

If a very large mass m_o , which is several orders of magnitude larger than the mass of the entire structure, is connected to a degree-of-freedom and a dynamic load p is applied to the same degree-of-freedom, then the acceleration of the degree-of-freedom, to a close approximation, is as follows:

$$\ddot{u} \;=\; \frac{1}{m_o}p$$

Equation 7-1.

In other words, the load that produces a desired acceleration $\ddot{u}$ is approximately

$$p \;=\; m_o \ddot{u}$$

Equation 7-2.

The accuracy of this approximation increases as m_o is made larger in comparison to the mass of the structure. The only limit for the size of m_o is numeric overflow in the computer. Generally, the value of m_o should be approximately 10^6 times the mass of the entire structure for an enforced translational degree-of-freedom and 10^6 times the mass moment of inertia of the entire structure for a rotational DOF. The factor 10^6 is a safe limit that should produce approximately six digits of numerical accuracy.

The large mass method is implemented in direct transient and frequency response analysis by placing large masses m_o on all enforced degrees-of-freedom and supplying applied dynamic loads specified by Equation 7-2; that is, the function $\ddot{u}$ is input on entries normally used for the input of loads, and the scale factor m_o can be input on DAREA or DLOAD Bulk Data entries, whichever is more convenient. CMASSi or CONMi entries should be used to input the large masses.

7-2 is not directly helpful if enforced displacement or enforced velocity is specified rather than enforced acceleration. However, 7-2 can be made serviceable in frequency response analysis by noting that

$$\ddot{u} \;=\; i\omega\dot{u} \;=\; -\omega^2 u$$

Equation 7-3.

so that

$$p \;=\; m_o\ddot{u} \;=\; i\omega m_o\dot{u} \;=\; -\omega^2 m_o u$$

Equation 7-4.

The added factor (iw or —w^2) can be carried by the function tabulated on the TABLEDi entry used to specify the frequency dependence of the dynamic load.

In the case of transient analysis, provision is made on the TLOAD1 and TLOAD2 entries for you to indicate whether an enforced displacement, velocity, or acceleration is supplied (TYPE = 1, 2, or 3). NX Nastran then automatically differentiates a specified velocity once or a specified displacement twice to obtain an acceleration. The remaining required user actions are the same as for enforced acceleration.

In summary, the user actions for direct frequency and direct transient response are

- Remove any constraints from the enforced degrees-of-freedom.

- Apply large masses m_o with CMASSi or CONMi Bulk Data entries to the DOFs where the motion is enforced. The magnitude of m_o should be approximately 10^6 times the entire mass of the structure (or approximately 10^6 times the entire mass moment of inertia of the structure if the component of enforced motion is a rotation).

- In the case of direct frequency response, apply a dynamic load computed according to Equation 7-4 to each enforced degree-of-freedom.

- In the case of direct transient response,

  – Indicate in field 5 of the TLOAD1 and TLOAD2 entries whether the enforced motion is a displacement, velocity, or acceleration.

  – Apply a dynamic load to each enforced degree-of-freedom equal to m_ou , $m_o\dot{u}$, or $m_o\ddot{u}$, depending on whether the enforced motion is a displacement, velocity, or acceleration.

Be careful when using PARAM,WTMASS. The WTMASS parameter multiplies the large mass value, which changes the effective enforced acceleration to

$$\ddot{u} \;=\; \frac{p}{m_o \cdot WTMASS}$$

Equation 7-5.

Enforced velocity and enforced displacement are changed likewise.

You may well ask whether a stiff spring may be used instead of a large mass. In that case the applied load is

$$p = k_o u$$

Equation 7-6.

where k_o is the stiffness of the stiff spring and u is the enforced displacement. The large stiffness method certainly works, but the large mass method is preferred because it is easier to estimate a good value for the large mass than to estimate a good value for the stiff spring. In addition and more importantly, the large mass method is far superior when modal methods are used. If very stiff springs are used for modal analysis rather than very large masses, the vibration modes corresponding to the very stiff springs have very high frequencies and in all likelihood, are not included among the modes used in the response analysis. This is the main reason that large masses should be used instead of stiff springs.

The stiff spring method is advantageous in the case of enforced displacement because it avoids the roundoff error that occurs while differentiating the displacement to obtain acceleration in the large mass method. The stiff spring method also avoids the problem of rigid-body drift when applying enforced motion on statically determinate support points. (Rigid-body drift means that the displacement increases continuously with time, which is often caused by the accumulation of small numerical errors when integrating the equations of motion.)

7.3   The Large Mass Method in Modal Transient and Modal Frequency Response

The steps described in the previous section must also be followed when a modal method of response analysis is used. Also, if the enforced degrees of freedom are not sufficient to suppress all rigid-body motions, which may be the case for an airplane in flight, additional DOFs that describe the remaining rigid-body motions can also be entered on the SUPORT entry. Use of the SUPORT entry is discussed in "Rigid-body Modes." Note that the use of the SUPORT entry is optional.

The rigid-body mode(s) can be removed from consideration either by not computing them or by using PARAM,LFREQ,r where r is a small positive number (0.001 Hz, for example). If this is done, the displacements, velocities, and accelerations obtained are relative to the overall motion of the structure and are not absolute response quantities. Stresses and element forces are the same as when the rigid-body modes are included because the rigid-body modes do not contribute to them. Rigid-body modes can be discarded to remove rigid-body drift.

Rigid-body modes occur when the structure is unconstrained and large masses are applied at the DOFs which, if constrained, result in a statically determinate structure. Redundant constrained DOFs, which result in a statically indeterminate structure, present a different situation when the constraints are removed and large masses are applied at those redundant DOFs. In that case, very low-frequency modes occur but they are not all rigid-body modes; some are modes that represent the motion of one large mass relative to the others. These very low-frequency, nonrigid-body modes do contribute to element forces and stresses, and they must be retained in the solution. In some cases, their frequencies are not necessarily small; they may be only an order or two in magnitude less than the frequency of the first flexible mode. If PARAM,LFREQ,r is used to remove the true rigid-body modes, then r must be set below the frequency of the first relative motion mode.

Consider the two-dimensional clamped bar in Figure 7-1. Assume that each end of the bar is subjected to the same enforced acceleration time history in the y-direction. One way to model the bar is to use two large masses (one at each end), which are unconstrained in the y-direction. This model provides two very low-frequency modes: one that is a rigid-body mode and one that is not. The second mode contributes to element forces and stresses, and removing its contribution leads

to an error because with two such large masses, one mass can drift over time relative to the other mass. A better way to model the case of identical inputs at multiple locations is to use one large mass connected to the end points by an RBE2 element. This model provides only one rigid-body mode, which can be safely discarded if only the answers relative to the structure are desired.
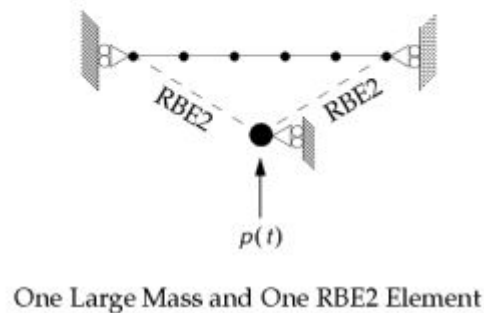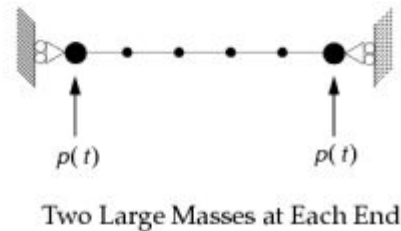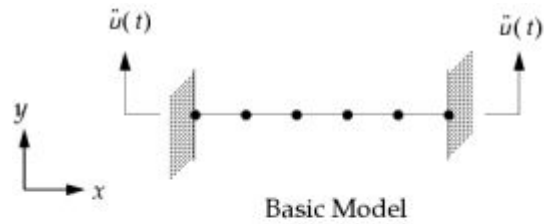
Figure 7-1. Clamped-Clamped Bar Undergoing Enforced Acceleration

7.4   User Interface for the Large Mass Method

There is no special user interface for the large mass method other than to specify a large mass at excitation DOFs and to specify the large force. For transient response the type of enforced motion (displacement, velocity, or acceleration) must be specified on the TLOADi entries. The remainder of the input is identical to that of frequency response ("Frequency Response Analysis") or transient response ("Transient Response Analysis") analysis.

The force applied at a point is the product of terms from the DLOAD, DAREA, and TABLEDi entries. The scaling of the large force is arbitrary; it can be on any one or more of these entries. These entries follow:

DLOAD Format

1	2	3	4	5	6	7	8	9	10
DLOAD	SID	S	S1	L1	S2	L2	-etc.-		

DAREA Format

DAREA	SID	P1	C1	A1	P2	C2	A2		

TABLED1 Format

TABLED1	TID								
	x1	y1	x2	y2	x3	y3	-etc.-	ENDT	

TABLED2 Format

TABLED2	TID	X1							
	x1	y1	x2	y2	x3	y3	-etc.-		

TABLED3 Format

TABLED3	TID	X1	X2						
	x1	y1	x2	y2	x3	y3	-etc.-		

TABLED4 Format

TABLED4	TID	X1	X2	X3	X4				
	A0	A1	A2	A3	A4	A5	-etc.-		

The TABLED4 entry defines a power series and is convenient in frequency response for enforced constant velocity or displacement.

Frequency Response

If a DLOAD entry is used to scale RLOAD1 input, the applied force magnitude in terms of NX Nastran input is

$$P_i(f) \ = \ S \cdot \sum_i (S_i \cdot A_i [C_i(f) + i \cdot D_i(f)])$$

Equation 7-7.

where S and S_i are input on the DLOAD Bulk Data entry, A_i is input on the DAREA entry, and $C_i(f)$ and $D_i(f)$ are input on the TABLEDi entries. Note that the "i" non-subscript term in the expression $i \cdot D_i(f)$ is $\sqrt{-1}$.

If a DLOAD entry is used to scale RLOAD2 input, the applied force magnitude in terms of NX Nastran input is

$$P_i(f) = S \cdot \sum_i (S_i \cdot A_i \cdot B_i(f))$$

Equation 7-8.

where S and S_i are input on the DLOAD Bulk Data entry, A_i is input on the DAREA entry, and $B_i(f)$ is input on the TABLEDi entry.

Specification of the large force value depends upon whether acceleration, velocity, or displacement is enforced.

Enforced Acceleration

Enforced acceleration is the easiest to apply since the required force is directly proportional to the desired acceleration times the large mass:

$$p(\omega) = m_o \ddot{u}(\omega)$$

Equation 7-9.

Enforced Velocity

Enforced velocity requires a conversion factor

$$p(\omega) = i\omega m_o \dot{u} = i(2\pi f) m_o \dot{u}(\omega)$$

Equation 7-10.

For constant velocity $\dot{u}(\omega)$, it may be easiest to use the RLOAD1 and TABLED4 entries because the imaginary term $i \cdot D_i(f)$ of Equation 7-7 and the frequency-dependent term $2\pi f$ can be specified directly.

Enforced Displacement

Enforced displacement also requires a conversion factor

$$p(\omega) = -\omega^2 m_o u = -(2\pi f)^2 m_o u(\omega)$$

Equation 7-11.

For constant displacement $u(w)$, it may be easiest to use the TABLED4 entry because the frequency-dependent term $-(2\pi f)^2$ can be specified directly.

Transient Response

For transient response, the type of enforced motion (displacement, velocity, or acceleration) is specified with the TYPE field (field 5) on the TLOAD1 and TLOAD2 Bulk Data entries.

TLOAD1 Format

1	2	3	4	5	6	7	8	9	10
TLOAD1	SID	DAREA	DELAY	TYPE	TID				

TLOAD2 Format

TLOAD2	SID	DAREA	DELAY	TYPE	T1	T2	F	P	
	C	B							

TYPE 0 (or blank) = applied force (default) TYPE 1 = enforced displacement TYPE 2 = enforced velocity TYPE 3 = enforced acceleration

NX Nastran converts enforced displacements and velocities into accelerations by differentiating once for velocity and twice for displacement. Note that for enforced acceleration, you can specify either force (TYPE = 0 or blank) or acceleration (TYPE = 3); they are the same for the large mass method.

You still need to use the large mass when specifying any type of enforced motion in transient response analysis.

7.5   Examples

This section provides several examples showing the input and output. These examples are:

Model	Analysis Type	Enforced Motion
bd07two	Frequency Response	Constant Acceleration
bd07bar1	Transient Response	Ramp, Acceleration
bd07bar2	Transient Response	Ramp, Displacement
bd07bar3	Transient Response	Ramp, Displacement, Discard Rigid-Body Mode

These examples are described in the sections that follow.

Two-DOF Model

Consider the two-DOF model first introduced in "Real Eigenvalue Analysis" and shown below in Figure 7-2. For this example, apply a constant magnitude base acceleration of 1.0m/sec^2 over the frequency range of 2 to 10 Hz and run modal frequency response with 5% critical damping in all modes. The acceleration input is applied to the large mass (grid point 3). The input file for this model is shown in Figure 7-3.
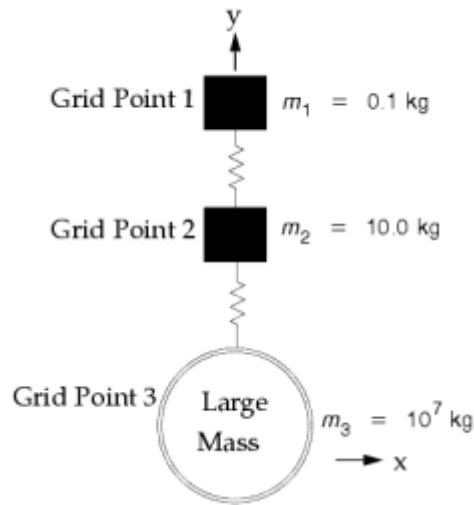
Figure 7-2. Two-DOF Model with Large Mass

```
$   FILE bd07two.dat
$
$   TWO-DOF SYSTEM
$   CHAPTER 7, ENFORCED MOTION
$
TIME 5
SOL 111
CEND
TITLE = TWO-DOF SYSTEM
SUBTITLE = MODAL FREQUENCY RESPONSE
LABEL =  ENFORCED CONSTANT ACCELERATION MAGNITUDE
$
$   SPECIFY MODAL EXTRACTION
METHOD = 10
$
$   SPECIFY DYNAMIC INPUT
DLOAD = 999
FREQ = 888
SDAMPING = 777
$
$   SELECT OUTPUT
DISPLACEMENT(PHASE,PLOT) = ALL
ACCELERATION(PHASE,PLOT) = ALL
$
$   XYPLOTS
$
... X-Y plot commands ...
$
BEGIN BULK
$
$......2.......3.......4.......5.......6.......7.......8.......9.......10.....$
$
$   LARGE MASS AT BASE GRID POINT
CONM2   999     3               1.0E7
$
$   LOAD DEFINITION  (INCLUDES SCALE FACTORS FOR ENFORCED ACCELERATION)
$DLOAD  SID     S       S1      RLOAD1
DLOAD   999     1.0E7   1.0     998
$RLOAD1 SID     DAREA                   TC
RLOAD1  998     997                     901
$DAREA  SID     P1      C1      A1
DAREA   997     3       2       1.0
$TABLED4 TID    X1      X2      X3      X4                              +TAB4
TABLED4 901     0.      1.      0.      100.                            +TAB901
$+TAB4  A0      A1      A2      A3      A4      A5
+TAB901 1.0     ENDT
$
$ MODAL EXTRACTION
$EIGRL  SID     V1      V2      ND      MSGLVL
EIGRL   10      -1.     30.             0
$
$   FREQUENCY RANGE 2-10 HZ
$FREQ1  SID     F1      DF      NDF
FREQ1   888     2.      0.05    160
$
$   MODAL DAMPING OF 5% CRITICAL
$TABDMP1 TID    TYPE
$+TAB1  F1      G1      F2      G2      ETC
TABDMP1 777     CRIT                                                    +TABD7
+TABD7  0.      0.05    100.    0.05    ENDT
$
```

```
... basic model ...
$
ENDDATA
```

Figure 7-3. Input File for Enforced Constant Acceleration

The large mass value is chosen as 1.0E7 kilograms and is input via the CONM2 entry. The scale factor for the load (1.0E7) is input on the DLOAD Bulk Data entry. The factor of 1.0E7 is approximately six orders of magnitude greater than the overall structural mass (10.1 kg). The TABLED4 entry defines the constant acceleration input. (One of the other TABLEDi entries can also be used, but the TABLED4 entry is chosen to show how to use it for enforced constant velocity and displacement later in this example.)

Figure 7-4 shows the X-Y plots resulting from the input point (grid point 3) and an output point (grid point 1). The plots show acceleration and displacement magnitudes. Note that the acceleration input is not precisely 1.0m/sec^2 ; there is a very slight variation between 0.9999 and 1.0000 due to the large mass approximation.
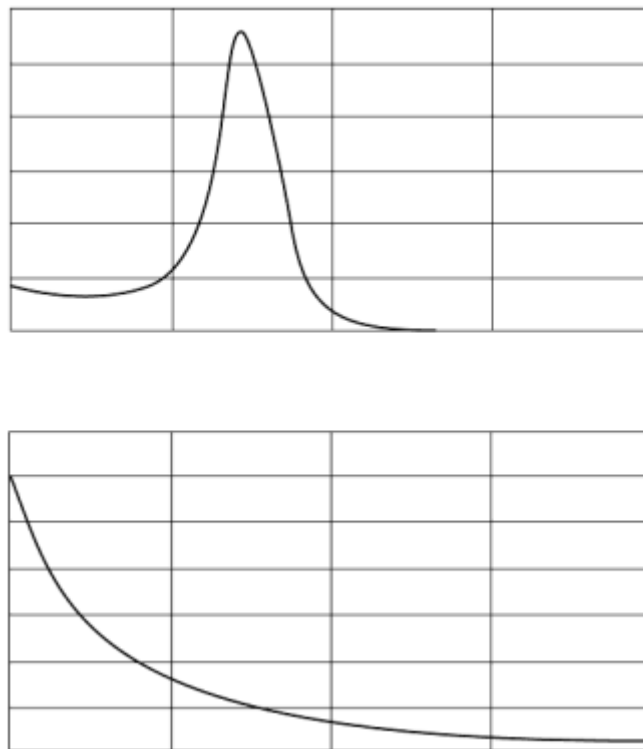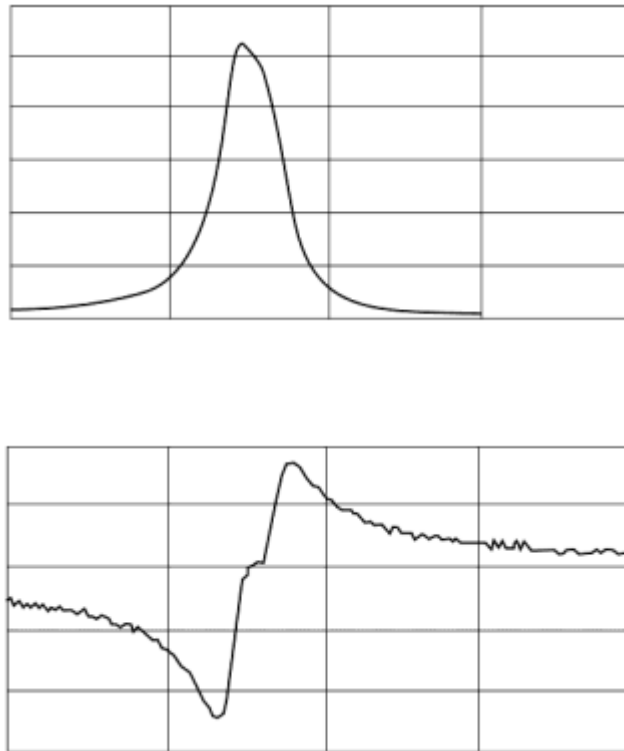
Figure 7-4. Displacements and Accelerations for the Two-DOF Model

This model was analyzed with several values of large mass. Table 7-1 shows the results. Note that the model with the 10^6 mass ratio is the model discussed earlier. Peak frequency response results are compared for each model, and the natural frequencies are compared to those of the constrained model in "Real Eigenvalue Analysis" . The table shows that a mass ratio of 10^6 is a good value to use for this model.

Table 7-1. Models with Different Large Mass Ratios

Ratio of Large Mass to Structure	Natural Frequencies[1] (Hz)	Response Peaks (m/sec^2)			
		$\ddot{x}_{1_{max}}$	$\ddot{x}_{2_{max}}$	$\ddot{x}_{3_{max}}$	$\ddot{x}_{3_{min}}$
10^2	10^{-9} 4.8011 5.3025	52.0552	6.5531	1.0335	0.9524
10^4	10^{-10} 4.7877 5.2910	52.2823	6.7641	1.0003	0.9995
10^6	0.0 4.7876 5.2909	52.2836	6.7661	1.0000	0.9999

Table 7-1. Models with Different Large Mass Ratios					
Ratio of Large Mass to Structure	Natural Frequencies[1] (Hz)	Response Peaks (m/sec²)			
		$\ddot{x}_{1_{max}}$	$\ddot{x}_{2_{max}}$	$\ddot{x}_{3_{max}}$	$\ddot{x}_{3_{min}}$
10^8	0.0 4.7876 5.2909	52.2836	6.7662	1.0000	1.0000

[1]Resonant frequencies for the constrained model are 4.7876 and 5.2909 Hz.

This model can also be changed to apply constant velocity or constant displacement at its base. Figure 7-5 is an abridged input file for the model, showing the Bulk Data entries required for enforced constant acceleration, enforced constant velocity, and enforced constant displacement. Note that only one of these is usually applied to any model, but all three are shown here for comparison purposes.

```
$.......2.......3.......4.......5.......6.......7.......8.......9.......10.....$
$
$ ENTRIES FOR ENFORCED MOTION
$
$ LARGE MASS AT BASE GRID POINT
CONM2 999 3 1.0E7
$
$ LOAD DEFINITION
$DLOAD SID S S1 RLOAD1
DLOAD 999 1.0E7 1.0 998
$DAREA SID P1 C1 A1
DAREA 997 3 2 1.0
$
$.......2.......3.......4.......5.......6.......7.......8.......9.......10.....$
$
$ UNIQUE ENTRIES FOR ENFORCED CONSTANT ACCELERATION MAGNITUDE
$
$RLOAD1 SID DAREA TC
RLOAD1 998 997 901
$TABLED4 TID X1 X2 X3 X4 +TAB4
TABLED4 901 0. 1. 0. 100. +TAB901
$+TAB4 A0 A1 A2 A3 A4 A5
+TAB901 1.0 ENDT
$
$.......2.......3.......4.......5.......6.......7.......8.......9.......10.....$
$
$ UNIQUE ENTRIES FOR ENFORCED CONSTANT VELOCITY MAGNITUDE
$
$RLOAD1 SID DAREA TD
RLOAD1 998 997 902
$TABLED4 TID X1 X2 X3 X4 +TAB4
TABLED4 902 0. 1. 0. 100. +TAB902
$+TAB4 A0 A1 A2 A3 A4 A5
+TAB902 0.0 6.283185 ENDT
$
$.......2.......3.......4.......5.......6.......7.......8.......9.......10.....$ $
$
$ UNIQUE ENTRIES FOR ENFORCED CONSTANT DISPLACEMENT MAGNITUDE
$
$RLOAD1 SID DAREA TC
RLOAD1 998 997 903
$TABLED4 TID X1 X2 X3 X4 +TAB4
TABLED4 903 0. 1. 0. 100. +TAB903
$+TAB4 A0 A1 A2 A3 A4 A5
+TAB903 0.0 0.0 -39.4784 ENDT
```

Figure 7-5. Bulk Data Entries for Enforced Constant Motion

Each input utilizes the TABLED4 entry. The TABLED4 entry uses the algorithm

$$Y = \sum_{i=0}^{N} A_i \left(\frac{x - X1}{X2} \right)^i$$

Equation 7-12.

where x is input to the table, Y is returned, and N is the degree of the power series. When $x<X3$, $X3$ is used for x ; when $x>X4$, $X4$ is used for x . This condition has the effect of placing bounds on the TABLED4 entry; note that there is no extrapolation outside of the table boundaries. There are N+1 entries to this table.

Constant acceleration is the easiest to apply since the force is proportional to the mass for all frequencies. The power series for this case becomes

$$A0 + A1 \left(\frac{x - X1}{X2} \right)$$

Equation 7-13.

where:

A0   =   1.0
$X1$ =   0.0
$X2$ =   1.0

Therefore, these terms define a constant (1.0 in this case).

Constant velocity involves a scale factor that is directly proportional to circular frequency ($2\pi f$). The power series for this case becomes

$$A0 + A1 \left(\frac{x - X1}{X2} \right)$$

Equation 7-14.

where:

A0   =   0.0
A1   = 2π= 6.283185
$X1$ =   0.0
$X2$ =   1.0

Note that a phase change of 90 degrees is also required; this change is input using the TD field (field 7) of the RLOAD1 entry.

Constant displacement involves a scale factor that is proportional to the circular frequency squared $(2\pi f)^2$ with a sign change. The power series for this case becomes

$$A0 + A1\left(\frac{x - X1}{X2}\right) + A2\left(\frac{x - X1}{X2}\right)^2$$

Equation 7-15.

where:

A0 = 0.0
A1 = 0.0
A2 = —$(2\pi)^2$ = —39.4784
X1 = 0.0
X2 = 1.0

Table 7-2 summarizes the coefficients for the power series.

Table 7-2. Coefficients for the Power Series

Type of Excitation	A0	A1	A2
Enforced u	0.0	0.0	—$(2\pi)^2$
Enforced $\dot{u}$	0.0	2π	—
Enforced $\ddot{u}$	1.0	—	—

Cantilever Beam Model

Consider the cantilever beam first introduced in Frequency Response Analysis and shown in Figure 7-6. In this case the planar model is analyzed for bending; therefore, only three DOFs per grid point are considered: T1 (x-translation), T2 (y-translation), and R3 (z-rotation). An acceleration ramp function in the y-direction is enforced at the base (grid point 1) by applying a large mass and a force. T1 and R3 are constrained at grid point 1 since the enforced motion is in only the T2-direction. Modal transient response analysis (SOL 112) is run with 5% critical damping used for all modes. Modes up to 3000 Hz are computed with the Lanczos method. Figure 7-7 shows the idealized ramp function and the NX Nastran implementation. The excitation is not cut off abruptly; instead it is cut off over two time steps. A time step of 0.001 second is used, and the analysis is run for 1.0 second. Figure 7-8 shows the abridged input file.
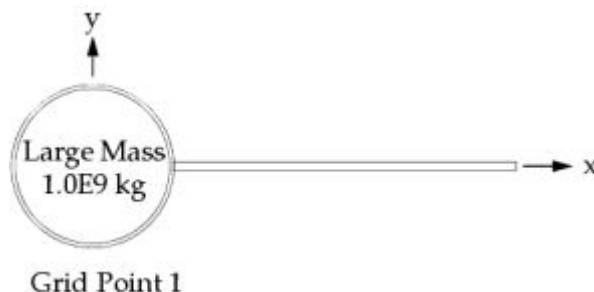
Figure 7-6. Beam Model with Large Mass
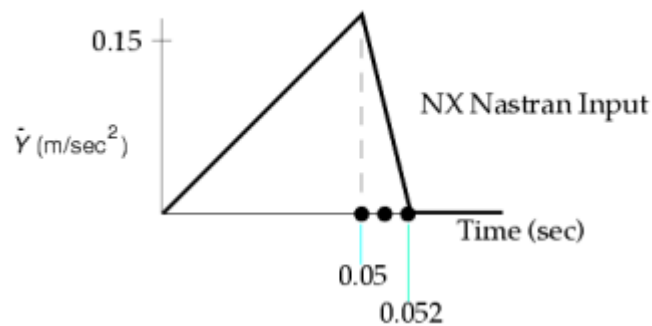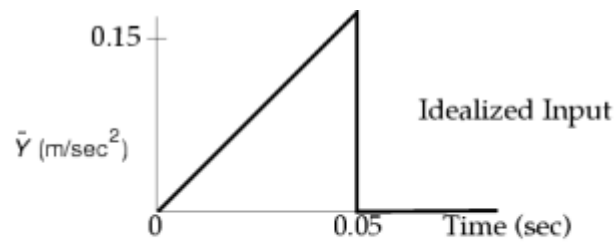
Figure 7-7. Idealized Ramp Function Versus NX Nastran Ramp Function

```
$ FILE bd07bar1.dat
$
$ CANTILEVER BEAM MODEL
$ CHAPTER 7, ENFORCED MOTION
$
SOL 112
TIME 10
CEND
TITLE = CANTILEVER BEAM
SUBTITLE = MODAL TRANSIENT RESPONSE
LABEL = ENFORCED ACCELERATION
$
SPC = 21
DLOAD = 22
TSTEP = 27
SDAMPING = 25
$
METHOD = 10
$
$ PHYSICAL OUTPUT REQUEST
SET 11 = 1,11
DISPLACEMENT(PLOT) = 11
ACCELERATION(PLOT) = 11
$
$ XYPLOTS
$
... X-Y plot commands ...
$
BEGIN BULK
$
$.......2.......3.......4.......5.......6.......7.......8.......9.......10.....$
$
$ LARGE MASS OF 1.0E9
$CONM2 EID G CID M
CONM2 15 1 1.0E9
$
$ CONSTRAIN MASS IN 1,6 DIRECTIONS
SPC 21 1 16
$
$ DYNAMIC LOADING
$DLOAD SID S S1 L1
DLOAD 22 1.0E9 0.102 23
$TLOAD1 SID DAREA DELAY TYPE TID
TLOAD1 23 24 0 25
```

```
$DAREA SID P1 C1 A1
DAREA 24 1 2 0.15
$TABLED1 TID +TABL1
$+TABL1 X1 Y1 X2 Y2 ETC.
TABLED1 25 +TABL1
+TABL1 0.0 0.0 0.05 1.0 0.052 0.0 0.1 0.0 +TABL2
+TABL2 ENDT
$
$ CONVERT WEIGHT TO MASS: MASS = (1/G)*WEIGHT
$ G = 9.81 m/sec**2 --> WTMASS = 1/G = 0.102
PARAM WTMASS 0.102
$
$EIGRL SID V1 V2 MSGLVL
EIGRL 10 -1. 3000. 0
$
$TSTEP SID N1 DT1 NO1
TSTEP 27 1000 0.001 1
$
$ MODAL DAMPING OF 5% IN ALL MODES
$TABDMP1 TID TYPE +TABD
$+TABD F1 G1 F2 G2 ETC.
TABDMP1 25 CRIT +TABD
+TABD 0. 0.05 1000. 0.05 ENDT
$
... basic model ...
$
ENDDATA
```

Figure 7-8. Abridged Input File for Enforced Acceleration

A large mass of 1.0E9 kg is placed at grid point 1. This grid point is constrained in the T1- and R3-directions but is free in the T2-direction. The load is scaled to give a peak input acceleration of 0.15m/sec^2. This scaling is performed by applying a scale factor of 1.0E9 in the S field (field 3) of the DLOAD entry, a scale factor of 0.102 in the S1 field (field 4) of the DLOAD entry, and a factor of 0.15 in the A1 field (field 5) of the DAREA entry. The applied load is scaled by 0.102 because the large mass is also scaled by 0.102 due to the PARAM,WTMASS entry (see Equation 7-5). The time variation is specified with the TABLED1 entry. The TLOAD1 entry specifies the type of loading (field 5) as 0 (applied force); this gives the same answers if the type is specified as 3 (enforced acceleration).

Figure 7-9 shows the displacement and acceleration response at grid points 1 (base) and 11 (tip). Note that at the end of the acceleration pulse the base has a constant velocity, and therefore, a linearly increasing displacement.
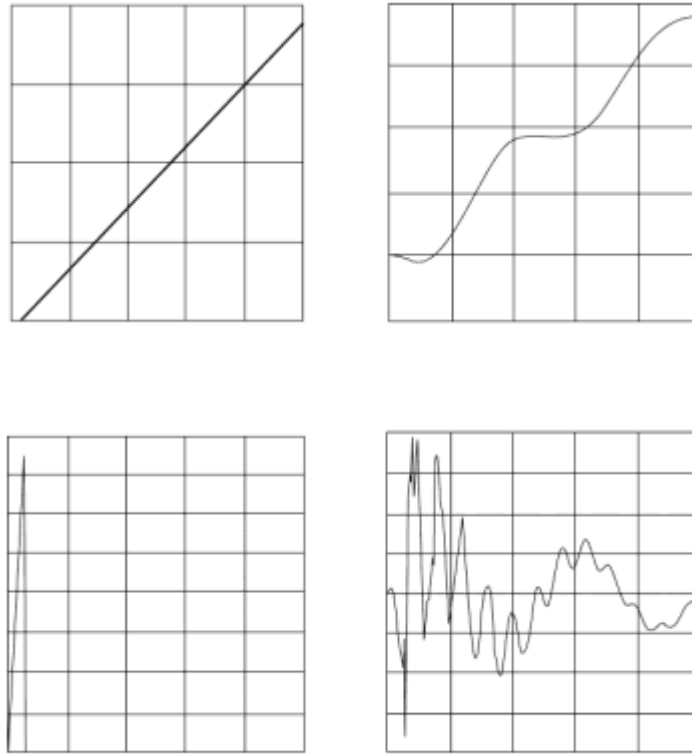
Figure 7-9. Response for Enforced Acceleration

Next, consider the same model with a 0.015 meter displacement imposed instead of an acceleration. The same ramp time history function is used (with a peak enforced displacement of 0.015 meter) so that the only change to the input file is to change the excitation type from 0 (applied force) to 1 (enforced displacement on field 5 of the TLOAD1 entry) and the amplitude in the DAREA entry from 0.15 to 0.015. Figure 7-10 shows the idealized input displacement time history. Figure 7-11 shows the displacement and acceleration response at grid points 1 and 11.
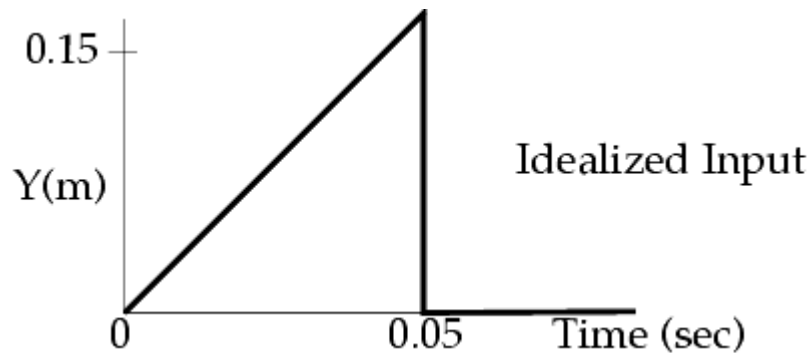
Figure 7-10. Response for Enforced Displacement (With the Rigid-Body Mode)
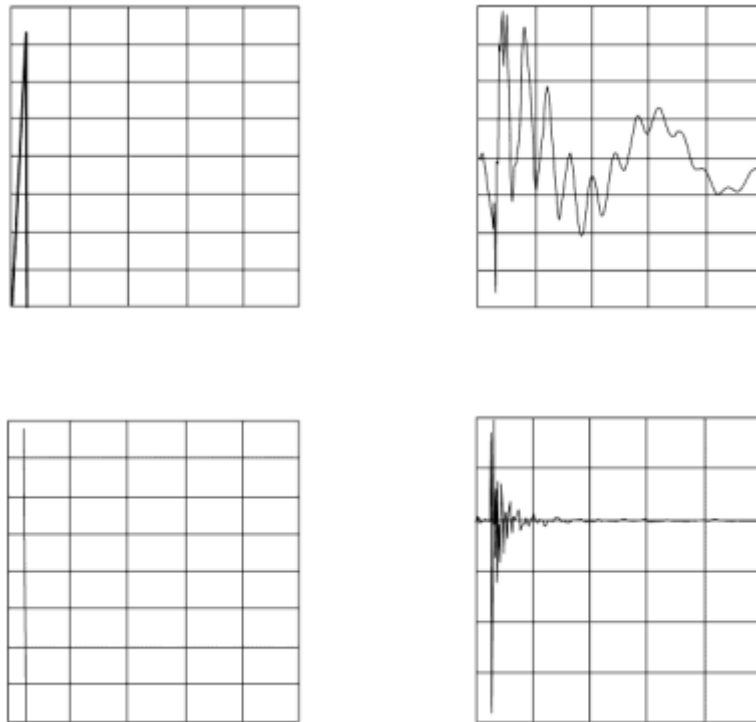
Figure 7-11. Response for Enforced Displacement (Without the Rigid-Body Mode)

Now, consider a change to the enforced displacement run. In this case, remove the rigid-body mode's contribution either by not computing the rigid-body mode (by setting V1 to a small positive value, such as 0.01 Hz) or by neglecting the rigid-body mode in the transient response (by setting PARAM,LFREQ to a small positive number, such as 0.01 Hz). Figure 7-12 shows the resulting displacement and acceleration responses at grid points 1 and 11. Note that the responses are relative to the structure and are not absolute. The relative displacement of grid point 1 should be zero, and it is very close to zero (i.e., 10^{-10}) as a result of the sufficiently large mass.
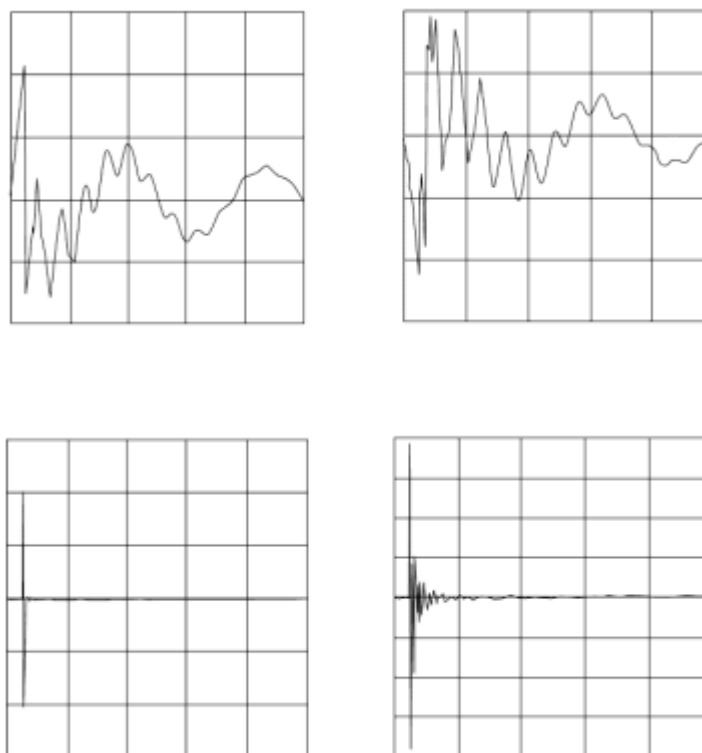
Figure 7-12. Response for Enforced Displacement (Without the Rigid-Body Mode)

Chapter

8   Restarts in Dynamic Analysis

- *Overview*

- *Automatic Restarts*

- *Structure of the Input File*

- *User Interface*

- *Determining the Version for a Restart*

- *Examples*

8.1   Overview

A restart is a logical way of continuing from a previous run without having to start from the beginning. The savings can be in terms of both time and money. Some forms of restarts are used practically every day perhaps without you realizing it. An example of a restart can be as simple as reading a book. Normally, you probably do not finish reading a book in one continuous stretch. You may read a hundred pages today and another fifty pages tomorrow, and so on. Each time that you continue from where you left off previously is a restart. It is much more time consuming and impractical to start from page one every time that you pick up the book.

This analogy can be applied to NX Nastran. In the case of a static analysis, the most expensive and time consuming part of the run is the decomposition of the stiffness matrix. This fact is especially true for large models. Now suppose after completing the original run, you want to obtain additional output (e.g., stresses, displacements, etc.) or add more load cases. You can always start from the beginning and redo the whole problem, or you can perform a restart at a fraction of the time and cost. In the case of additional output requests and additional load conditions, the decomposition of the stiffness matrix, which was performed in the previous run, is not redone if restart is used.

In dynamic analysis, the calculation of normal modes is, in general, the most expensive operation. Therefore, a common application of restart is the performance of a transient or frequency response analysis by restarting from the normal modes calculation, which was saved in the database from a previous run. This restart process avoids the recalculation of the normal modes.

The cost of restarting is measured in the disk space required to store data blocks for subsequent use. Judging whether to not save data blocks and simply rerun the analysis or to save data blocks for restarting is determined by several factors, such as the amount of available disk space and your computer's solution speed. Note that the database can be copied to tape which then provides you with more free disk space. When you are ready to perform a restart, this database can then be copied from the tape back to your disk.

8.2   Automatic Restarts

NX Nastran's Executive System effectively uses modern database technology. Among its many features, the Executive System implements the NASTRAN Data Definition Language (NDDL). With NDDL, NX Nastran is able to use automatic restart logic with the superelement solution sequences. For more information on the NDDL, refer to the *NX Nastran DMAP Programmer's Guide*.

In order to maintain upward compatibility, we retained the old solution sequences and added a new series of superelement solution sequences. These solution sequences are also known as the Structured Solution Sequences (solution sequences numbers greater than 100). The automatic restart logic is available only for these solution sequences. Restart is no longer available for the rigid format solution sequences. Use these new Structured Solution Sequences rather than the old ones since all the new features are automatically implemented in these new solution sequences. Improvements made to the automatic restart logic for Version 67 and later, have made it even more efficient and robust. In this chapter, we will address the restart logic for the Structured Solution Sequences. Note that you are not required to use superelements in order to use the superelement solution sequences. If you are familiar with the rigid format solutions, converting to the new Structured Solution Sequences only requires that you replace the solution command. For example, a normal modes run only requires replacing the SOL 3 command with the SOL 103 command in the Executive Control Section.

8.3   Structure of the Input File

Before presenting details on how restarts work, the following simple flow diagram of the NX Nastran input file structure may be beneficial:

Table 8-1. Structure of the NX Nastran Input File

NASTRAN Statement	Optional
File Management Statements	Optional
Executive Control Statements	Required Section
CEND	Required Delimiter
Case Control Commands	Required Section
BEGIN BULK	Required Delimiter
Bulk Data Entries	Required Section
ENDDATA	Required Delimiter

This order must be followed as shown for all NX Nastran input files. For details regarding the statements, commands, and entries, see the *NX Nastran Quick Reference Guide*.

8.4   User Interface

There are two types of runs—cold start and restart runs—which are described below.

Cold Start Run

The first run, which is called the cold start run, is identical to what you usually do for submitting an NX Nastran job with the exception that the database must be saved. In addition to your normal output files (e.g., F06 file), four database files are created as a result of this run, assuming that the default database initialization values are used. The convention for the filenames is machine dependent; therefore, you should refer to the machine-dependent documentation for your specific computer for the exact syntax. For a typical UNIX machine with an NX Nastran input file called "run1.dat", the following sample submittal command can be used:

```
nastran run1
```

where "nastran" is the name of the shell script for executing NX Nastran. Note that the above submittal command assumes that you have not modified the "scr=no" option as your default nastran submittal option. The "scr=no" option saves the database at the end of the run, which is required if you intend to perform restarts. The following four database files are created as a result of the above command.

```
run1.DBALL
run1.MASTER
run1.USROBJ
run1.USRSOU
```

The last two files are not needed and can be deleted if you do not want to store your own DMAP, which is usually the case.

Restart Run

NASTRAN Statement Section

This section is normally the same as your cold start run. The BUFFSIZE must not be changed in a restart run.

File Management Section (FMS)

This section tells NX Nastran that you are performing a restart run. The RESTART statement is required in any restart run. The general format for the RESTART statement is as follows:

```
RESTART VERSION=a,b
```

where "a" is the version from which you restart (default value for "a" is LAST) and "b" indicates whether version "a" is to be kept (KEEP) or deleted (NOKEEP) at the end of the run. The default value for "b" is NOKEEP. Due to the default values, the following two restart statements are identical:

```
RESTART VERSION=LAST,NOKEEP
RESTART
```

Whenever a restart is performed, a new version number is automatically created in the database. For each restart, the current version number is incremented by one regardless of whether the job ran successfully or not. There are two exceptions to this rule and they are discussed later on. You also need to tell NX Nastran which database you want to attach to your current run. There are several ways to accomplish this; one way is to use the ASSIGN statement. For example, if you are restarting from the database created by run1.dat, the following FMS statements can be used in your current run:

```
RESTART
ASSIGN   MASTER='run1.MASTER'
```

For UNIX machines, the filenames are case sensitive. They should be entered exactly as they were created in the cold start run and enclosed with single quotes as shown above.

An alternate way is to use the DBS statement on the submittal line instead of the ASSIGN statement. Assuming the current run is called run2.dat, then the equivalent submittal statement is as follows:

```
    nastran run2 dbs=run1
```

The ASSIGN statement is not needed in this case as long as the run1 database files are contained in the submittal directory. Again the syntax is for a typical UNIX machine. You should refer to the machine-dependent documentation for your particular computer for specific details.

Executive Control Section

This section is the same as your normal run with the exception of perhaps the "SOL x" command. For example, if you are performing a normal modes analysis in run1.dat, then the "SOL x" command in run1.dat should reference "SOL 103". In run2.dat, if you are performing a modal transient restart from run1.dat, then the "SOL x" statement in this case should then reference "SOL 112". DIAGs can be turned on or off.

Case Control Section

The automatic restart logic compares the modifications made to the Case Control and Bulk Data Sections in determining which operations need to be processed upon restart. Therefore, you must

be very careful with the changes that you make in your restart run. Adhering to the following rules will avoid unnecessary reprocessing of previously completed operations.

- You must include all "solution-type" related Case Control commands, which are unchanged as compared to the cold start run, in your restart run. In other words, do not make unnecessary LOAD, SPC, MPC, or METHOD command changes or remove them from the Case Control Section unless these are actual changes. This process is clarified later with the example problems.

- Output requests can be modified. A typical example can be a request of the eigenvector printout which was not requested in the cold start run.

Bulk Data Section

As mentioned in the previous section, the automatic restart logic compares the changes made in the Bulk Data Section and determines the path that it follows. A copy of the Bulk Data is stored for each version. The restart run must not contain any Bulk Data entry that was included in the previous runs and saved in the database. The Bulk Data Section in the current restart run should contain only new entries, changed entries, and/or the deletion of old entries from the database. This philosophy is slightly different than the one used in the Case Control Section.

Table 8-2 shows an example where a normal modes run is performed in the cold start run. The eigenvalue table and the grid point weight generator output are requested in this run. For the restart run, the eigenvector output is desired; furthermore, reprinting the grid point weight generator output is not desired. One way to accomplish this is to delete the existing "PARAM,GRDPNT,0" entry from the database.

Table 8-2. Listing of the Cold Start and Restart Input Files

Cold Start Run	Restart Run

```
$
$ FILENAME  - run1.dat
$
$ TYPICAL UNIX SUBMITTAL
$ COMMAND
$
$ nastran run1
$
SOL 103
TIME 5
CEND
TITLE = COLDSTART RUN
METHOD = 10
SPC = 1
BEGIN BULK
$
GRID,1,,0.,0.,0.
GRID,2,,10.,0.,0.
CROD,1,10,1,2
PROD,10,1,1.0
MAT1,1,1.+7,,.32
EIGRL,10,,,1
CONM2,100,2,,10.
PARAM,WTMASS,.00259
$
$ REQUEST FOR GRID POINT
$ WEIGHT GENERATOR
$ OUTPUT
$
PARAM,GRDPNT,0
$
```

```
$
$ FILENAME - run2.dat
$
$ TYPICAL UNIX SUBMITTAL
$ COMMAND
$
$ nastran run2
$
RESTART
ASSIGN MASTER='run1.MASTER'
SOL 103
TIME 5
CEND
TITLE = RESTART RUN
METHOD = 10
SPC = 1
$
$ REQUEST FOR EIGENVECTOR
$ PRINTOUT
$
DISP = ALL
BEGIN BULK
$
$ SKIP GRID POINT WEIGHT
$ GENERATOR OUTPUT
$
/,7
$
ENDDATA
```

Table 8-2. Listing of the Cold Start and Restart Input Files

Cold Start Run	Restart Run

```
SPC1,1,123456,1
SPC1,1,23456,2
$
ENDDATA
```

The format for the delete entry is "/,K1,K2" where K1 and K2 are the sorted Bulk Data sequence number of the first and last entries in the sequence to be removed, respectively. In the event that K2 = K1, the following two entries are identical:

```
/,K1,K2
/,K1
```

The values of K1 and K2 can be obtained in the F06 file of your cold start run as long as the Bulk Data is echoed with the "ECHO = SORT" Case Control command, which is the default option. From Figure 8-1, the sorted Bulk Data count for "PARAM,GRDPNT,0" is "7"; therefore, the delete entry in this case can be either "/,7,7" or "/,7". No additional Bulk Data entry is required for this restart run since there are no other changes involved for this model. Note that the same SPC and METHOD commands are used in both the cold start and restart runs since neither the actual boundary condition nor the desired eigenvalue calculation has changed.

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

 CARD
 COUNT      .   1  ..   2  ..   3  ..   4  ..   5  ..   6  ..   7  ..   8  ..   9  ..  10  .
   1-       CONM2  100   2            10.
   2-       CROD   1     10     1      2
   3-       EIGRL  10                  1
   4-       GRID   1            0.      0.      0.
   5-       GRID   2            10.     0.      0.
   6-       MAT1   1     1.+7           .32     .097
   7-       PARAM  GRDPNT 0
   8-       PARAM  WTMASS .00259
   9-       PROD   10    1      1.0
  10-       SPC1   1     23456  2
  11-       SPC1   1     123456 1
            ENDDATA
      TOTAL COUNT=        12
```

Figure 8-1. Echo of the Sorted Bulk Data Input for the Cold Start Run

You can include as many of these delete entries as necessary. However, if the case requires many changes, it is probably more convenient to delete the entire old Bulk Data Section from the database and replace it with the fully revised Bulk Data Section, including the new and modified entries. This operation can be accomplished by including the full new Bulk Data plus the following entry in the Bulk Data Section of the restart run:

```
/1,i
```

where i is any positive integer that is greater than or equal to the number of Bulk Data entries from your cold start run.

For conventional dynamic analysis (i.e., non-superelement), restarts involving model changes (e.g., changing the thickness of a plate) are not very efficient. Therefore, the savings is probably minimal, if any. However, in the case of additional output requests or a restart from a modes run to a response run, the savings can be substantial. This type of restart is covered extensively in "Examples" . For superelement analysis, even restarts involving model changes can be beneficial as long as these changes are localized.

8.5   Determining the Version for a Restart

Not all versions in the database are restartable. For each run, a message is printed near the top of the F04 file indicating the version number of the current job. In general, if a job completes without any fatal messages, then that particular version is restartable. It is a good idea to keep a brief log of all the restartable versions since this is probably the most reliable source of information regarding whether a version is restartable. If a restart job failed, e.g., due to Bulk Data error, then this newly created version is not restartable, which is indicated by the following error message at the bottom of the F06 file.

```
 *** USER WARNING MESSAGE 784 (XCLNUP) VERSION = yyy PROJECT = "zzz"
OF THIS DATA BASE IS NOT VALID FOR RESTART PURPOSES.
USER ACTION: SUBSEQUENT RESTARTS SHOULD REFERENCE VERSION = xxx
OR A PRIOR VALID VERSION
```

If the version is not restartable, you must restart from a previous valid version. The "xxx" and "yyy" above denote version numbers. The "zzz" denotes a project description provided by you. This project description is alphanumeric and can contain up to 40 characters (the default is blank). The project description is often not used and is an optional statement.

If, for some reason, the records for the old runs are no longer available, then the DBDIR FMS statement can be used to query the database contents to find out which versions are being stored in the database. The following simple setup is all that is required for this purpose.

```
ASSIGN MASTER='ddddd.MASTER'
DBDIR VERSION=*,PROJECT=*
ENDJOB
```

where ddddd.MASTER is the name of the database being used.

The Executive Control, Case Control, or Bulk Data Section is not required in this case. Furthermore, a new version is not created in this case. Near the top of the F06 output, a PROJECT VERSION DIRECTORY TABLE is printed listing all the versions in the database. A "**" next to a version number indicates that this particular version was deleted from the database. This deletion may be due to the NOKEEP option or the use of the DBCLEAN command when performing a restart run. A version number with a "**" next to it is not restartable. A version number without a "**" next to it is restartable if the run that created the version did not terminate with UWM 784. For more advanced users, this DBDIR FMS statement can also be used to check the database directory for the existence of data blocks, e.g., UG. If the job fails very early in the run (e.g., error in the FMS section), then a new version may not be created.

It is always good practice to back up the database on a regular basis. If the system aborts the run (e.g., disk space exhausted or time quota exceeded on a system level), then there is a chance that the database is corrupted and will not allow restarts. Another good practice to ensure that only good models are retained in the database is to perform the following:

• Use RESTART VERSION = a,KEEP

  where "a" is a version number.

• If a version contains errors or is no longer of interest, then use the FMS statement DBCLEAN to remove obsolete or incorrect versions of the model from the database. Removing these versions allows the Executive System to reuse some of this space for new versions.

8.6   Examples

The examples perform a typical series of runs starting from a normal modes run and restarting into transient and frequency response analyses. Table 8-3 summarizes this series of nine runs

along with a brief description. Listings of the nine runs are also included (Figures 8-2 through 8-10).

			Table 8-3. Typical Series of Restart Runs		
Run Sequence Number	**Name of Input File**	**Solution Sequence Number**	**Description of Runs**	**Version Created**	**Version Deleted**
1	bd08bar1.dat	103	Perform a normal modes cold start analysis and save the database.	1	None
2	bd08bar2.dat	103	Restart from run number 1 and request eigenvector output.	2	None
3	bd08bar3.dat	103	The first two modes of the structure are very close to one of the forcing frequencies. The structure is modified in order to stay away from resonance. This run restarts from run number 2 to delete the old PBAR entry and replace it with the modified PBAR entry. The modes are recalculated, and the eigenvector output is requested in this run.	3	2
4	bd08bar4.dat	N/A	This run deletes the data blocks in the database previously occupied by Version 1. This space can then be reused by future restarts. Note that the statement DBCLEAN does not reduce the database size. It merely removes some of its contents so that this space can be reused. A new version is not created as a result of this run. This is an optional run especially if disk space is of no concern to you.	None	1

Table 8-3. Typical Series of Restart Runs

Run Sequence Number	Name of Input File	Solution Sequence Number	Description of Runs	Version Created	Version Deleted
5	bd08bar5.dat	112	This is a transient restart run from the modes saved in Version 3. The applied load is a unit step function. The modes calculated in run number 3 are also saved at the end of this run. Since the calculation of the modes is the most expensive operation in a dynamic analysis, it is probably a good idea to save Version 3 once you have confidence in the results. This way you can always restart from this version. Partial output is shown at the top of Figure 8-11. A 1% critical damping value is applied to the structure.	4	None
6	bd08bar6.dat	112	This is another transient restart run using the solution from run number 5. The purpose of this run is to request additional output. Partial output is shown at the bottom of Figure 8-11. Note that the maximum displacement at grid point 11 is the same as in run number 5 as expected since you are asking for data recovery on the same point due to the same loading condition.	5	4

Table 8-3. Typical Series of Restart Runs					
Run Sequence Number	Name of Input File	Solution Sequence Number	Description of Runs	Version Created	Version Deleted
7	bd08bar7.dat	112	This is another restart run from Version 6 with a different load condition (triangular pulse). Partial output is shown in Figure 8-12. In this case, you can just as easily restart from Version 3.	6	5
8	bd08bar8.dat	111	This is a frequency response restart run. Note that this can also be restarted from Version 3 since it was saved in the database. Partial output is shown in Figure 8-13. A 2% critical damping value is applied to the structure.	7	6
9	bd08bar9.dat	N/A	This is a database directory printout run. As shown in Figure 8-14, there are seven versions in the database. Run numbers 4 and 9 did not create any new versions. Only Versions 3 and 7 are restartable. This is an optional run.	None	None

Remark

If the results for run number 1 are not going to be used for any future purposes, then you may consider making run number 3 as a cold start run instead of a restart run. Model changes do not save you much time, if any, in a non-superelement analysis. By making run 3 a cold start run, you reduce the total amount of disk space required. In this case, run number 4 is not necessary since you are starting with a new database. However, if you want to keep both physical models in the database, then run number 3 should be a restart run as shown in this example. An application of this can be a parametric study of two different configurations. This type of restart allows you to make efficient data recovery or response analysis from two different physical models. However, this type of restart is not used often in a non-superelement analysis since, in general, it is not very efficient. However, in a superelement analysis (see "Advanced Dynamic Analysis Capabilities"), this type of restart can still be very efficient since the changes can be localized to a small region.

```
$   FILE - bd08bar1.dat
$
$   NORMAL MODES RUN
```

```
$
ID CANT BEAM
SOL 103
TIME 10
CEND
TITLE =  CANTILEVER BEAM - NORMAL MODES - COLD START RUN
SPC = 1
METHOD = 10
$
BEGIN BULK
$
CBAR    1       1       1       2       0.      1.      0.
CBAR    2       1       2       3       0.      1.      0.
CBAR    3       1       3       4       0.      1.      0.
CBAR    4       1       4       5       0.      1.      0.
CBAR    5       1       5       6       0.      1.      0.
CBAR    6       1       6       7       0.      1.      0.
CBAR    7       1       7       8       0.      1.      0.
CBAR    8       1       8       9       0.      1.      0.
CBAR    9       1       9       10      0.      1.      0.
CBAR    10      1       10      11      0.      1.      0.
EIGRL   10      -0.1    50.
GRID    1               0.0     0.      0.
GRID    2               0.3     0.      0.
GRID    3               0.6     0.      0.
GRID    4               0.9     0.      0.
GRID    5               1.2     0.      0.
GRID    6               1.5     0.      0.
GRID    7               1.8     0.      0.
GRID    8               2.1     0.      0.
GRID    9               2.4     0.      0.
GRID    10              2.7     0.      0.
GRID    11              3.0     0.      0.
MAT1    1       7.1+10                  .33     2.65+4
PARAM   AUTOSPC YES
PARAM   WTMASS  .102
PBAR    1       1       6.158-4 3.-8    3.-8    6.-8    2.414
SPC1    1       123456  1
$
ENDDATA
```

Figure 8-2. Input File for Normal Modes Run

```
$  FILE - bd08bar2.dat
$
$  NORMAL MODES RUN
$  REQUEST EIGENVECTOR PRINTOUTS FROM PREVIOUS RUN
$
RESTART VERSION=1,KEEP
ASSIGN MASTER='bd08bar1.MASTER'
$
ID CANT BEAM
SOL 103
TIME 10
CEND
TITLE =  EIGENVECTORS DATA RECOVERY RESTART RUN
SPC = 1
METHOD = 10
DISP = ALL  $  PRINT EIGENVECTORS
$
BEGIN BULK
$
ENDDATA
```

Figure 8-3. Input File for Requesting Eigenvectors

```
$  FILE - bd08bar3.dat
$
$  NORMAL MODES RUN
$  MODIFY PBAR
$
RESTART
ASSIGN MASTER='bd08bar1.MASTER'
$
ID CANT BEAM
SOL 103
```

```
TIME 10
CEND
TITLE =  CANTILEVER BEAM - NORMAL MODES - RESTART RUN
SPC = 1
METHOD = 10
DISP = ALL
$
BEGIN BULK
$
$  DELETE OLD PBAR ENTRY, LINE 26 OF SORTED BULK DATA COUNT
$
/,26
$
$  ADD NEW PBAR ENTRY
$
PBAR,1,1,6.158-4,2.9-8,3.1-8,6.-8,2.414
$
ENDDATA
```

Figure 8-4. Input File for Modifying a Bar Element

```
$  FILE -  bd08bar4.dat
$
assign master='bd08bar1.MASTER'
dbclean version=1
endjob
```

Figure 8-5. Input File for Cleaning a Database

```
$  FILE - bd08bar5.dat
$
$  THIS IS A TRANSIENT RESTART RUN FROM THE MODES
$  CALCULATED BY THE RUN "bd08bar3.dat"
$
RESTART VERSION=3,KEEP
ASSIGN MASTER='bd08bar1.MASTER'
ID CANT BEAM
SOL 112
TIME 10
CEND
TITLE    =  TRANSIENT RESTART - UNIT STEP FUNCTION INPUT
SUBTITLE =  REQUEST DISPLACEMENT TIME HISTORY AT GRID POINT 11
SPC = 1
METHOD = 10
SET 1 = 11
DISP = 1
SUBCASE 1
  SDAMP   = 100
  TSTEP   = 100
  DLOAD   = 100
$
BEGIN BULK
$
$   ADDITIONAL ENTRIES FOR DYNAMIC LOADS
$   FOR UNIT STEP FUNCTION
$
$        SID    DAREA   DELAY   TYPE    TID
TLOAD1   100    101                     102
$
DAREA,101,11,3,1.0
$
TABLED1,102,,,,,,,,+TBL1
+TBL1,0.0,0.0,.001,1.0,10.0,1.0,ENDT
$
$   TRANSIENT TIME STEPS
$
$        SID    N(1)    DT(1)   NO(1)
TSTEP    100    600     .001     5
$
$   MODAL DAMPING TABLE
$
TABDMP1,100,CRIT,,,,,,,+TDAMP
+TDAMP,0.,.01,200.,.01,ENDT
$
ENDDATA
```

Figure 8-6. Input File for Transient Response

```
$  FILE - bd08bar6.dat
$
$  THIS IS ANOTHER TRANSIENT RESTART RUN.  THE PURPOSE
$  OF THIS RUN IS TO REQUEST ADDITIONAL OUTPUT.
$
RESTART
ASSIGN MASTER='bd08bar1.MASTER'
ID CANT BEAM
SOL 112
TIME 10
CEND
TITLE   = T R A N S I E N T    R E S T A R T
SUBTITLE = U N I T    S T E P   F U N C T I O N   I N P U T
SPC = 1
METHOD = 10
SET 1 = 11
SET 2 = 10
ACCE  = 2
SUBCASE 1
  SDAMP   = 100
  TSTEP   = 100
  DLOAD   = 100
$
$
$  PLOT RESULTS
$
$...X-Y plot commands ...
$
BEGIN BULK
$
$
ENDDATA
```

Figure 8-7. Input File for an Additional Output Request

```
$  FILE - bd08bar7.dat
$
$  THIS IS ANOTHER TRANSIENT RESTART RUN USING
$  A DIFFERENT LOAD CONDITION.  NOTE THAT SINCE
$  THERE ARE NO MODEL CHANGES, THE SAME MODES
$  WERE USED FROM THE DATABASE FOR THE RESPONSE
$  CALCULATIONS.
$
RESTART
ASSIGN MASTER='bd08bar1.MASTER'
ID CANT BEAM
SOL 112
TIME 10
CEND
$
$  NOTE THAT TITLE CHANGES HAVE NO EFFECT
$  ON SOLUTION PROCESS, THEY ONLY CHANGE THE
$  PRINTOUT TITLE
$
TITLE   = T R A N S I E N T    R E S T A R T
SUBTITLE = TRIANGLE PULSE - 1.0 AT T=0 AND 0.0  AFTER .2 SEC
SPC = 1
METHOD = 10
SET 1 = 11
DISP = 1
SUBCASE 1
  SDAMP   = 100
  TSTEP   = 100
  DLOAD   = 300
$
$  PLOT RESULTS
```

```
$
$...X-Y plot commands ...
$
BEGIN BULK
$
$          SID      DAREA   DELAY    TYPE     TID
TLOAD1     300      301                       302
$
DAREA,301,11,3,1.0
$
TABLED1,302,,,,,,,,+TBL3
+TBL3,0.0,0.0,.001,1.0,.20,0.0,10.0,0.0,+TBL4
+TBL4,ENDT
$
ENDDATA
```

Figure 8-8. Input File for an Additional Transient Load

```
$  FILE - bd08bar8.dat
$
$  THIS IS RESTART RUN TO PERFORM FREQUENCY RESPONSE
$
RESTART
ASSIGN MASTER='bd08bar1.MASTER'
ID CANT BEAM
SOL 111
TIME 10
CEND
TITLE =  CANTILEVER BEAM - FREQUENCY RESPONSE RESTART
SPC = 1
METHOD = 10
SET 1 = 11
DISP(PHASE) = 1
SUBCASE 1
$
$  A TWO-PERCENT CRITICAL DAMPING IS
$  APPLIED TO THIS RUN AS OPPOSED TO
$  ONE-PERCENT CRITICAL DAMPING IN
$  THE TRANSIENT ANALYSIS
$
  SDAMP   = 1000
  DLOAD   = 1000
  FREQ    = 1000
$
$  PLOT RESULTS
$
$...X-Y plot commands ...
$
BEGIN BULK
$
$   ADDITIONAL ENTRIES FOR FREQUENCY RESPONSE
$
$          SID      DAREA    M        N        TC       TD
$
RLOAD1     1000     1001                       1002
$
DAREA,1001,11,3,0.1
$
TABLED1,1002
,0.,1.,200.,1.,ENDT
$
$    FORCING FREQUENCIES
$
$  RESONANT FREQUENCIES
$
FREQ,1000,2.03174,2.100632,12.59101,13.01795
```

```
FREQ,1000,34.90217,36.08563
$
$  SPREAD THROUGHOUT
$  FREQUENCY RANGE OF INTEREST
$  WITH BIAS BETWEEN
$  HALF-POWER POINTS
$
FREQ,1000,1.437,1.556,1.675,1.794,1.913
FREQ,1000,2.046,2.059,2.073,2.087
FREQ,1000,2.224,2.347,2.47,2.593,2.716
FREQ,1000,8.903,9.641,10.378,11.116,11.853
FREQ,1000,12.676,12.762,12.847,12.933
FREQ,1000,13.781,14.543,15.306,16.068,16.831
FREQ,1000,24.680,26.724,28.769,30.813,32.858
FREQ,1000,35.139,35.376,35.612,35.849
FREQ,1000,41.189,46.292,51.395,56.499,61.602
$
FREQ1,1000,0.,.5,200
$
$    DAMPING
$
TABDMP1,1000,CRIT,,,,,,,+DAMP
+DAMP,0.,.02,200.,.02,ENDT
$
ENDDATA
```

Figure 8-9. Input File for Frequency Response Analysis

```
$  FILE -  bd08bar9.dat
$
assign master='bd08bar1.MASTER'
dbdir
endjob
```

Figure 8-10. Input File to Print the Database Dictionary

```
     *****     PARTIAL OUTPUT FROM bd08bar5.f06    *****     POINT-ID =    11
                               D I S P L A C E M E N T   V E C T O R

      TIME       TYPE        T1            T2            T3           R1           R2            R3
      .0          G         .0            .0            .0           .0          .0            .0
    5.000000E-03  G     9.024590E-22  1.614475E-21  1.923969E-05   .0      -3.001072E-05  3.518218E-20
    1.000000E-02  G     3.367912E-21  8.536304E-21  7.739825E-05   .0      -1.118816E-04  1.271978E-19
                                                        .
    2.300003E-01  G     3.941339E-19  8.273585E-18  7.832402E-03   .0      -3.655237E-03  5.515358E-18
    2.350003E-01  G     3.956722E-19  8.583470E-18  7.862186E-03   .0      -3.686721E-03  5.464303E-18
    2.400003E-01  G     3.957035E-19  9.029934E-18  7.880123E-03   .0      -3.709195E-03  5.756178E-18
    2.450003E-01  G     3.944803E-19  9.498899E-18  7.886210E-03   .0      -3.726098E-03  6.049827E-18
    2.500003E-01  G     3.927639E-19  9.878397E-18  7.884440E-03   .0      -3.754001E-03  6.040225E-18
    2.550003E-01  G     3.909298E-19  1.020556E-17  7.873885E-03   .0      -3.796297E-03  5.837433E-18
    2.600002E-01  G     3.885339E-19  1.063947E-17  7.844862E-03   .0      -3.831216E-03  5.871013E-18
    2.650001E-01  G     3.848278E-19  1.128651E-17  7.785291E-03   .0      -3.828911E-03  6.419444E-18
    2.700001E-01  G     3.796167E-19  1.207232E-17  7.690622E-03   .0      -3.778875E-03  7.287524E-18
                                                        .
    5.949959E-01  G     1.986670E-19 -1.304284E-17  4.027668E-03   .0      -2.010369E-03 -4.506177E-18
    5.999959E-01  G     2.106813E-19 -1.285776E-17  4.244778E-03   .0      -2.079488E-03 -4.219749E-18
 ---------------------------------------------------------------------------------------------------
                       *****    PARTIAL OUTPUT FROM bd08bar6.f06    *****
                       X Y - O U T P U T   S U M M A R Y   ( R E S P O N S E )

 SUBCASE  CURVE  FRAME              XMIN-FRAME/   XMAX-FRAME/   YMIN-FRAME/    X FOR      YMAX-FRAME/
   ID     TYPE    NO.    CURVE ID.   ALL DATA     ALL DATA      ALL DATA      YMIN        ALL DATA
    1   DISP     1      11( 5)   0.000000E+00  5.999959E-01  0.000000E+00  0.000000E+00  7.886210E-03   2.
                                0.000000E+00  5.999959E-01  0.000000E+00  0.000000E+00  7.886210E-03   2.
    1   ACCE     2      10( 5)   0.000000E+00  5.999959E-01 -8.511892E-01  2.650001E-01  9.618800E-01   5.
                                0.000000E+00  5.999959E-01 -8.511892E-01  2.650001E-01  9.618800E-01   5.
```

Figure 8-11. Partial Output from Transient Analysis with Unit Step Function Input

```
     *****     PARTIAL OUTPUT FROM bd08bar7.f06    *****
```

```
    POINT-ID =      11
                                        D I S P L A C E M E N T    V E C T O R

      TIME         TYPE         T1              T2              T3              R1              R2              R3
    .0             G         .0              .0              .0              .0              .0              .0
    5.000000E-03   G      8.975580E-22    1.606407E-21    1.913647E-05    .0           -2.984776E-05    3.499062E-20
    1.000000E-02   G      3.317352E-21    8.430443E-21    7.628838E-05    .0           -1.102005E-04    1.252464E-19
                                                           .
    1.650002E-01   G      2.077055E-19    2.232167E-18    4.099007E-03    .0           -1.833254E-03    1.845365E-18
    1.700002E-01   G      2.104969E-19    2.275425E-18    4.180374E-03    .0           -1.908738E-03    1.502202E-18
    1.750002E-01   G      2.131340E-19    2.401124E-18    4.255632E-03    .0           -2.000815E-03    1.338691E-18
    1.800002E-01   G      2.146708E-19    2.724639E-18    4.308840E-03    .0           -2.072395E-03    1.631772E-18
    1.850002E-01   G      2.144943E-19    3.204978E-18    4.328179E-03    .0           -2.097317E-03    2.240218E-18
    1.900002E-01   G      2.129652E-19    3.684523E-18    4.313660E-03    .0           -2.081293E-03    2.749993E-18
    1.950002E-01   G      2.109444E-19    4.053196E-18    4.272623E-03    .0           -2.048310E-03    2.916553E-18
                                                           .
    5.949959E-01   G      1.315062E-19   -1.196586E-17    2.607416E-03    .0           -1.203858E-03   -4.331415E-18
    5.999959E-01   G      1.410901E-19   -1.111419E-17    2.775622E-03    .0           -1.249621E-03   -3.777846E-18

                   X Y - O U T P U T    S U M M A R Y    ( R E S P O N S E )

 SUBCASE   CURVE FRAME                       XMIN-FRAME/     XMAX-FRAME/     YMIN-FRAME/      X FOR        YMAX-FRAME/
    ID      TYPE   NO.      CURVE ID.          ALL DATA        ALL DATA        ALL DATA        YMIN          ALL DATA
    1   DISP      1      11( 5)           0.000000E+00    6.000000E-01   -4.039956E-03    4.199981E-01    4.328179E-03    1.8
                                          0.000000E+00    5.999959E-01   -4.039956E-03    4.199981E-01    4.328179E-03    1.8
```

Figure 8-12. Partial Output from Transient Analysis with a Triangular Pulse

```
                 *****    PARTIAL OUTPUT FROM bd08bar8.f06    *****
    POINT-ID =      11
                                 C O M P L E X    D I S P L A C E M E N T    V E C T O R
                                              (MAGNITUDE/PHASE)

    FREQUENCY     TYPE         T1              T2              T3              R1              R2              R3
    .0             G      2.016639E-20    4.838324E-19    4.085192E-04    .0            2.024146E-04    3.208819E-19
                          .0              .0              .0              .0            180.0000        .0
    5.000000E-01   G      2.136733E-20    5.467976E-19    4.323836E-04    .0            2.134119E-04    3.549737E-19
                          359.4255        359.0918        359.4344        .0            179.4676        359.2066
                                                           .
    2.073000E+00   G      4.228285E-19    1.831657E-16    8.402211E-03    .0            3.874130E-03    8.302084E-17
                          303.5255        169.4034        303.5720        .0            123.7533        170.2924
    2.087000E+00   G      4.789151E-19    1.747242E-16    9.514135E-03    .0            4.382166E-03    7.877973E-17
                          288.0493        145.7774        288.0963        .0            108.2796        146.6783
    2.100632E+00   G      5.003335E-19    1.561973E-16    9.936970E-03    .0            4.572148E-03    7.006467E-17
                          270.0177        122.0074        270.0652        .0            90.2504         122.9200
    2.224000E+00   G      1.560747E-19    1.943312E-17    3.091950E-03    .0            1.408677E-03    8.308275E-18
                          199.3224        32.8670         199.3742        .0            19.5776         33.8900
    2.347000E+00   G      7.916779E-20    5.971262E-18    1.564160E-03    .0            7.050781E-04    2.427647E-18
                          190.2228        19.2229         190.2792        .0            10.5024         20.3631
                                                           .
    9.950000E+01   G      .0              .0              5.272466E-07    .0            8.710860E-07    1.024487E-21
                          .0              .0              180.4391        .0            .6610           180.7005
    1.000000E+02   G      .0              .0              5.216914E-07    .0            8.615909E-07    1.013485E-21
                          .0              .0              180.4361        .0            .6566           180.6966

                   X Y - O U T P U T    S U M M A R Y    ( R E S P O N S E )

 SUBCASE   CURVE FRAME                       XMIN-FRAME/     XMAX-FRAME/     YMIN-FRAME/      X FOR        YMAX-FRAME/
    ID      TYPE   NO.      CURVE ID.          ALL DATA        ALL DATA        ALL DATA        YMIN          ALL DATA
    1   DISP      1      11( 5,--)       1.000000E-01    1.000000E+02    5.216914E-07    1.000000E+02    9.936970E-03    2.1
                                         0.000000E+00    1.000000E+02    5.216914E-07    1.000000E+02    9.936970E-03    2.1
    1   DISP      1      11(--, 11)      1.000000E-01    1.000000E+02    1.804361E+02    1.000000E+02    3.594344E+02    5.0
                                         0.000000E+00    1.000000E+02    0.000000E+00    0.000000E+00    3.594344E+02    5.0
```

Figure 8-13. Partial Output from Frequency Response Analysis

```
                 *****    PARTIAL OUTPUT FROM bd08bar9.f06    *****
 P R O J E C T    V E R S I O N    D I R E C T O R Y    P R I N T
    PROJECT_ID                              ASSIGNED INT. VALUE    VERSION_ID     CREATION TIME
    ----------                              -------------------    ----------    -------------
    " B L A N K "                                    1             **      1      9/ 2/93   13:22.20
                                                                   **      2      9/ 2/93   13:23. 0
                                                                           3      9/ 2/93   13:23.25
                                                                   **      4      9/ 2/93   13:23.59
                                                                   **      5      9/ 2/93   13:24.28
                                                                   **      6      9/ 2/93   13:24.57
                                                                           7      9/ 2/93   13:25.26
```

Figure 8-14. Partial Output from a Database Directory Run

Chapter

9   *Plotted Output*

- *Overview*

- *Structure Plotting*

- *X-Y Plotting*

9.1   Overview

Plotted output is important in verifying your model and understanding its results. Plots show information in a format that is much easier to interpret than printed output. Plots are especially important for dynamic analysis because the analysis can produce voluminous output. For example, consider a transient response analysis for which there are 1000 output time steps, 100 grid points of interest, and 10 elements of interest. Printed output is too large to interpret efficiently and effectively, and it does not easily show the time variation. Plotted output overcomes these problems.

There are two kinds of plotted output: structure plots and X-Y plots. Structure plots can depict the entire structure or a portion of it. Structure plots are useful for verifying proper geometry and connectivity. They also can be used to show the deformed shape or stress contours at a specified time or frequency. X-Y plots, on the other hand, show how a single-response quantity, such as a grid point displacement or element stress, varies across a portion or all of the time or frequency range.

There are numerous commercial and in-house plotting programs that interface to NX Nastran for structure and/or X-Y plotting.

The commercial programs are similar because they operate interactively. The NX Nastran plot capabilities, on the other hand, are performed as a batch operation, which means that you predefine your plots when you make your NX Nastran run.

This chapter briefly describes the kinds of plots available in NX Nastran. Detailed information is located in the *NX Nastran Reference Manual*.

9.2   Structure Plotting

Structure plotting is performed to verify the model's geometry and element connectivity prior to performing a dynamic analysis.

After (or during) the analysis, structure plotting is performed to view deformed shapes and contours. For dynamic response, deformed shape and contour plots can be made for normal modes analysis (for which there is a plot, or set of plots, per mode), frequency response analysis (for which there is a plot, or set of plots, per output frequency), and transient response analysis (for which there is a plot, or set of plots, per output time).

Structure plot commands are described in the *NX Nastran User's Guide*. In the NX Nastran input file, structure plotting commands are listed in the OUTPUT(PLOT) section, which immediately precedes the Bulk Data Section. The structure plotting commands define the set of elements to be plotted (SET), the viewing axes (AXES), the viewing angles (VIEW) as well as the plot type and parameters (PLOT). Optionally, the scale of the plotted deformation (MAXIMUM DEFORMATION) can be specified; if not specified, the plotted deformation is scaled such that the maximum deformation is 5% of the maximum dimension of the structure.

Figure 9-1 shows the structure plotting commands applied to a normal modes analysis of the bracket model. Figure 9-2 shows the resulting structure plots. The first plot shows the undeformed shape, and the next two plots show the undeformed shape overlaid on the deformed shapes for modes 1 and 2. The default is chosen such that the maximum plotted deformation is 5% of the maximum dimension of the bracket; the actual maximum deformation is printed at the top of the plot. The plots shown in this chapter were converted to PostScript format for printing on a PostScript-compatible printer.

```
   $  PLOT COMMANDS FOR BRACKET NORMAL MODES ANALYSIS
   ... Executive, Case Control ...
```

```
$
$  OUTPUT REQUESTS
DISPLACEMENT(PLOT) = ALL
$
$  STRUCTURE PLOTS
OUTPUT(PLOT)
CSCALE = 3.0
SET 333 = ALL
AXES MX,MY,Z
VIEW 20.,20.,20.
FIND SCALE, ORIGIN 5, SET 333
$  PLOT UNDEFORMED SHAPE
PLOT SET 333, ORIGIN 5
$  PLOT DEFORMED, UNDEFORMED SHAPES
PLOT MODAL DEFORMATION 0,1
PLOT MODAL DEFORMATION 0,2
$
BEGIN BULK
$
... Bulk Data ...
$
ENDDATA
```

Figure 9-1. Normal Modes Structure Plot Commands for the Bracket Model

Undeformed Shape

Max. Def. = 3.82

Mode 1

Max. Def. = 5.06

Mode 2
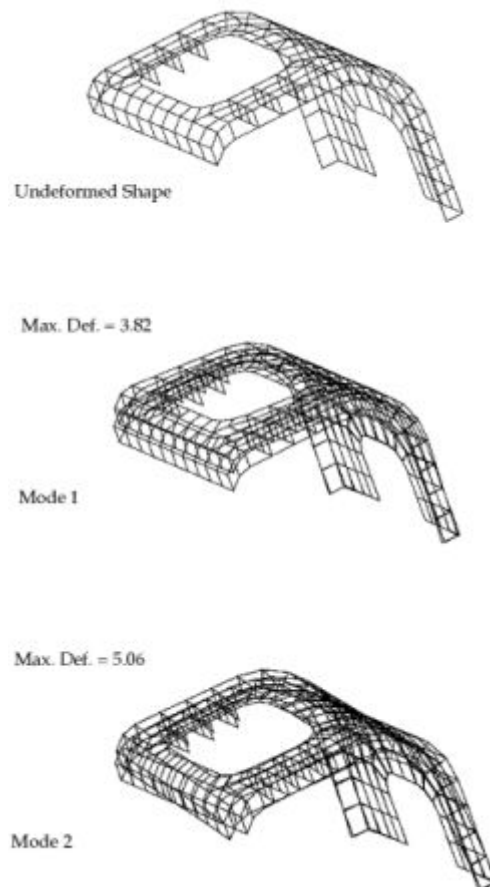
Figure 9-2. Normal Modes Structure Plots for the Bracket Model

Figure 9-3 shows the structure plotting commands applied to a modal frequency response analysis of the cantilever beam model. The displacements are computed in magnitude/phase

form. PARAM,DDRMM,-1 and PARAM,CURVPLOT,1 are required in the Bulk Data to create structure plots at specified frequencies. (Note that PARAM,DDRMM,-1 generally increases the amount of computer time and is not recommended unless otherwise required.) Figure 9-4 shows the resulting plots. The first plot shows the undeformed shape, and the next plot shows the magnitude of response at 2.05 Hz, which is overlaid on the undeformed shape. The default is chosen such that the maximum plotted deformation is 5% of the length of the bar; the actual maximum deformation is printed at the top of the plot.

```
$   PLOT COMMANDS FOR BAR MODAL FREQUENCY RESPONSE
$   --- MAGNITUDE, PHASE ---
... Executive, Case Control ...
$
$   OUTPUT REQUESTS
$   MAGNITUDE, PHASE
DISPLACEMENT(PHASE,PLOT) = ALL
$
$   STRUCTURE PLOTS
OUTPUT(PLOT)
$   DEFINE ELEMENTS IN PLOT SET
SET 333 = ALL
$   PLOT AXES:  R=-Z, S=X, T=Y
AXES MZ,X,Y
VIEW 0.,0.,0.
FIND SCALE, ORIGIN 5, SET 333
$   PLOT UNDEFORMED SHAPE
PLOT SET 333, ORIGIN 5
$   PLOT DEFORMED SHAPE AT 2.05 HZ
PLOT FREQUENCY DEFORMATION 0 RANGE 2.05,2.051 MAGNITUDE SET 333
$
BEGIN BULK
$
$ REQUIRED FOR "FREQUENCY FROZEN" STRUCTURE PLOTS
PARAM,DDRMM,-1
PARAM,CURVPLOT,1
$
... rest of Bulk Data ...
$
ENDDATA
```

**Figure 9-3. Frequency Response Structure Plot Commands for the Bar Model –
Magnitude/Phase**
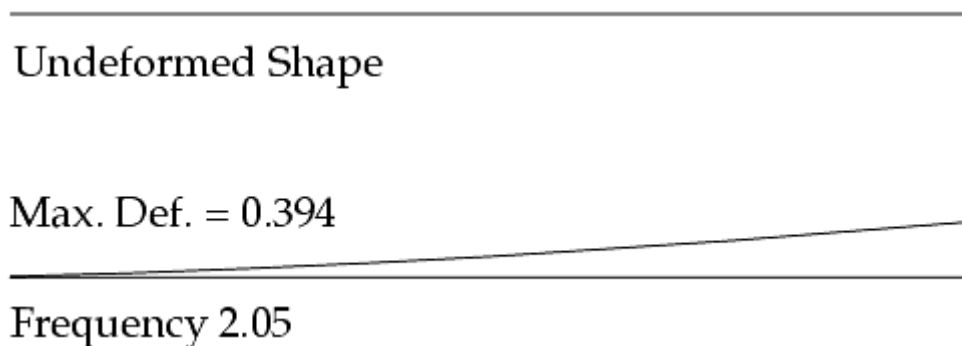
Figure 9-4. Frequency Response Structure Plots for the Bar Model – Magnitude/Phase

The same plots are regenerated except that now the displacements are in real/imaginary format (the default). Figure 9-5 shows the plot commands. Note that the imaginary component is selected by PHASE LAG -90. Figure 9-6 shows the resulting plots. The default is chosen such

that the maximum plotted deformation is 5% of the length of the bar; the actual maximum deformation is printed at the top of the plot.

```
$  PLOT COMMANDS FOR BAR MODAL FREQUENCY RESPONSE
$  --- REAL, IMAGINARY ---
... Executive, Case Control ...
$
$  OUTPUT REQUESTS
$  REAL, IMAGINARY
DISPLACEMENT(PLOT) = ALL
$
$  STRUCTURE PLOTS
OUTPUT(PLOT)
$  DEFINE ELEMENTS IN PLOT SET
CSCALE = 1.8
SET 333 = ALL
$  PLOT AXES:  R=-Z, S=X, T=Y
AXES MZ,X,Y
VIEW 0.,0.,0.
FIND SCALE, ORIGIN 5, SET 333
$  PLOT UNDEFORMED SHAPE
PLOT SET 333, ORIGIN 5
$  PLOT DEFORMED SHAPE--REAL
PLOT FREQ DEFORM 0 RANGE 2.05,2.051 SET 333
$  PLOT DEFORMED SHAPE--IMAGINARY
PLOT FREQ DEFORM 0 RANGE 2.05,2.051 PHASE LAG -90. SET 333
$
BEGIN BULK
$
$ REQUIRED FOR "FREQUENCY FROZEN" STRUCTURE PLOTS
PARAM,DDRMM,-1
PARAM,CURVPLOT,1
$
... rest of Bulk Data ...
$
ENDDATA
```

Figure 9-5. Frequency Response Structure Plot Commands for the Bar Model - Real/Imaginary
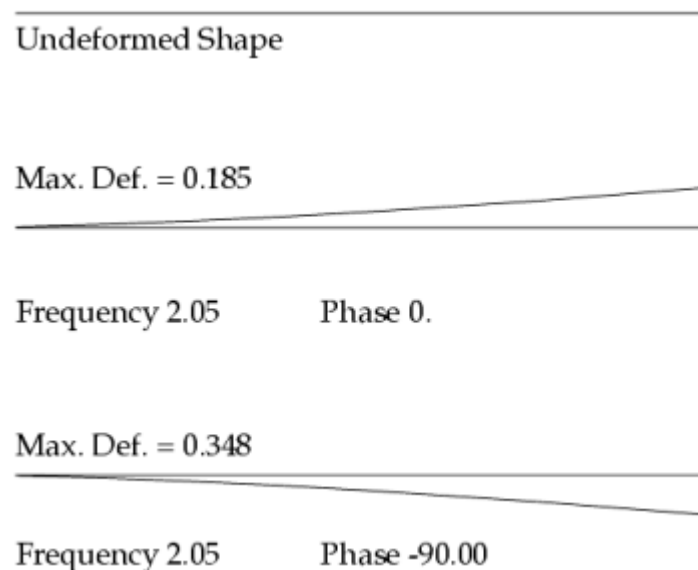
Figure 9-6. Frequency Response Structure Plots for the Bar Model – Real/Imaginary

Figure 9-7 shows the structure plotting commands applied to a modal transient response analysis of the cantilever beam model. PARAM,DDRMM,-1 is required in the Bulk Data in order to create structure plots at various times and/or frequencies. (Note that PARAM,DDRMM,-1 generally increases the amount of computer time and is not recommended unless otherwise required.) Figure 9-8 shows the resulting plots. The plots show the displacements at the following times: 0.25, 0.50, and 0.75 seconds, which are overlaid on the undeformed shape. The default is chosen such that the maximum plotted deformation is 5% of the length of the bar; the actual maximum deformation is printed at the top of the plot. For this case it may be better to specify a maximum deformation so that the plots will show relative amplitudes.

```
$  PLOT COMMANDS FOR BAR MODAL TRANSIENT RESPONSE
... Executive, Case Control ...
$
$  OUTPUT REQUESTS
DISPLACEMENT(PLOT) = ALL
$
$  STRUCTURE PLOTS
OUTPUT(PLOT)
$  DEFINE ELEMENTS IN PLOT SET
SET 333 = ALL
$  PLOT AXES:  R=-Z, S=X, T=Y
AXES MZ,X,Y
VIEW O.,O.,O.
FIND SCALE, ORIGIN 5, SET 333
$  PLOT DEFORMED SHAPE AT TIMES 0.25,0.5,0.75
PLOT TRANSIENT DEFORMATION 0 RANGE 0.25,0.251 SET 333
PLOT TRANSIENT DEFORMATION 0 RANGE 0.50,0.501 SET 333
PLOT TRANSIENT DEFORMATION 0 RANGE 0.75,0.751 SET 333
$
BEGIN BULK
$
$  REQUIRED FOR "TIME FROZEN" STRUCTURE PLOTS
PARAM,DDRMM,-1
$
... rest of Bulk Data ...
$
ENDDATA
```

Figure 9-7. Transient Response Structure Plot Commands for the Bar Model

Max. Def. = 0.040

Time 0.25

Max. Def. = 0.069

Time 0.50

Max. Def. = 0.057

Time 0.75
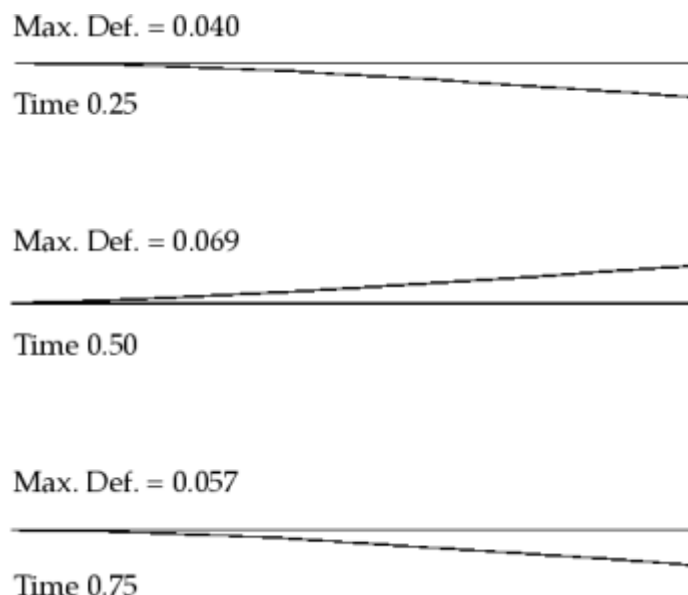
Figure 9-8. Transient Response Structure Plots for the Bar Model

Although structure plotting is best performed in an interactive environment outside of NX Nastran, the batch structure plotting capability in NX Nastran is nevertheless a useful tool for model verification and results processing. The batch plotting capability can save time and effort when many plots are required for a model that is run repeatedly.

9.3   X-Y Plotting

X-Y plots are used to display frequency and transient response results where the x-axis is frequency or time and the y-axis is any output quantity. Unlike structure plotting, which is often performed in an interactive environment, X-Y plotting is ideal for the NX Nastran batch environment due to the large volume of data.

X-Y plot commands are contained in the OUTPUT(XYPLOT) section that immediately precedes the Bulk Data Section. You define the titles (XTITLE and YTITLE) and plots (XYPLOT). You can specify the plots to be generated in log format (XLOG and YLOG), and you can specify different line styles. You can also specify that the plots are to be made in pairs (with a top and a bottom plot), which is particularly useful for frequency response results when you want to display magnitude/phase or real/imaginary pairs. Details about X-Y plotting and its commands are located in the *NX Nastran User's Guide*.

Element force and stress component numbers are also described in the manual. The X-Y plot commands use numbers to identify a single component of grid point and element data. Note that frequency response requires the use of complex force and stress components.

Once a good set of X-Y plot commands is established, it is wise to use this set repeatedly. The examples that follow provide a good starting point.

Figure 9-9 shows X-Y plot commands for a modal frequency response analysis of the cantilever beam model. Plots are made in pairs in magnitude/phase format. The "t" as the second letter in XTGRID, YTGRID, YTLOG, and YTTITLE corresponds to the top plot of each pair; the letter "b" corresponds to the bottom plot (XBGRID, YBGRID, etc.) Plots are made for the applied loads (OLOAD) at grid points 6 and 11, displacements (DISP) of grid points 6 and 11, bending moment (ELFORCE) at end A in plane 1 for element 6, and modal displacements (SDISP) for modes 1 and 2. For CBAR elements, force component 2 is the bending moment component at end A in plane 1

for real or magnitude output, and force component 10 is the similar component for imaginary or phase output. Figure 9-10 shows the resulting plots. The plots in this chapter were converted to PostScript format for printing on a PostScript-compatible printer.

The X-Y plotter makes reasonable choices of upper and lower bounds for the axes for both the x- and y-axes on most plots. If it does not, the bounds can be fixed with the XMIN, XMAX, YMIN, YMAX, and their variations for half-frame curves. Instances where setting bounds explicitly results in better plots include the following situations:

- If you expect a variable to be constant or vary only slightly but want to plot it to confirm that it indeed does not vary, set the YMIN and YMAX to include the expected value, but separate them by at least 10 percent of their average value.

  For this case the automatic bound selection chooses bounds very close to each other to make the data fill up the plot. This selection causes the bounds to be nearly equal and magnifies the scale of the plot orders of magnitude larger than other plots made of varying functions. The response appears to be erratic when, in fact, it is smooth within engineering criteria. The extreme cases occur when the function varies only in its last digit. Then the function appears to be oscillating between the upper and lower limits or it can even cause a fatal error due to numerical overflow when it attempts to divide numbers by the difference YMAX-YMIN.

- The automatic bound selector tends to round up the bounds to integer multiples of 100. When plotting phase angles, bounds that cause grid lines at 90 degrees are more readable. For example, for a plot that traverses the range of 0 to 360 degrees, the usual selected bounds are 0 to 400 degrees. If you prefer to have grid lines drawn at integer multiples of 90 degrees, set YMIN to 0.0 and YMAX at 360.0 degrees.

- When plotting log plots, any bounds you input may be rounded up or down to a value that the plotter considers more reasonable. In general, changing bounds on log plots sometimes requires experimentation before a reasonable set can be found.

- Producing good quality plots is an interactive process whether the plot is produced by an interactive or batch plotter. The restart feature discussed in "Restarts In Dynamic Analysis" can reduce the computer costs for this iteration since restarts performed to change only plot requests are made efficiently.

```
$  X-Y PLOT COMMANDS FOR BAR MODAL FREQUENCY RESPONSE
$  --- MAGNITUDE, PHASE ---
...Executive, Case Control ...
$
$  APPLIED LOAD OUTPUT
SET 15 = 6,11
OLOAD(PHASE,PLOT) = 15
$
$  PHYSICAL OUTPUT REQUEST
SET 11 = 6,11
DISPLACEMENT(PHASE,PLOT) = 11
$
$  MODAL SOLUTION SET OUTPUT
SET 12 = 1,2
SDISP(PHASE,PLOT) = 12
$
$  ELEMENT FORCE OUTPUT
SET 13 = 6
ELFORCE(PHASE,PLOT) = 13
$
OUTPUT(XYPLOT)
XTGRID = YES
YTGRID = YES
XBGRID = YES
```

```
YBGRID = YES
$
$  PLOT RESULTS
XTITLE = FREQUENCY
$
YTLOG = YES
YTTITLE = DISPL. MAG. 6
YBTITLE = DISPL. PHASE 6
XYPLOT DISP /6(T2RM,T2IP)
YTTITLE = DISPL. MAG. 11
YBTITLE = DISPL. PHASE 11
XYPLOT DISP /11(T2RM,T2IP)
$
YTTITLE = SDISP. MAG. MODE 1
YBTITLE = SDISP. PHASE MODE 1
$ XYPLOT SDISP /mode(T1)
XYPLOT  SDISP  /1(T1RM,T1IP)
YTTITLE = SDISP. MAG. MODE 2
YBTITLE = SDISP. PHASE MODE 2
XYPLOT  SDISP   /2(T1RM,T1IP)
$
YTTITLE = BEND. MOMENT A1 EL. 6 MAG.
YBTITLE = BEND. MOMENT A1 EL. 6 PHASE
XYPLOT ELFORCE /6(2,10)
$
YTLOG = NO
YBMAX = 90.0
YBMIN = 0.0
CURVELINESYMBOL = -2
YTMAX = 4.0
YTMIN = 0.0
YTTITLE = LOAD MAG. 6
YBTITLE = LOAD PHASE 6
XYPLOT OLOAD /6(T2RM,T2IP)

YTMAX = 8.0
YTMIN = 0.0
YTTITLE = LOAD MAG. 11
YBTITLE = LOAD PHASE 11
XYPLOT OLOAD /11(T2RM,T2IP)
$
BEGIN BULK
$
... Bulk Data ...
$
ENDDATA
```

Figure 9-9. X-Y Plot Commands for the Bar Frequency Response Analysis
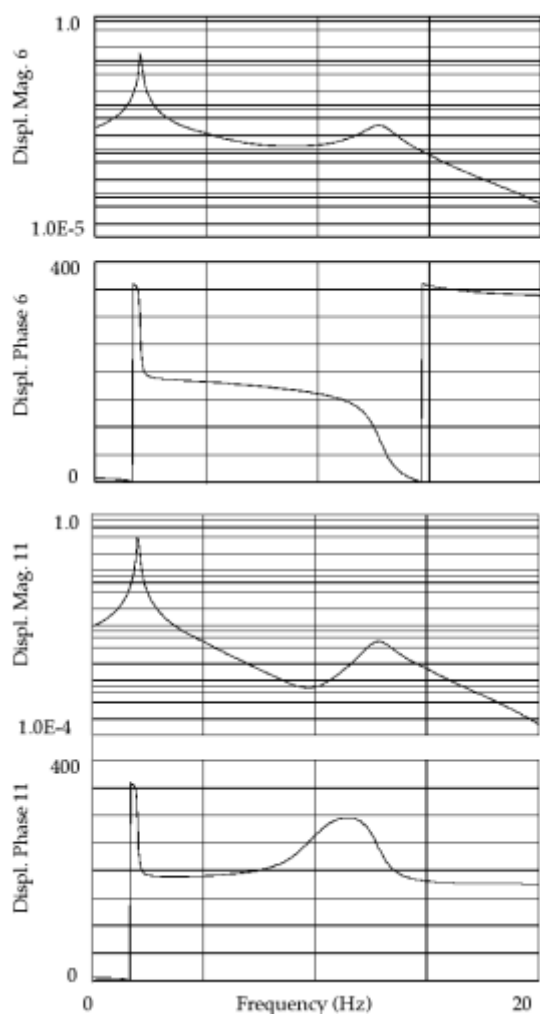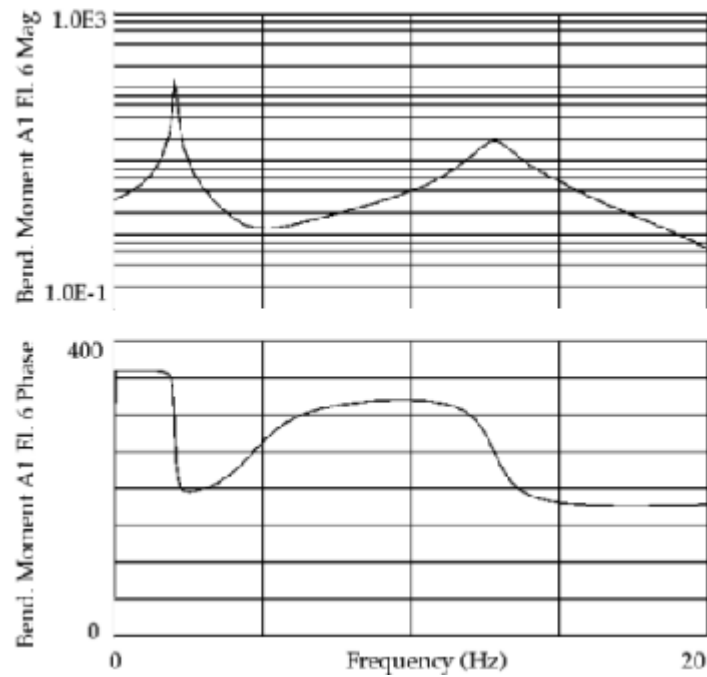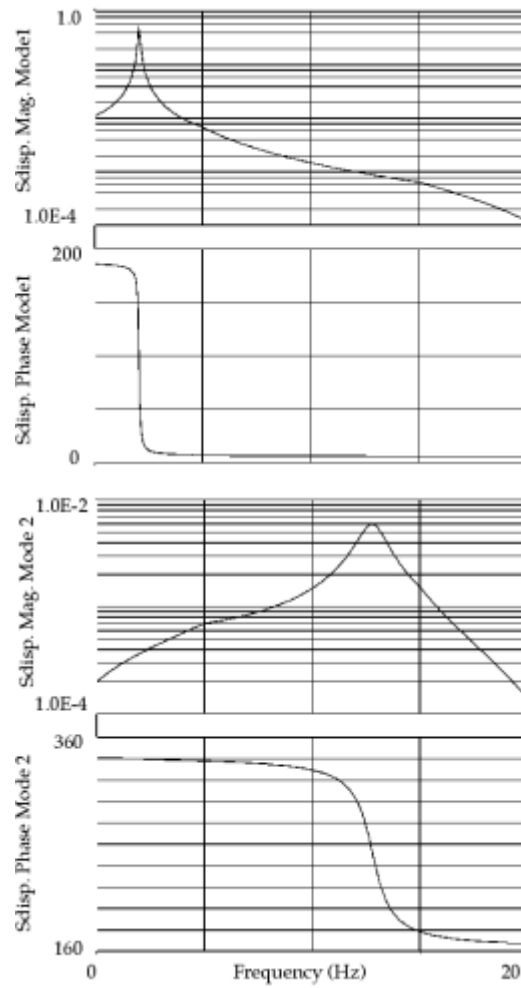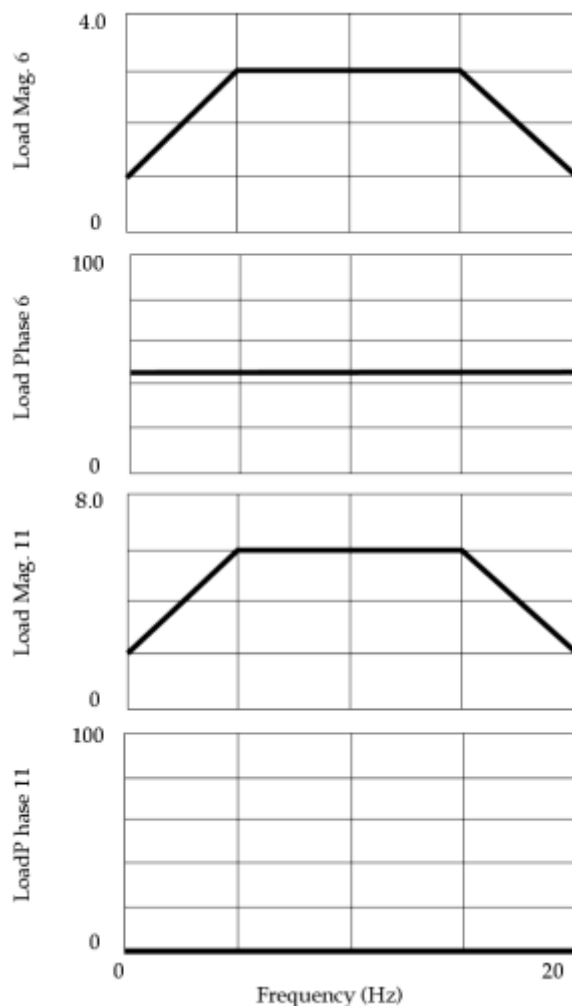
Figure 9-10. X-Y Plots for the Bar Frequency Response Analysis

Figure 9-11 shows X-Y plot commands for a modal transient response analysis of the cantilever beam model. Plots are made for the applied loads (OLOAD) at grid points 6 and 11, displacements (DISP) of grid points 6 and 11, accelerations (ACCE) for grid points 6 and 11, bending moment (ELFORCE) at end A in plane 1 for element 6, and modal displacements (SDISP) for modes 1 and 2. Figures 9-12 through 9-16 show the resulting plots.

```
$   X-Y PLOT COMMANDS FOR BAR MODAL FREQUENCY RESPONSE
$   --- MAGNITUDE, PHASE ---
...Executive, Case Control ...
$
$   APPLIED LOAD OUTPUT
SET 15 = 6,11
OLOAD(PHASE,PLOT) = 15
$
$   PHYSICAL OUTPUT REQUEST
SET 11 = 6,11
DISPLACEMENT(PHASE,PLOT) = 11
$
$   MODAL SOLUTION SET OUTPUT
SET 12 = 1,2
SDISP(PHASE,PLOT) = 12
$
$   ELEMENT FORCE OUTPUT
SET 13 = 6
ELFORCE(PHASE,PLOT) = 13
$
OUTPUT(XYPLOT)
XTGRID = YES
YTGRID = YES
XBGRID = YES
```

```
YBGRID = YES
$
$  PLOT RESULTS
XTITLE = FREQUENCY
$
YTLOG = YES
YTTITLE = DISPL. MAG. 6
YBTITLE = DISPL. PHASE 6
XYPLOT DISP /6(T2RM,T2IP)
YTTITLE = DISPL. MAG. 11
YBTITLE = DISPL. PHASE 11
XYPLOT DISP /11(T2RM,T2IP)
$
YTTITLE = SDISP. MAG. MODE 1
YBTITLE = SDISP. PHASE MODE 1
$ XYPLOT SDISP /mode(T1)
XYPLOT  SDISP  /1(T1RM,T1IP)
YTTITLE = SDISP. MAG. MODE 2
YBTITLE = SDISP. PHASE MODE 2
XYPLOT  SDISP  /2(T1RM,T1IP)
$
YTTITLE = BEND. MOMENT A1 EL. 6 MAG.
YBTITLE = BEND. MOMENT A1 EL. 6 PHASE
XYPLOT ELFORCE /6(2,10)
$
YTLOG = NO
YBMAX = 90.0
YBMIN = 0.0
CURVELINESYMBOL = -2
YTMAX = 4.0
YTMIN = 0.0
YTTITLE = LOAD MAG. 6
YBTITLE = LOAD PHASE 6
XYPLOT OLOAD /6(T2RM,T2IP)

YTMAX = 8.0
YTMIN = 0.0
YTTITLE = LOAD MAG. 11
YBTITLE = LOAD PHASE 11
XYPLOT OLOAD /11(T2RM,T2IP)
$
BEGIN BULK
$
... Bulk Data ...
$
ENDDATA
```

Figure 9-11. X-Y Plot Commands for the Bar Transient Response Analysis
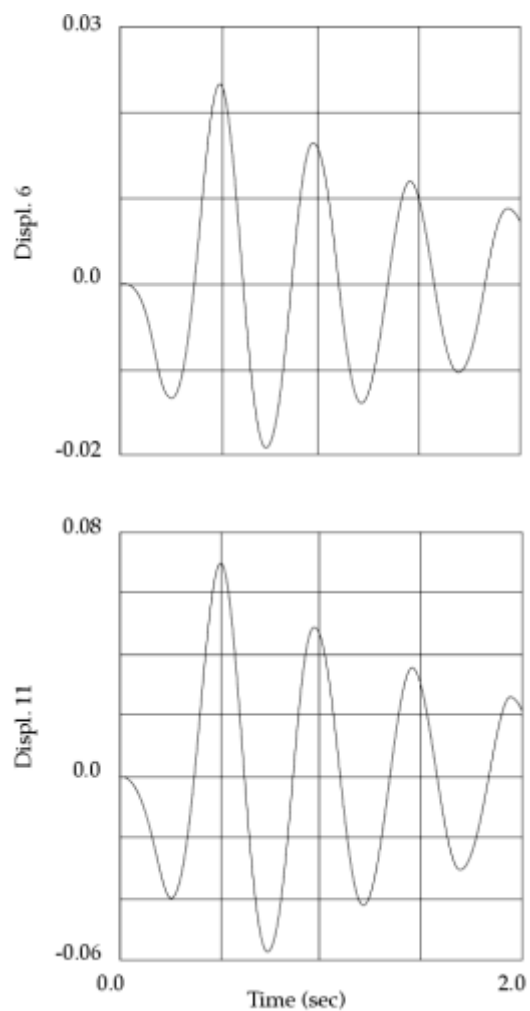
Figure 9-12. X-Y Plots for the Bar Transient Response Analysis
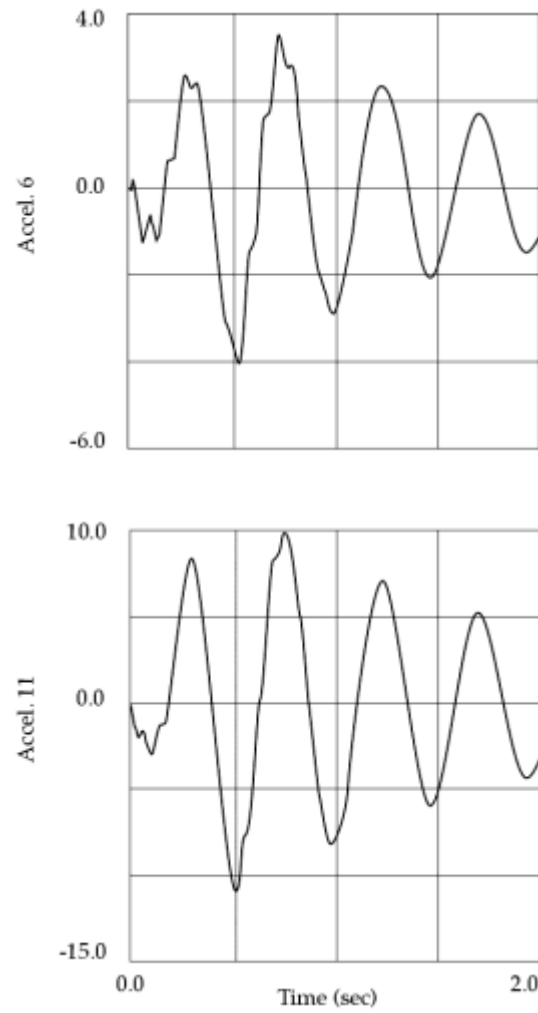
Figure 9-13. X-Y Plots for the Bar Transient Response Analysis
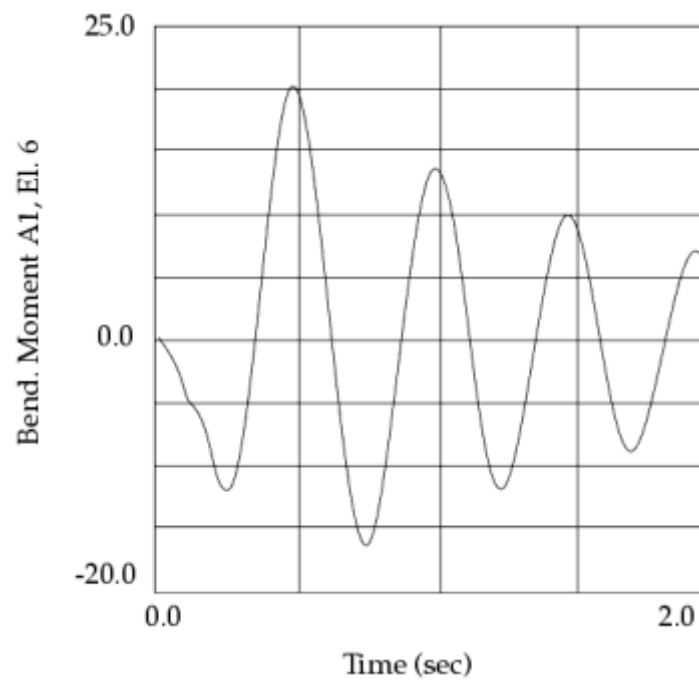
Figure 9-14. X-Y Plots for the Bar Transient Response Analysis
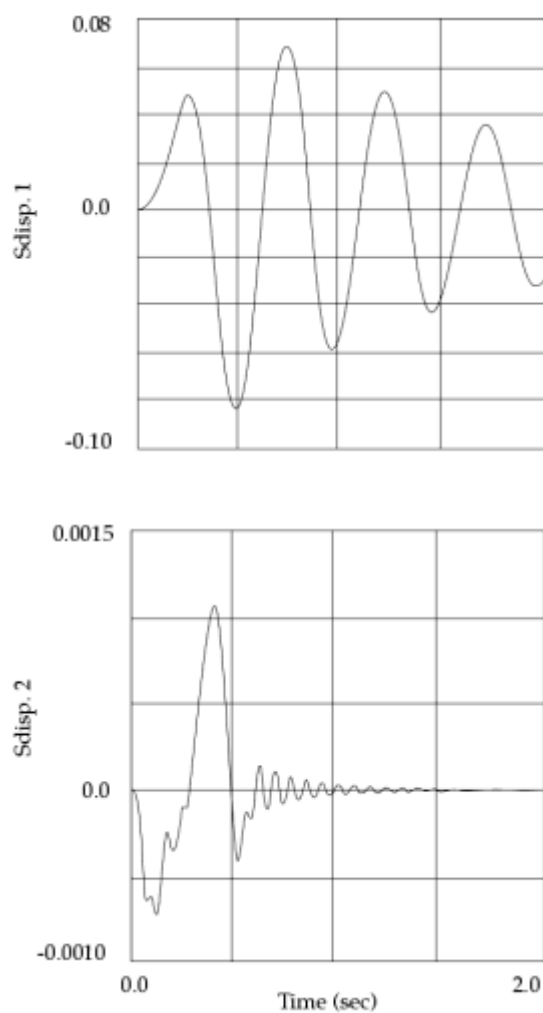
Figure 9-15. X-Y Plots for the Bar Transient Response Analysis
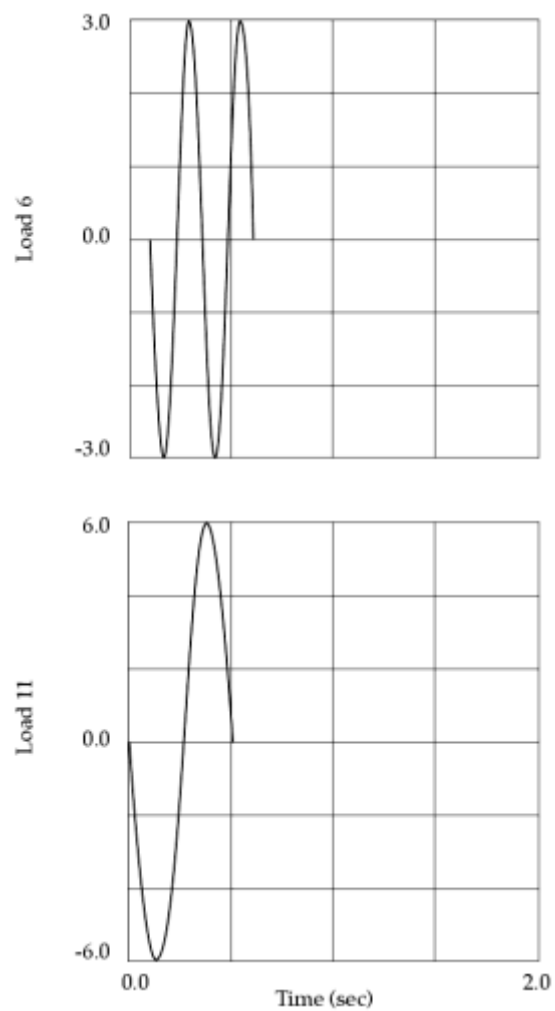
Figure 9-16. X-Y Plots for the Bar Transient Response Analysis

Chapter

10   Guidelines for Effective Dynamic Analysis

- *Overview*

- *Overall Analysis Strategy*

- *Units*

- *Mass*

- *Damping*

- *Boundary Conditions*

- *Loads*

- *Meshing*

- *Eigenvalue Analysis*

- *Frequency Response Analysis*

- *Transient Response Analysis*

- *Results Interpretation and Verification*

- *Computer Resource Requirements*

10.1 Overview

Dynamic analysis is more complicated than static analysis because of more input (mass, damping, and time- and frequency-varying loads) and more output (time- and frequency-varying results). Results from static analysis are usually easier to interpret, and there are numerous textbook solutions for static analysis that make it relatively easy to verify certain static analyses. Nevertheless, the guidelines in this chapter help you to perform dynamic analysis in a manner that will give you the same level of confidence in the dynamic results that you would have with static results.

This chapter covers the following topics:

- Overall analysis strategy

- Units

- Mass

- Damping

- Boundary conditions

- Loads

- Meshing

- Eigenvalue analysis

- Frequency response analysis

- Transient response analysis

- Results interpretation

- Computer resource estimation

10.2 Overall Analysis Strategy

Part of any analysis strategy, whether it be for dynamic analysis or static analysis, is to gain confidence with the modeling procedures first. The best way to accomplish this is to run small, simple models, preferably models that have textbook solutions. The references (see "References and Bibliography") provide numerous textbook solutions. Start with a simple model first and then gradually add complexity, verifying the results at each stage. Follow the steps outlined below and in Figure 10-1.

Once you have confidence in a small model and are ready to analyze your actual model, again do the analysis in steps. The following is a suggested order for performing dynamic analysis on any structure:

1. Create the initial model only; do not apply any loads. Verify the model's connectivity, element and material properties, and boundary conditions. Make sure that mass is specified for this model.

2. Perform a static analysis (SOL 101) first in order to verify proper load paths and overall model integrity. (Note that you have to constrain the structure for static analysis even if you were not planning to do so for dynamic analysis.) For a three-dimensional model, you

should run three load cases, each with a 1g gravity load applied in a different direction. Compute displacements and SPC forces, and verify the results. Check for unusually large grid point displacements and unreasonable SPC forces. Using a graphical postprocessor can aid you at this step.

Next, apply static loads that have the same spatial distribution that your subsequent dynamic loads have. Verify the results for reasonableness. Do not go to dynamic analysis until you are satisfied with the results from your static analysis.

It is recommended at this stage that the model contain PARAM,GRDPNT,n (where n is a reference grid point or 0, the origin of the basic coordinate system). Verify the results from the grid point weight generator in order to ensure that the model's rigid-body mass and inertia look reasonable. This step, in conjunction with the static analysis results, helps to ensure that the proper mass units are specified.

3. Perform an eigenvalue analysis (SOL 103) next. Compute only a few modes first, verify their frequencies, and view their mode shapes for reasonableness. If your graphical postprocessor can animate the mode shapes, do so because that helps you to visualize them. Things to check at this step are local mode shapes, in which one or a few grid points are moving a very large amount relative to the rest of the model (this can indicate poor stiffness modeling in that region), and unwanted rigid-body modes (which can arise due to improper specification of the boundary conditions or a mechanism).

    Once you are satisfied with these results, perform the full eigenvalue analysis (for as many modes as you need).

4. If you have frequency-dependent loads, perform frequency response analysis (SOL 108 or SOL 111) using the dynamic load spatial distribution. If your structure is constrained, then apply the dynamic load at only one frequency, which should be 0.0 Hz. Compare the 0.0 Hz displacement results to the static analysis displacement results. The results should be the same if direct frequency response (without structural damping) is used. If the results are not equal, then there is probably an error in the specification of the dynamic load, and you should check the LSEQ and DAREA entries. If modal frequency response (without structural damping) is used, then the 0.0 Hz results should be close to the static results; the difference is due to modal truncation.

    Next apply the load across the entire frequency range of interest. If you are running modal frequency response, then make sure that you have enough modes to ensure accurate results for even the highest forcing frequency. Also be sure to have a small enough Δf in order to accurately capture the peak response. Verify these results for reasonableness (it may be easier to look at magnitude and phase results instead of real and imaginary results, which are the default values).

    If your ultimate goal is a transient response analysis for which damping is to be neglected, then the frequency response analysis can also omit damping. However, if damping is to be included, then use the correct damping in your frequency response analysis. The proper specification of damping can be verified by looking at the half-power bandwidth.

    Plots are important at this stage to assist in results interpretation. X-Y plots are necessary in order to see the variation in response versus frequency. Deformed structure plots at a frequency near a resonant frequency can also help to interpret the results. If structure plots are made, look at the imaginary component because the single degree-of-freedom (SDOF) displacement response at resonance is purely imaginary when damping is present (this response does not occur in practice because the response is usually due to several modes).

5. If you have time-dependent loads, perform transient response (SOL 109 or SOL 112) analysis. If your structure is constrained, apply the load "very quickly" (over one or two time

steps) as a step function and look at the displacement results. The duration of the analysis needs to be as long as the period of the lowest frequency mode. For an SDOF system, a quickly applied load results in a peak displacement response that is twice the response resulting from the same load applied statically. This peak response does not occur in your actual model because of multiple modes and damping, but the results should be close. If your structure is unconstrained, the displacements will grow with time (unless the rigid-body modes are excluded in a modal transient response analysis). However, the stresses should be roughly twice those from the static analysis. In any event, examine the results via X-Y plots to ensure reasonableness.

Once you are satisfied, apply the correct time variation to the load and compute the results. Again, use X-Y plots to verify the accuracy of the results.

6. Finally, perform any other dynamic analyses, such as response spectrum analysis, random response, nonlinear transient response, or dynamic response optimization. The confidence gained by using the previous steps first helps to ensure that you have an accurate model at this stage.

These and other guidelines are described further in the remainder of this chapter.
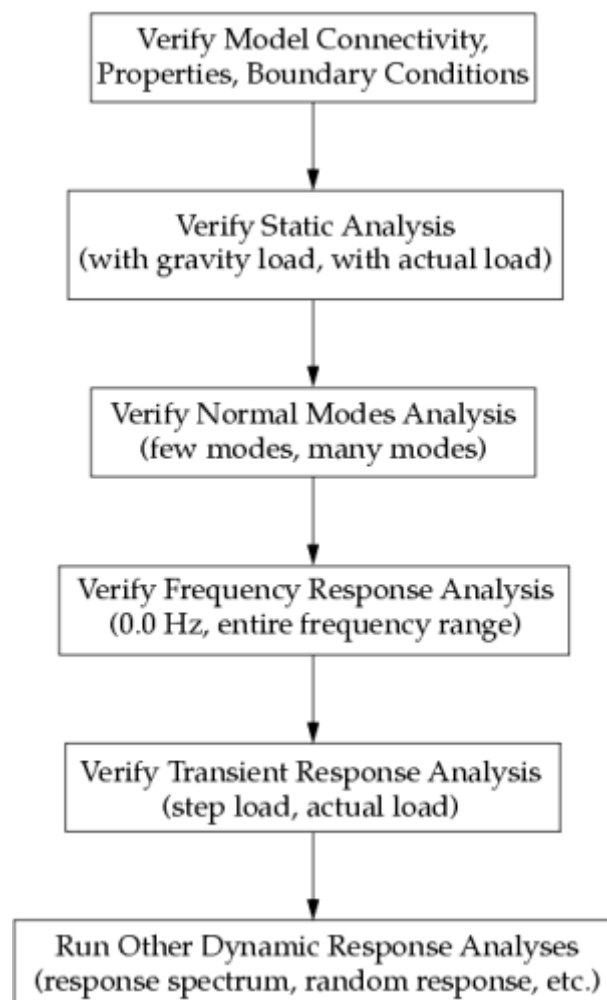
Figure 10-1. Simplified Flow Chart of the Overall Analysis Strategy

10.3   Units

Mistakes in units and boundary conditions (Boundary Conditions) are among the most common mistakes made when performing finite element analysis. NX Nastran does not assume a particular set of units, but it does assume that they are consistent.

It is up to you to ensure that the units that you use are both consistent and correct. Table 2-2 lists consistent units for common variables.

There are several ways to verify units. For mass, you can print the results from the grid point weight generator and verify that the mass is correct. For stiffness, you can apply a simple load and verify that the resulting static displacements seem reasonable. For both, you can verify that the natural frequencies are reasonable.

These checks assume that you have enough knowledge about your structure to know when the results are reasonable and when a mistake has been made. In other words, running small models and/or proceeding through dynamic analysis via the steps outlined in the previous section are necessary in order to be confident that the correct units are specified.

10.4   Mass

Mistakes with mass primarily involve mistakes in mass units as described in the previous section. A common mistake is to mix mass and weight units. Using PARAM,WTMASS does not solve this problem because it scales all mass and weight input (except certain types of direct input matrices) and still leaves the mixture of units. Also, the use of PARAM,WTMASS can have unwanted effects as well because it also scales the large mass used in enforced motion, thereby scaling the value of the enforced motion input. In order to reduce the chances for error, it is recommended that only mass units be used everywhere. Therefore, avoid the use of weight input wherever possible as well as the use of PARAM,WTMASS.

One way to verify mass input is to apply gravity in several directions and print the SPC forces. This verification ensures that the mass is correctly applied and that the units are correct.

10.5   Damping

Proper specification of damping is probably the most difficult modeling input to verify because its verification can only be done via dynamic response analysis. In addition, its units are not necessarily familiar because damping is normally not a modeling input that you use frequently, unless dynamic response analysis is all you run. Also, there are several ways to specify damping in NX Nastran, which increase the chances of making a mistake. Finally, even though the damping units are correct and the damping is input correctly in NX Nastran, it is difficult to know that the damping specification itself is correct from a physical standpoint. While there are relatively easy checks for correctness in mass and stiffness input, including comparison to static and modal test data, there are no such easy checks for damping input.

The easiest way to specify damping is to use modal damping, which is often specified as the percentage of critical damping. The TABDMP1 Bulk Data entry is used to specify modal damping. The following are several potential mistakes associated with using modal damping.

- Forgetting to select the TABDMP1 entry using the SDAMPING Case Control command.

- Forgetting to specify the damping TYPE (field 3). The default damping type is structural damping, which is twice the critical damping percentage.

- Not making the table cover an adequate range of frequencies. Like almost all of the NX Nastran tables, the TABDMP1 entry extrapolates beyond the endpoints of the table by using the first two or the last two entered values. As a rule, you should try to provide enough input points so that the table lookup point is always an interpolated value.

Modal damping can be used only in modal frequency response and modal transient response. Other forms of damping have to be used for the direct methods of response. For frequency response analysis, GE (field 9 of the MAT1 entry) and PARAM,G define structural damping. These variables are also used to specify structural damping for transient response analysis but are not activated unless PARAM,W3 and PARAM,W4 are set to nonzero values. A common mistake is to forget to set these values.

Note that damping is additive, that is, the damping matrix is comprised of all of the input damping sources. For example, damping due to CDAMPi elements adds to damping due to PARAM,G and GE. Mixing of damping types can increase the chances for error, and you are cautioned against the mixing of damping types unless it is necessary.

In many cases damping is not an important consideration. For example, a structure's peak response due to an impulsive load is relatively unaffected by damping since the peak response occurs during the first cycle of response. Damping in a long duration transient excitation, such as an earthquake, can make a difference in the peak response on the order of 10 to 20% or so, but this difference is small when compared to the other modeling uncertainties. Therefore, it is often conservative to ignore damping in a transient response analysis.

For frequency response analysis, however, the value of damping is critical to the value of the computed response at resonance since the dynamic magnification factor is inversely proportional to the damping value. One way to verify the accuracy of the modal damping ratio input is to run the modal frequency response across the half-power bandwidth of the modal frequencies of interest as shown in Figure 10-2.
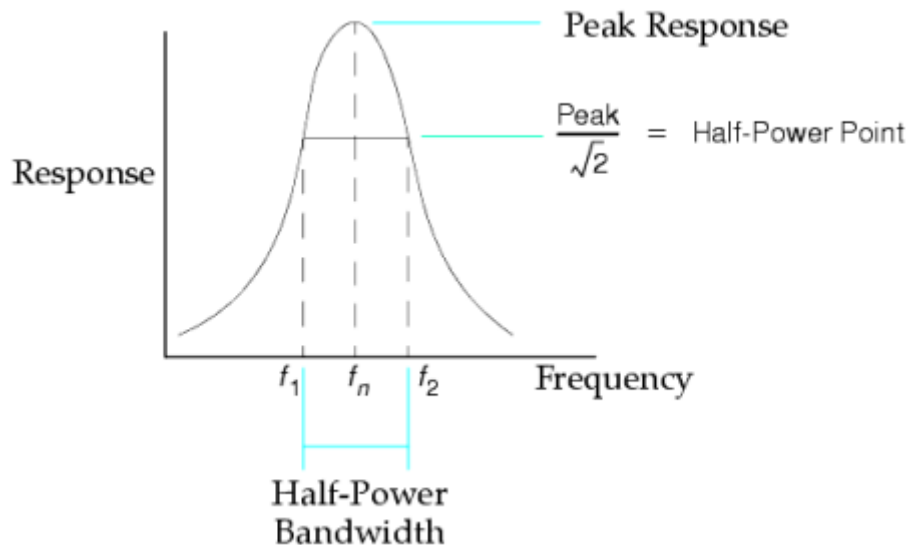
Figure 10-2. Half-Power Bandwidth

For lightly damped structures ($z < 0.1$), the approximate relationship between the critical damping ratio , the half-power bandwidth (f_2-f_1), and the resonant frequency f_n is

$$\zeta \approx \frac{f_2 - f_1}{2f_n}$$

Equation 10-1.

Another approximate way to verify damping is to run transient response analysis and look at the successive peak values of the free vibration response as shown in Figure 10-3. In direct transient response analysis, this plot can be generated by using a displacement initial condition; in modal transient response analysis, it can be generated by applying an impulsive force that ramps up and down quickly with respect to the dominant period of response. The logarithmic decrement Δ is the natural log of the amplitude ratio of two successive cycles of free vibration response given by

$$\Delta = \ln \frac{x_{(n-1)}}{x_n}$$

Equation 10-2.

The logarithmic decrement provides an approximate damping relationship for lightly damped structures as given by

$$\zeta \approx \frac{\Delta}{2\pi}$$

Equation 10-3.

In transient response analysis, remember to use PARAM,W3 or PARAM,W4 to include structural damping if GE or PARAM,G is used.
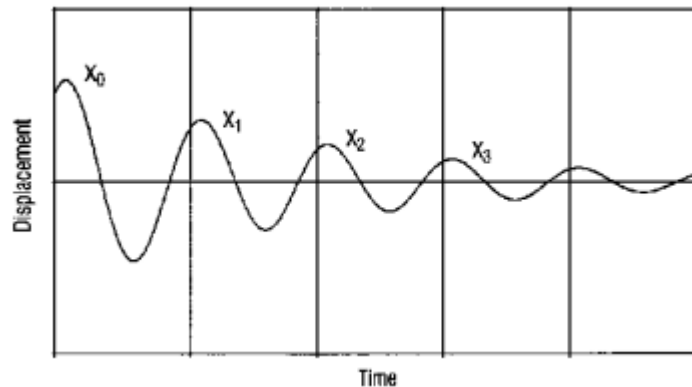
Figure 10-3. Damped Free Vibration Response

Both the half-power bandwidth method and the logarithmic decrement method assume an SDOF response. These approximations are less accurate when there are multiple modes of response; however, they are useful for verifying that the damping input is within a factor of two or three of the desired damping.

10.6   Boundary Conditions

The proper specification of boundary conditions is just as important for dynamic analysis as it is for static analysis. The improper specification of the boundary conditions leads to incorrect answers. One such improper specification of boundary conditions is forgetting to fully constrain the structure. Unlike static analysis, for which an under-constrained model does not run, an under-constrained model does run in dynamic analysis. You should perform an eigenvalue analysis first and verify that there are no unwanted rigid-body modes.

The large mass used for enforced motion simulates a constrained condition as well as adds a rigid-body mode. The value of the large mass is important for obtaining accurate answers. The large mass value must be large enough to properly simulate the constrained condition, but it must not be so large as to create numerical difficulties when solving the equations. A range of 10^3 to 10^6 times the overall structural mass is adequate to meet both conditions. One way to verify that a proper value is chosen is to run a normal modes analysis with the enforced DOF constrained via SPCs. Then, run a normal modes analysis with your choice of the large mass(es) and compare the frequencies of the flexible modes. If the frequencies compare favorably (i.e., to within four or five significant digits), then the large mass value is accurate. If the frequencies do not compare, then increase the value of the large mass(es). A common mistake is to use too low a value (or omit it entirely) for the rotational components. Using the grid point weight generator is very important to obtain the overall structural mass and inertias so that you can specify a good value for the large mass(es).

10.7   Loads

Because of their time- or frequency-varying nature, it is more complicated to apply dynamic loads than it is to apply static loads. Therefore, it is important to verify that the dynamic loads are correctly specified and that there are no discontinuous loads.

The best way to verify proper dynamic load specification is to plot the loads as a function of time or frequency. Another way to verify proper dynamic load specification is to print the loads as a function of time or frequency. Use the OLOAD Case Control command to plot or print the loads.

10.8   Meshing

An adequate element mesh is required to create an accurate model. For static analysis the mesh density is primarily controlled by the load paths; the element mesh must be fine enough so that there is a smooth transition of stress from one element to another in the region of interest.

Load paths are also important for dynamic analysis, but there is an additional consideration: the mesh must be fine enough to accurately represent the deformed shape of the modes of interest. If many modes are to be considered in the analysis, then the model must be fine enough to accurately represent the mode shapes of the highest modes of interest. Table 10-1 shows the frequencies resulting from several cantilever beam models; each column represents the same structure but with a different number of elements. Note that the frequencies are closer to the theoretical results as the model fineness in increased. In addition, note that the error is greatest in the higher modes. This table shows the computed frequencies compared to theory; it does not show the computed mode shapes compared to theory. However, the comparison for mode shapes shows even more error than is shown for the frequencies.

A general rule is to use at least five to ten grid points per half-cycle of response amplitude. Figure 10-4 shows the theoretical response shape for the fifth mode. Note that there are four half-cycles in the mode shape, which means that 20 to 40 grid points at a minimum are recommended to

accurately represent that mode shape. This modeling guideline is also reflected in Table 10-1, which shows that the 40-element model is much more accurate than the 10-element model.

Mode	Frequencies (Hz)					
	Theory	10-Element Model	40-Element Model	Theory	70-Element Model	100-Element Model
1	2.08	2.07	2.08	2.08	2.08	2.08
2	13.24	12.81	13.00	13.24	13.01	13.01
3	36.47	35.50	36.37	36.47	36.41	36.42
4	71.52	68.84	71.22	71.52	71.34	71.36
5	118.21	112.53	117.65	118.21	117.90	117.95

Table 10-1. Frequencies for a Cantilever Beam Model (Lumped Mass)

Four Half-Cycles
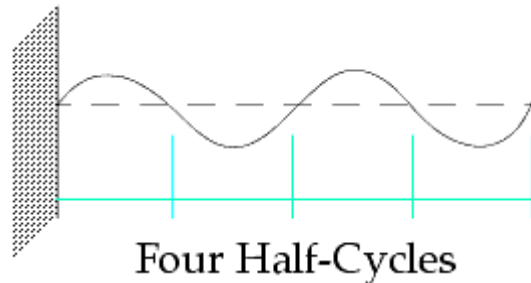
Figure 10-4. Fifth Mode Shape of a Cantilever Beam

Another way to verify the accuracy of the mesh density is to apply static loads that give a deformed shape the same as the mode of interest and perform stress discontinuity calculations. This process can be laborious and is not recommended as a general checkout procedure.

10.9 Eigenvalue Analysis

In addition to the meshing guidelines described earlier, the other primary factor in eigenvalue analysis is the proper selection of the eigenvalue solution method. The Lanczos method is the best overall method and is the recommended method to use. The automatic Householder method is useful for small, dense matrices. The SINV method is useful when only a few modes are required. The other methods should be regarded as backup methods.

Carefully examine the computed frequencies and mode shapes. Viewing only one or the other is usually not enough to verify accuracy of your model. Modes with 0.0 Hz (or computational zero) frequencies indicate rigid-body or mechanism modes. If these modes are unintended, then there is a mistake in boundary conditions or connectivity. The existence and cause of unintended zero-frequency modes can also be ascertained from the mode shapes. In addition, mode shape plots are useful for assessing local modes, in which a group of one or a few grid points displaces and the rest of the structure does not. Local modes may also be unintended and are often the result of incorrect connectivity or element properties.

10.10 Frequency Response Analysis

Several factors are important for computing accurate frequency response results. These factors include the number of retained modes (for modal frequency response analysis), the frequency

increment Δf , and damping. These guidelines are only approximate but are nevertheless useful. Running a normal modes analysis first helps to compute accurate frequency response results.

Number of Retained Modes

Use enough modes to cover the range of excitation. For example, if the model is to be excited from 1 to 100 Hz, you must use all of the modes with frequencies up to at least 100 Hz. This is only a minimum requirement, however. A better guideline is to use enough modes to cover two to three times the range of excitation in order to provide accurate answers at the high end of the frequency range. For example, when excitation is applied to 100 Hz, modes with frequencies up to 200 to 300 Hz should all be used.

Size of the Frequency Increment

The size of the frequency increment Δf must be small enough to ensure that the magnitude of the peak response is accurately computed. To ensure this, you need to choose a frequency increment small enough so that there are at least five to ten increments within the half-power bandwidth frequencies (illustrated in Figure 10-2).

The frequency increment is defined by Δf on the FREQ1 Bulk Data entry. Note that FREQ and FREQ2 entries can be used in conjunction with FREQ1 to define more solutions in the areas of resonance (the frequencies of these solutions should have been determined by a prior normal modes analysis). A nonuniform Δf imposes no cost increase relative to a uniform Δf .

Relationship of Damping to the Frequency Increment

The response at resonance is inversely proportional to the amount of damping, but the half-power bandwidth is directly proportional to the amount of damping. For lightly-damped structures (z < 0.1), an approximate relationship between the half-power bandwidth (f_2- f_1), resonant frequency f_n , and critical damping ratio z is given by Equation10-1. This equation can be rewritten to define the appropriate value of Δf :

$$f_2 - f_1 \;=\; (m-1)\Delta f$$

Equation 10-4.

$$\zeta \approx \frac{f_2 - f_1}{2f_n} \;=\; \frac{(m-1)\Delta f}{2f_n}$$

Equation 10-5.

$$\Delta f \;=\; \frac{2\zeta f_n}{(m-1)}$$

Equation 10-6.

where m is the number of frequency points within the half-power bandwidth. For example, with 1% critical damping and six points used to define the half-power bandwidth, the maximum

frequency increment is 0.0004 $\cdot f_n$. The frequency increment is smaller for lighter damped structures.

Another good check is looking at the X-Y plots. If the response curves are not smooth, there is a good chance that Δf is too large.

Verification of the Applied Load

The applied load can be verified by exciting your model at 0.0 Hz and comparing the results to a static solution with the same spatial load distribution. The 0.0 Hz results should match the static results if direct frequency response analysis is used without structural damping. If modal frequency response analysis is used without structural damping, those results should be close to the static results; any difference is due to mode truncation. If the 0.0 Hz results do not match the static results, check the LSEQ and DAREA entries. Also, use OLOAD to print the applied force in order to compare the loads.

10.11   Transient Response Analysis

Several factors are important in computing accurate transient response. These factors include: the number of retained modes (for modal transient response), the integration time step Δt, the time duration of the computed response, and damping. The guidelines are only approximate but are nevertheless useful. Running a normal modes analysis first helps to compute transient response.

Number of Retained Modes

In modal transient response analysis, a larger number of modes produces a more accurate response (although at the expense of increased run times). The number of modes must be large enough to cover the frequency range of interest. (The term "range of interest" means the range of frequencies whose response is to be computed as well as the range of frequencies for which the loading is applied.) As a general rule, there should be enough modes to cover a range up to about two times the highest frequency of interest. For example, if response needs to be computed to 100 Hz, then enough modes should be used in the transient response to cover up to at least 200 Hz. As another general rule, you should use enough modes to cover a range up to two to ten times the dominant frequency of response.

A sufficient number of modes must be retained to cover the time and spatial distribution of loading. For example, if the applied transient load has a spatial distribution resembling the fifth mode shape, then that mode should be included in the transient response.

Size of the Integration Time Step

The value of the integration time step Δt, denoted by DT on the TSTEP Bulk Data entry, is important for the accurate integration of the equations of motion. Rough guidelines for the selection of Δt are as follows:

- Δt must be small enough to accurately capture the magnitude of the peak response, which means that at least ten time steps per cycle of the highest mode be used. For example, if the highest frequency of interest is 100 Hz, then Δt should be 0.001 second or smaller.

- Δt must be small enough to accurately represent the frequency content of the applied loading. If the applied loading has a frequency content of 1000 Hz, then Δt must be 0.001

second or less (preferably much less in order to represent the applied loading with more than one point per cycle).

The integration time step can be changed in a transient response analysis, but it is not recommended. Much of the cost of direct transient response occurs with the decomposition of the dynamic matrix, which occurs only once if there is a constant Δt. A new decomposition is performed every time Δt changes, thereby increasing the cost of the analysis if the time increment is changed. Therefore, the recommendation is to determine the smallest Δt required based on the loading and frequency range and then use that Δt throughout the entire transient response analysis.

Duration of the Computed Response

The length of the time duration is important so that the lowest flexible (e.g., non-rigid body) mode oscillates through at least one cycle. For example, if the lowest flexible mode has a frequency of 0.2 Hz, then the time duration of the computed response should be at least 5.0second. A better guideline is to make the duration the longer of the following: twice the period of the lowest flexible mode or one period of the lowest flexible mode after the load has been removed or reached a constant value. The time duration is set on the TSTEP entry by multiplying (the integration time step) by the number of time steps (N).

Value of Damping

The proper selection of the damping value is relatively unimportant for analyses that are of very short duration, such as a crash impulse or a shock blast. The specification of the damping value is most important for long duration loadings (such as earthquakes) and is critical for loadings (such as sine dwells) that continually add energy into the system.

Verification of the Applied Load

The applied load can be verified by applying the load suddenly (over one or two time increments) and comparing the results to a static solution with the same spatial load distribution. The transient results should have a peak value of twice the static results. If the transient results are not twice the static results, check the LSEQ and DAREA entries. Another way to verify the applied load is to inspect it visually via the X-Y plots.

10.12   Results Interpretation and Verification

Because of the time- and frequency-varying nature of dynamic analysis, results can be more difficult to interpret than for static analysis. The key to proper results interpretation is plotting. Structure plotting is necessary for a proper understanding of the deformed shapes of the modes, and X-Y plotting is necessary for a proper understanding of the frequency and transient response.

Comparing results to hand calculations, to known results from similar models, or to test data is also very useful. Do not accept any analysis results without first performing some type of verification.

Consider the cantilever beam model shown in Figure 10-5. This is a planar model of the cantilever beam used in the examples in Chapter 3 through Chapter 7. Figure 10-5 shows the loads applied to the beam.
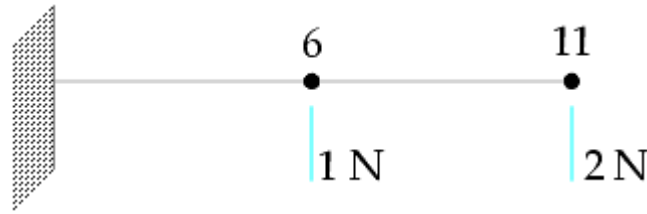
Figure 10-5. Cantilever Beam Model with Static Loads

A static analysis (SOL 101) is run first. Then, modal frequency response (SOL 111) is run from 0.0 to 20.0 Hz with a damping ratio of 2% critical damping used for all modes. Modes are computed up to 500 Hz using the Lanczos method. Finally, modal transient response (SOL112) is run with the time variation shown in Figure 10-6. Modes are computed up to 3000Hz (using the Lanczos method), and a damping ratio of 2% critical damping is used for all modes.
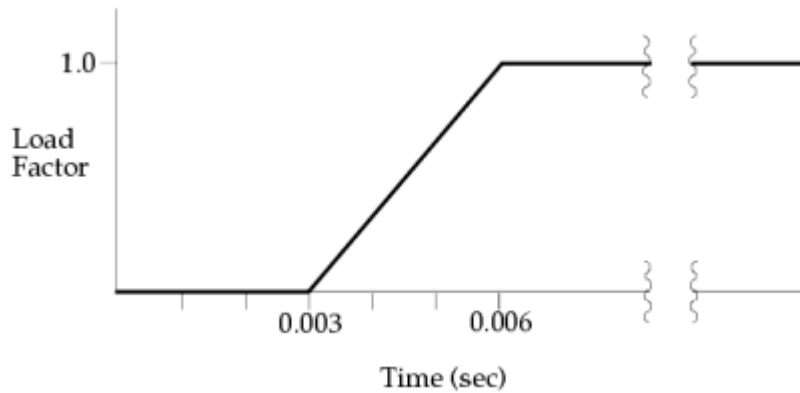
Figure 10-6. Time Variation of Transient Loads

Figure 10-2 shows the y-displacements for grid points 6 and 11. As expected, the frequency response results at 0.0 Hz are the same as the static analysis results (see Figure 10-). The frequency response results at 2.05 Hz (very near the first natural frequency) are approximately 25 times the static analysis results. The factor of 25 is the dynamic amplification factor at resonance for a damping ratio of 2% critical damping.

$$\text{Amplification Factor at Resonance} = \quad \frac{1}{2\zeta} = \frac{1}{2(.02)} = 25$$

The transient response results are approximately twice the static analysis results. The factor of two is the amplification of response for a transient load applied suddenly (see "Transient Response Analysis").

Table 10-2. Comparison of Results for the Cantilever Beam Model

	Static Analysis Results	Frequency Response Analysis Results		Transient Response Peak Displacement
		0.0 Hz	2.05 Hz	
Displacement 6 Y	3.17E-3	3.17E-3	7.61E-2	6.29E-3
Displacement 11 Y	9.77E-3	9.77E-3	2.25E-1	1.87E-2

10.13 Computer Resource Requirements

The efficiency of a dynamic analysis can be measured based on computer resource requirements, such as computer runtime, disk storage, and memory. These requirements increase as the problem size increases, but they can also vary greatly depending on other factors.

In general, a dynamic analysis uses more computer resources than a static analysis. In normal modes analysis, the requirements vary depending on the density of the model, the eigenvalue extraction method used, the number of modes requested, and the type of decomposition (symmetric or sparse).

If your model has already been built, you can use the ESTIMATE utility to estimate resource requirements. See:

- "ESTIMATE" in the *NX Nastran Installation and Operations Guide*

If you haven't yet created your model but still want to estimate resource needs, then read the following.

In the past, we have established benchmark runs to provide guidelines on performance for normal modes analyses. We used a cylindrical plate model scaled to various sizes and solved for ten modes using different solution methods, principally the Sparse Lanczos method. Testing has shown that this model provides a very good guideline for most industry models.

Memory requirements increase with problem size. Empirical formulas were developed for these benchmark problems to estimate the memory requirement given the number of degrees-of-freedom (DOF) contained in the model.

For the equation

$$\text{Log(mem)} = (.7934) \cdot \text{Log(DOF)} - 2.3671$$

Equation 10-7.

mem	The memory required in megabytes.
DOF	The number of degrees-of-freedom contained in the model.

Note that these memory requirements are for no spill conditions. The symmetric solver can run with less memory, but because of spill conditions, the performance is degraded in both computer runtime and disk space usage.

Empirical formulas were developed for these benchmark problems to estimate the disk space requirement given the number of degrees-of-freedom (DOF) contained in the model.

For the equation

$$\text{Log(space)} = (1.0815) \cdot \text{Log(DOF)} - 2.5411$$

Equation 10-8.

space	The disk space usage in megabytes.
DOF	Represents the number of degrees-of-freedom contained in the model.

These values were obtained from a 32-bit computer and therefore should be doubled for a 64-bit computer. If you wish to calculate more than ten modes, then multiply the space calculated

above by 1.18^(#modes/10 - 1). For example, for 30 modes you would multiply the space value calculated above by 1.18^2.

Chapter

11   Advanced Dynamic Analysis Capabilities

- *Overview*

- *Dynamic Reduction*

- *Complex Eigenvalue Analysis*

- *Response Spectrum Analysis*

- *Random Vibration Analysis*

- *Mode Acceleration Method*

- *Fluid Structure Interaction*

- *Nonlinear Transient Response Analysis*

- *Superelement Analysis*

- *Design Optimization and Sensitivity*

- *Control System Analysis*

- *Aeroelastic Analysis*

- *DMAP*

11.1   Overview

The previous chapters describe the most common types of dynamic analysis: normal modes analysis, frequency response analysis, and transient response analysis. However, NX Nastran contains numerous additional types of dynamic analyses; many of these types are documented in the *NX Nastran Advanced Dynamic Analysis User's Guide*.

The advanced dynamic analysis capabilities include

• Dynamic reduction

• Complex eigenanalysis

• Response spectrum analysis

• Random response analysis

• Mode acceleration method

• Fluid-structure interaction

• Nonlinear transient response analysis

• Superelement analysis

• Design sensitivity and optimization

• Control systems

• Aeroelastic analysis

• DMAP

These capabilities are described briefly in the remainder of this chapter.

11.2   Dynamic Reduction

Dynamic reduction is an optional numerical approach that can be used in NX Nastran to reduce a dynamic model to one with fewer degrees-of-freedom. Typically, the intent of dynamic reduction is to increase the efficiency of a dynamic solution by working with smaller matrices while maintaining the dynamic characteristics of the system.

Dynamic reduction is used for a number of reasons. One possible reason may be that a particular model may be too large from a computer resource standpoint (computer runtime and/or disk space) to be solved without using reduction. A related issue is that the model may have more detail than required. Many times dynamic analyses are performed on models that were initially created for detailed static stress analyses, which typically require refined meshes to accurately predict stresses. When these static models are used in a dynamic analysis, the detailed meshes result in significantly more detail than is reasonably required to predict the natural frequencies and mode shapes accurately.

Static condensation (also called Guyan reduction) is the available method in NX Nastran for the dynamic reduction of the eigenequation prior to modal extraction.

Once the natural frequencies and mode shapes are calculated for the reduced model, these modes can be used in the transient or frequency response solution process.

Static Condensation

In the static condensation process you select a set of dynamic DOFs called the a-set; these are the retained DOFs that form the analysis set. The complementary set is called the o-set and is the set of DOFs that are omitted from the dynamic analysis through a reduction process. The process distributes the o-set mass, stiffness, and damping to the a-set DOFs by using a transformation that is based on a partition of the stiffness matrix (hence the term static condensation). This reduction process is exact only for static analysis, which leads to approximations in the dynamic response. The a-set DOFs are defined by the ASET or ASET1 Bulk Data entries, and the o-set DOFs are defined by the OMIT or OMIT1 Bulk Data entries.

It is emphasized that dynamic reduction is an optional technique and is best left to the dynamic analysis specialist. Whereas dynamic reduction was required in the days of small computer memory and disk space, now it is no longer required due to increased computer resources and better eigenanalysis methods (in particular, the Lanczos method).

11.3 Complex Eigenvalue Analysis

Complex eigenvalue analysis is used to compute the damped modes of structures and assess the stability of systems modeled with transfer functions (including servomechanisms and rotating systems).

Complex eigenvalue analysis solves for the eigenvalues and mode shapes based on the following equation in operator notation:

$$[p^2 M + pB + K]\{u\} = 0$$

Equation 11-1.

where:

$$p = \alpha + i\omega$$

This equation is similar to that for normal modes analysis (see "Real Eigenvalue Analysis") except that damping is added and the eigenvalue is now complex. In addition, the mass, damping, and stiffness matrices may be unsymmetrical, and they may contain complex coefficients.

Complex eigenvalue analysis is controlled with the EIGC Bulk Data entry (similar to the EIGRL or EIGR Bulk Data entries for normal modes analysis). There are four methods of solution: upper Hessenberg, complex Lanczos, determinant search, and inverse power. Complex eigenvalue analysis is available as a direct method (SOL 107), in which the equations are of the same size as the number of physical variables. Complex eigenvalue analysis is also available as a modal method (SOL 110), in which undamped modes are first computed and then are used to transform the matrices from physical to modal variables.

11.4 Response Spectrum Analysis

Response spectrum analysis is an approximate method of computing the peak response of a transient excitation applied to a simple structure or component. This method is used in civil engineering to predict the peak response of a component on a building that is subjected to earthquake excitation; it is also used in aerospace engineering to predict the peak response of equipment in a spacecraft that is subjected to an impulsive load due to stage separation. Because

it is an approximate method, response spectrum analysis is often used as a design tool. Response spectrum analysis is also called shock spectrum analysis.

There are two parts to response spectrum analysis: (1) generation of the spectrum and (2) use of the spectrum for dynamic response such as stress analysis. Both are available in NX Nastran.

Figure 11-1 depicts the generation of a response spectrum.

Transient excitation (a) is applied to a base structure (b), from which transient response (c) is computed for each floor. This response is applied to a series of oscillators (d), for which the peak response is plotted (e). Steps (d) and (e) are repeated for different damping values to form response spectra as shown below.
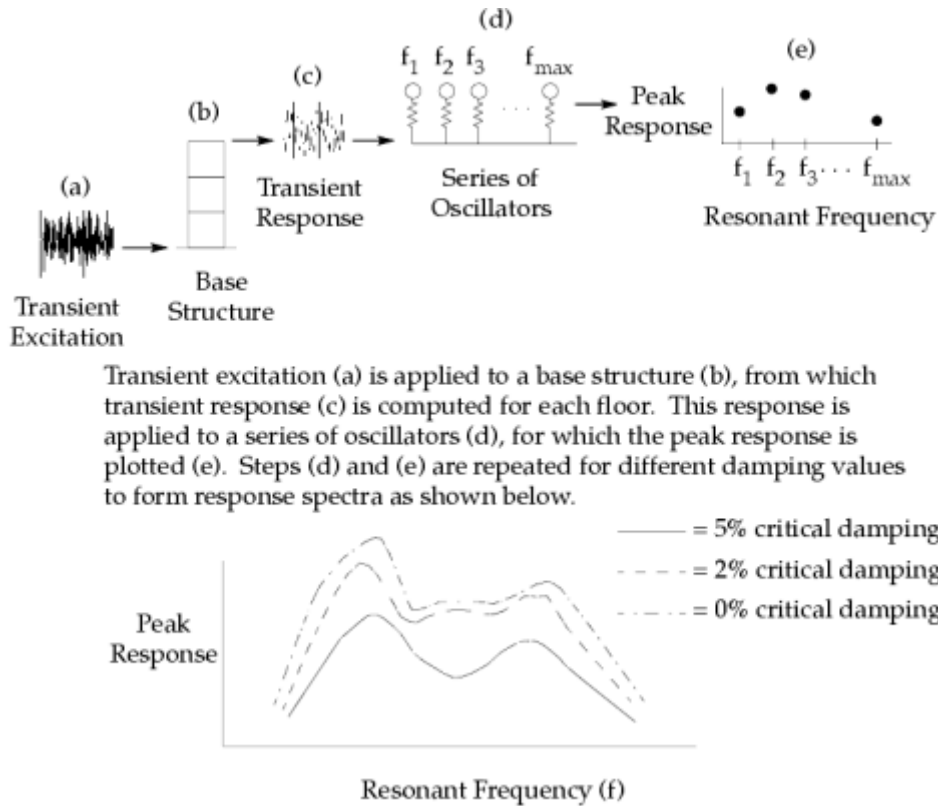
Figure 11-1. Response Spectrum Generation

Note that the peak response for one oscillator does not necessarily occur at the same time as the peak response for another oscillator. Also note that there is no phase information since only the magnitude of peak response is computed. It is assumed in this process that each oscillator mass is very small relative to the base structural mass so that the oscillator does not influence the dynamic behavior of the base structure.

Once a spectrum is computed, it can be used for the dynamic response analysis of an NX Nastran model of the component. For example, the spectrum generated for a floor in a building that is subjected to an earthquake can then be applied to a complex model of a piece of equipment attached to that floor. The peak response of each mode of the equipment model is obtained from the spectrum, and these peak modal responses are combined to create the overall response.

Because the peak responses do not all occur at the same time and only the magnitude of peak responses are computed, various methods are used to combine the peak responses into the overall response. The combination methods implemented in NX Nastran are SRSS (square root of the sum of the squares), ABS (absolute values), and NRL (U.S. Navy shock design modal summation). The typical response quantities computed are grid point displacements and element stresses.

11.5   Random Vibration Analysis

Random vibration is vibration that can be described only in a statistical sense. The instantaneous magnitude is not known at any given time; rather, the magnitude is expressed in terms of its statistical properties (such as mean value, standard deviation, and probability of exceeding a certain value).

Examples of random vibration include earthquake ground motion, ocean wave heights and frequencies, wind pressure fluctuations on aircraft and tall buildings, and acoustic excitation due to rocket and jet engine noise. These random excitations are usually described in terms of a power spectral density (PSD) function.

NX Nastran performs random response analysis as a postprocessing step after frequency response analysis. The frequency response analysis is used to generate the transfer function, which is the ratio of the output to the input. The input PSD multiplies the transfer function to form a response PSD. The input PSD can be in the form of auto- or cross-spectral densities. Random response output consists of the response PSD, autocorrelation functions, number of zero crossings with positive slope per unit time, and RMS (root-mean-square) values of response.

11.6   Mode Acceleration Method

The mode acceleration method is an alternate form of data recovery for modal frequency response and modal transient response. The mode acceleration method accounts for the higher truncated modes to give more accurate answers than either the mode displacement or matrix methods (see Modal Frequency Response Analysis and "Modal Transient Response Analysis"), particularly if the number of retained modes is small compared to the number of physical degrees-of-freedom.

The higher modes respond in a quasi-static manner to lower frequency excitation. Therefore, the inertia and damping forces contain little contribution from these higher modes. In the mode acceleration method the inertia and damping forces are first computed from the modal solution. These forces are then combined with the applied forces and are used to obtain more accurate displacements by performing a static analysis. The mode acceleration method provides more accurate answers but with increased cost.

The mode acceleration method is selected by specifying PARAM,DDRMM,-1 and PARAM,MODACC,0.

11.7   Fluid Structure Interaction

Four major methods are available in NX Nastran to model fluid effects. These methods are described below.

Hydroelastic Analysis

Small motions of compressible or incompressible fluids coupled to a structure may be analyzed with this option. The fluid is modeled with axisymmetric hydroelastic elements (CFLUIDi), which may be connected to an arbitrary structure modeled with an axisymmetric wetted surface. Each fluid point (RINGFL) on a cross section defines the scalar pressure, which is expanded to a Fourier series around the circumference. Complex modes and frequency response solutions are available for the coupled fluid-structure problems. Normal modes solutions are available for fluid-only problems. All solutions may include gravity effects (i.e., sloshing) on a free surface. This capability was developed specifically to analyze liquid-fueled booster rockets but may also be useful for problems involving other types of axisymmetric storage tanks.

Virtual Fluid Mass

Small motions of incompressible fluids may be coupled directly to the structure with this option. Fluids may be coupled to the interior and exterior surfaces (with infinite fluid boundaries). There is no explicit fluid model; only the wetted structural elements (ELIST) have to be defined. Although free surfaces are allowed, no gravity effects are included directly. Since the fluid is represented by a coupled mass matrix attached directly to the structural points, this capability is allowed in all dynamic solution sequences. This capability may be used to model a wide variety of fluid-structure interaction problems. Some examples are fuel tanks, nuclear fluid containers, drilling platforms, and underwater devices.

Coupled Acoustics

You may analyze the dynamics of compressible fluids contained within arbitrarily shaped structures with the coupled fluid-structure method. You would model a three-dimensional fluid with conventional solid elements (CHEXA, etc.) using acoustic property and material data. Each grid point in the fluid defines the pressure at its location as its degree of freedom. The fluid is automatically connected to the structure via the geometry and ACMODL Bulk Data inputs. You can connect acoustic absorber elements (CHACAB) to the structural surfaces to simulate soundproofing material. In addition, the CAABSF element is now available. This element acts as a thin layer of acoustic absorbing material along the fluid-structure interface. Version 69 introduced several features, which are parallel to those available for structural analysis, such as direct damping, modal damping and the ability to control the modes in a response analysis through the use of parameter. You can define panels to provide integrated response data. Effects of gravity, large motions, and static pressures are ignored. Complex eigenvalues, frequency response, and transient response are the available solution sequences. Design sensitivity and optimization processes may reference the acoustic outputs as responses with appropriate design constraints. Applications for the coupled fluid-structure option are automotive and truck interiors, aircraft cabins, and acoustic devices, such as loudspeakers and microphones.

Uncoupled Acoustics

Several methods are available in NX Nastran for the analysis of normal modes of compressible fluids bounded by rigid containers and/or free surfaces. One method is the "acoustic cavity" capability, which uses two-dimensional slot elements and axisymmetric ring elements to define the fluid region. This method was specifically developed for the acoustic analysis of solid rocket motor cavities. A better method is to use the three-dimensional fluid elements for the "coupled acoustics" described above and provide the appropriate boundary conditions.

11.8   Nonlinear Transient Response Analysis

The analysis techniques described thus far are applicable for linear-elastic analysis for which the mass, stiffness, and damping matrices are constant over time and constant for all values of applied force. However, there are many cases for which the matrices are not constant, and these cases must be solved with nonlinear analysis techniques.

Nonlinear analysis requires iterative solution methods, thereby making it far more computationally intensive than a corresponding linear analysis. Nonlinear transient response analysis is available in NX Nastran in SOL 129. Nonlinear problems are classified into three broad categories: geometric nonlinearity, material nonlinearity, and contact.

Geometric Nonlinearity

Geometrically nonlinear problems involve large displacements; "large" means that the displacements invalidate the small displacement assumptions inherent in the equations of linear analysis. For example, consider a thin plate subject to an out-of-plane load. If the deflection of the plate's midplane is approximately equal to the thickness of the plate, then the displacement is considered large, and a linear analysis is not applicable.

Another aspect of geometric nonlinear analysis involves follower forces. Consider a clamped plate subject to a pressure load. As shown in Figure 11-2, the load has followed the plate to its deformed position. Capturing this behavior requires the iterative update techniques of nonlinear analysis.
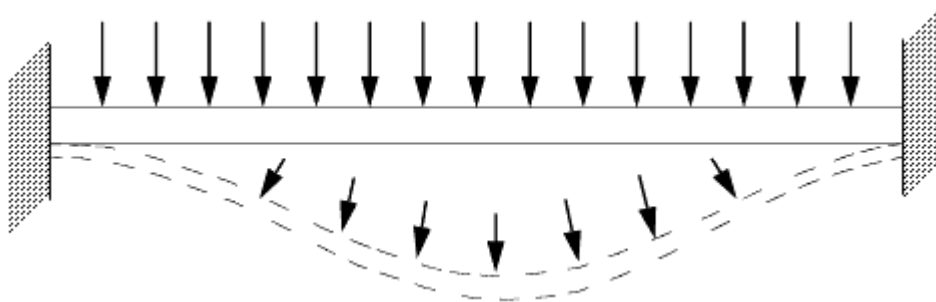
Figure 11-2. Follower Forces on a Cantilever Beam

Material Nonlinearity

Material nonlinear analysis can be used to analyze problems where the stress-strain relationship of the material is nonlinear. In addition, moderately large strain values can be analyzed. Examples of material nonlinearities include metal plasticity, materials (such as soils and concrete), and rubbery materials (where the stress-strain relationship is nonlinear elastic). Various yield criteria, such as von Mises or Tresca (for metals) and Mohr-Coulomb or Drucker-Prager (for frictional materials, such as soils or concrete), can be selected. Three hardening rules are available in NX Nastran: isotropic hardening, kinematic hardening, or combined isotropic and kinematic hardening. With such generality, most plastic material behavior with or without the Bauschinger effect can be modeled. In addition, gaps can be used to model the effects due to structural separation.

Contact

Contact occurs when two or more structures (or portions of structures) collide. Contact can be modeled as point-to-point contact (CGAP) or as contact along a line (BLSEG).

Nonlinear-Elastic Transient Response Analysis

There are numerous structures that contain nonlinear elastic elements. These elements possess nonlinear force-deflection relationships, yet they are elastic since they load and unload along the same force-deflection curve. Examples of the nonlinear elastic force-deflection curves are shown in Figure 11-3. These types of elements are common in many mechanical and structural systems including piping systems supported by nonlinear springs, chains (stiffness in tension only), or constant force devices; base-mounted equipment and structure supported by nonlinear shock isolation systems; structures with gaps (piping systems and buildings); and soil or

concrete structures (which only exhibit stiffness when in compression). Other systems exhibit nonlinearities that are proportional to the velocity, such as seat belts and piping supports.

(a) Bilinear Spring

(b) Preload Constant Force Device with Hard Stop

(c) Gap and Spring in Series

(d) Stiffness in Compression Only

(e) Softening System

(f) Hardening System
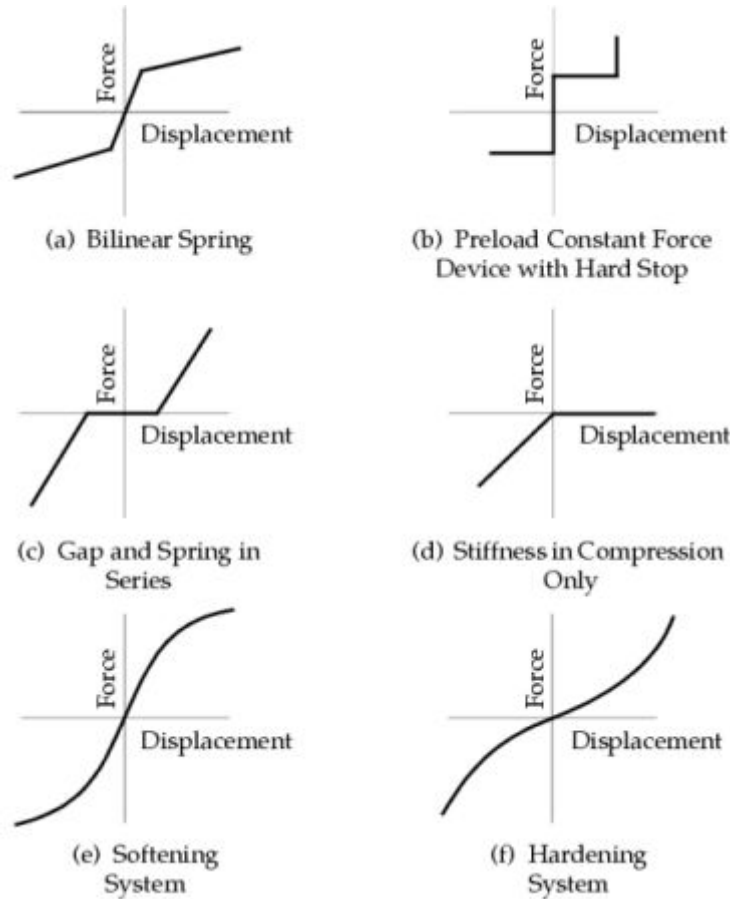
Figure 11-3. Examples of Nonlinear Elastic Systems

There are several methods in NX Nastran for solving nonlinear elastic problems. A general nonlinear solution scheme can be used, in which the material properties are specified as nonlinear elastic (NLELAST on the MATS1 entry). Nonlinear element stiffness matrices are generated, and equilibrium iterations and convergence checks are required at each load step. The new CBUSH element, introduced in Version 69 adds some capabilities in this area, also. In addition, gap elements (CGAP) can be used in conjunction with elastic elements to produce systems with piece-wise linear force-deflection curves. This process also requires the additional computations of nonlinear element stiffness matrix generation, equilibrium iteration, and convergence testing. Nonlinear analyses requiring these additional computations can be substantially more costly than an equivalent linear analysis even if the nonlinearities are relatively few in number.

An efficient technique called the pseudoforce method exists in NX Nastran, in which the localized nonlinearities are treated so that they require no additional computer time when compared to a purely linear analysis. In this method, which is available only for transient response solutions (SOLs 109 and 112, for example), deviations from linearity are treated as additional applied loads. The dynamic equations of motion are written as

$$[M]\{\ddot{u}(t)\} + [B]\{\dot{u}(t)\} + [K]\{u(t)\} = \{P(t)\} + \{N(t)\}$$

Equation 11-2.

where $[M]$, $[B]$, and $[K]$ denote the system mass, damping, and stiffness matrices, respectively. The vectors $\{(t)\}$ and $\{u(t)\}$ denote applied nodal loads and system displacements, respectively, as functions of time. The vector $\{N(t)\}$ denotes the nonlinear forces, which are added to the right-hand side of Equation 11-2 (and hence are treated as additional applied loads).

The nonlinear forces are evaluated at the end of one time step for use in the successive time step. The equations of motion therefore become the following:

$$[M]\{\ddot{u}(t)\} + [C]\{\dot{u}(t)\} + [K]\{u(t)\} = \{P(t)\} + \{N(t - \Delta t)\}$$

Equation 11-3.

Note that the nonlinear force lags the true solution by a time step, which may require using small integration time steps (i.e., smaller than those required for a purely linear analysis). Equation 11-3 can be solved in physical or modal coordinates (the nonlinearity itself must be expressed in terms of physical coordinates).

A nonlinear force can be used in conjunction with a linear elastic element to produce the desired force-deflection curve as illustrated in Figure 11-4. The nonlinear dynamic force is formulated using a NOLINi entry and a TABLEDi entry, which contains a force-versus-deflection table describing the nonlinear force. For desired force-deflection curves more complicated than the bilinear stiffness shown in the figure, the nonlinear force is made correspondingly more complex.
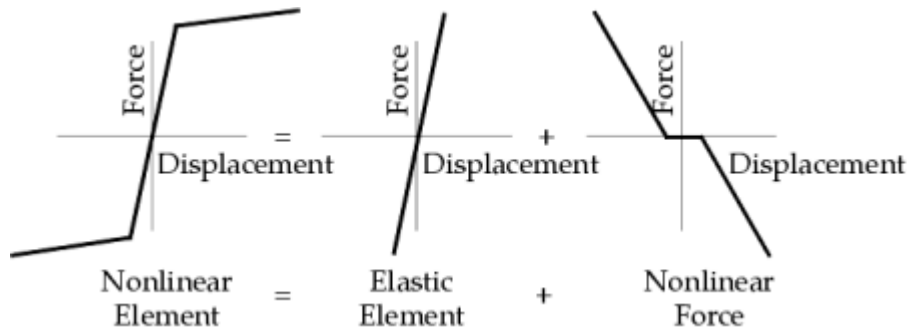
Figure 11-4. Formulation of a Nonlinear Element

Nonlinear Normal Modes Analysis

There are times when normal modes need to be calculated for a nonlinear structure. For inflatable, rotating, or tension structures, the stiffness—and therefore, the modes—depends upon the applied load and deformation. A nonlinear analysis can be restarted into SOL 103 to compute the normal modes of the deformed structure.

11.9 Superelement Analysis

Superelement analysis is a form of substructuring wherein a model is divided into pieces with each piece represented by a reduced stiffness, mass, and damping matrix. Each superelement is processed individually, and the solutions are then combined to solve the entire model. The final analysis (in which all of the individual superelement solutions are combined) involves much smaller matrices than would be required if the entire model were solved in a one step process. An example of a superelement model is shown in Figure 11-5.

Superelement analysis has the advantage of reducing computer resource requirements, especially if changes are made to only one portion (superelement) of the model; in this case, only the affected superelement needs to be reanalyzed and the final analysis repeated.

Superelement analysis presents procedural advantages as well, particularly when multiple engineering contractors are involved in an analysis. Imagine a model of a rocket and payload: one contractor models the booster, another contractor models the engines, and another contractor models the satellite. Each contractor can reduce his model to its boundary degrees-of-freedom, which is suitable for superelement analysis. The systems integrator then combines these reduced models into one model for a liftoff analysis. Superelement analysis has the advantage that matrices can be passed from one organization to another without revealing proprietary modeling details or concern about whether the same superelement interior grid point and element numbers are used by every participant.
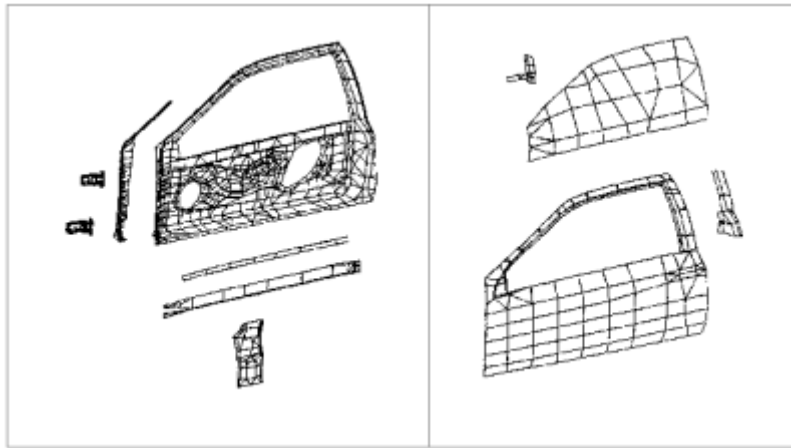
Figure 11-5. Superelements Used to Model a Car Door

Component Mode Synthesis

Component mode synthesis (CMS) is a form of superelement dynamic reduction wherein matrices are defined in terms of modal coordinates (corresponding to the superelement modes) and physical coordinates (corresponding to the grid points on the superelement boundaries). CMS is advantageous because there are fewer modal coordinates than physical coordinates—perhaps only one percent as many. In addition, CMS can utilize modal test data, thereby increasing the accuracy of the overall analysis.

11.10   Design Optimization and Sensitivity

Design optimization is used to produce a design that possesses some optimal characteristics, such as minimum weight, maximum first natural frequency, or minimum noise levels. Design optimization is available in NX Nastran in SOL 200, in which a structure can be optimized considering simultaneous static, normal modes, buckling, transient response, frequency response, and aeroelastic analyses.

Possible analysis response types include:

- Weight

- Volume

- Eigenvalues

- Buckling load factor

- Static displacement, stress, strain, and element force

- Composite stress, strain, and failure criterion

- Frequency response displacement, velocity, acceleration, stress, and force

- Transient response displacement, velocity, acceleration, stress, and force

- Damping level in a flutter analysis

- Trim and stability derivative responses for static aeroelastic analysis

A frequent task is to minimize the total structural weight while ensuring that the design still satisfies all of the design-related and performance-related constraints. The structural weight is termed the design objective, which the optimizer attempts to minimize. The design variables are the properties that can be changed to minimize the weight. The design variables shown in Figure 11-6 are the height and width of the flange and web. Constraints on the design variables may be a minimum allowable flange width w and a maximum allowable beam height h. With the tip load shown in the figure, one performance constraint on the design may be the maximum allowable transverse displacement at the tip, while another constraint may be the maximum allowable bending stress. The design-related and performance-related constraints place limits on the optimizer's ability to reduce the weight.
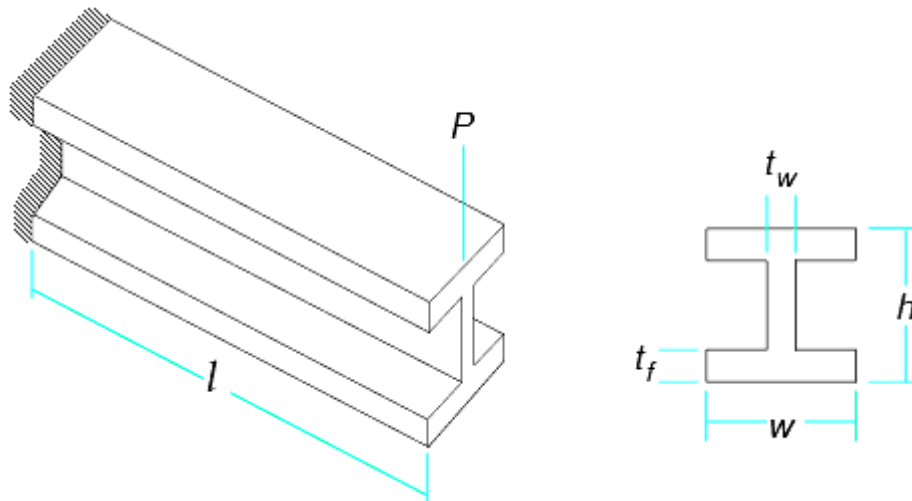
Figure 11-6. Cantilever I-Beam Example

The optimizer in NX Nastran uses sensitivity analysis. The sensitivity matrix [S] defines the change in response for a perturbation in design variable. A term in [S] is given by

$$S_{ij} = \frac{\partial R_i}{\partial X_j}$$

Equation 11-4.

where:

S_{ij}	=	i, j term in [S]
∂	=	partial derivative
R_i	=	i-th response
X_j	=	j-th design variable

The sensitivity is depicted in Figure 11-7, in which the sensitivity is the slope of the curve of response versus the design variable value. Design optimization and design sensitivity are described in greater depth in the *NX Nastran Design Sensitivity and Optimization User's Guide*.
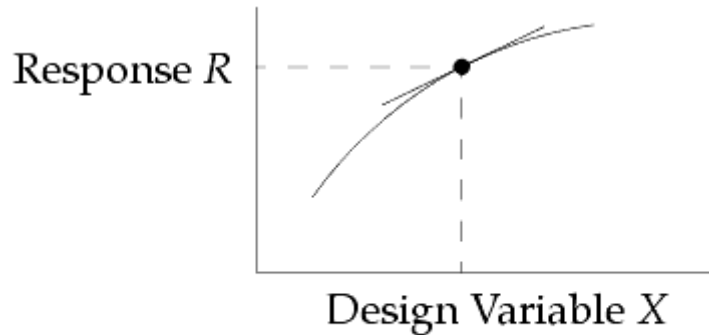
Figure 11-7. Response Sensitivity

11.11   Control System Analysis

A control system provides feedback (output) to an input. In aircraft, sample control systems are flap settings and actuator positions.

A control system involves a general input-output relationship called a transfer function. Transfer functions are represented in NX Nastran by the TF Bulk Data entry. Transfer functions can utilize grid points, extra points (EPOINT), and scalar points (SPOINT). Nonlinearities can be simulated via the NOLINi entries with which nonlinear transient loads are expressed as functions of displacements or velocities. Complex eigenvalue analysis can be used to determine stability when control systems include damping and unsymmetrical matrices.

11.12   Aeroelastic Analysis

NX Nastran provides efficient solutions of the problems of aeroelasticity, which is a branch of applied mechanics that deals with the interaction of aerodynamic, inertial, and structural forces. It is important in the design of airplanes, helicopters, missiles, suspension bridges, tall chimneys, and power lines. Aeroservoelasticity is a variation in which the interaction of automatic controls requires additional consideration.

The primary concerns of aeroelasticity include flying qualities (that is, stability and control), flutter, and structural loads arising from maneuvers and atmospheric turbulence. Methods of aeroelastic analysis differ according to the time dependence of the inertial and aerodynamic forces that are involved. For the analysis of flying qualities and maneuvering loads wherein the

aerodynamic loads vary relatively slowly, quasi-static methods are applicable. The remaining problems are dynamic, and the methods of analysis differ depending on whether the time dependence is arbitrary (that is, transient or random) or simply oscillatory in the steady state.

NX Nastran considers two classes of problems in dynamic aeroelasticity: aerodynamic flutter and dynamic aeroelastic response.

Aerodynamic Flutter

Flutter is the oscillatory aeroelastic instability that occurs at some airspeed at which the energy extracted from the airstream during a period of oscillation is exactly dissipated by the hysteretic damping of the structure. The motion is divergent in a range of speeds above the flutter speed. Flutter analysis utilizes complex eigenvalue analysis to determine the combination of airspeed and frequency for which the neutrally damped motion is sustained (see Figure 11-8).
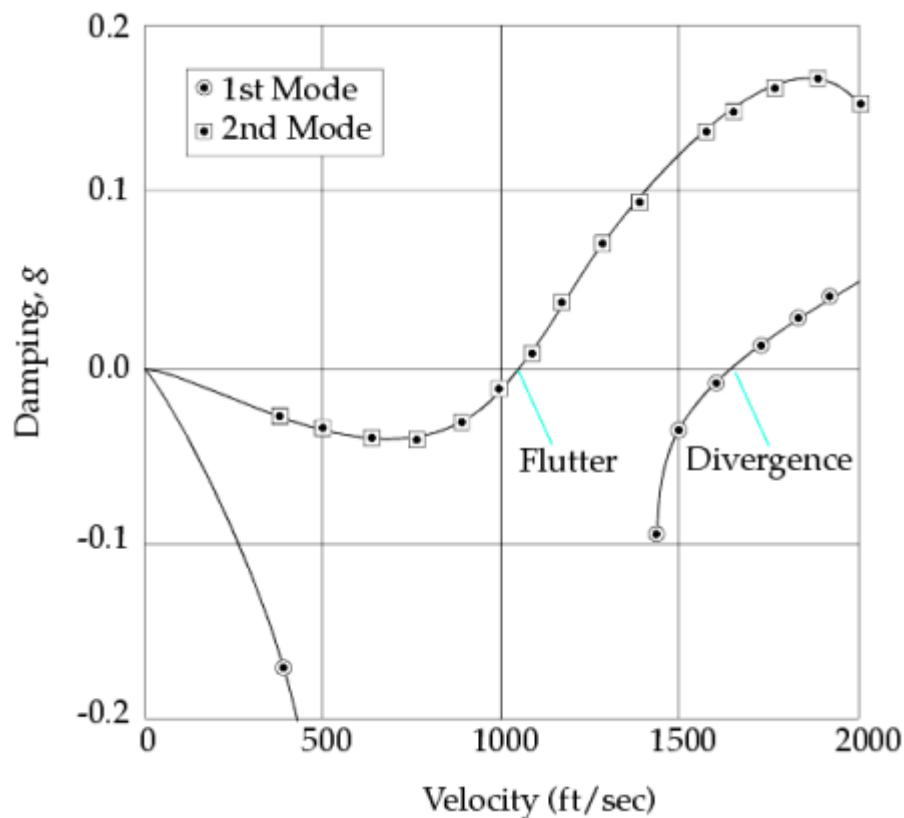
Figure 11-8. Flutter Stability Curve

Three methods of flutter analysis are provided in NX Nastran: the American flutter method (called the K method in NX Nastran), an efficient K method (called the KE method) for rapid flutter evaluations, and the British flutter method (called the PK method) for more realistic representation of the unsteady aerodynamic influence as frequency-dependent stiffness and damping terms. Complex eigenvalue analysis is used with the K method, and the QR transformation method is used with the KE and PK methods.

Dynamic Aeroelastic Response

The aeroelastic dynamic response problem determines the response of the aircraft to time- or frequency-varying excitations. Atmospheric turbulence is the primary example of this type of excitation, but wind shear and control surface motion can also have an aeroelastic component. Methods of generalized harmonic (Fourier) analysis are applied to the linear system to obtain the response to the excitation in the time domain. The gust response analysis may be regarded either as a stationary random loading or as a discrete gust.

The gust analysis capability computes the response to random atmospheric turbulence, discrete one-dimensional gust fields, and control surface motion and other dynamic loading. The random response parameters calculated are the power spectral density, root mean square response, and mean frequency of zero-crossings. The response to the discrete gust and control surface motion is calculated by direct and inverse Fourier transform methods since the oscillatory aerodynamics are known only in the frequency domain. The time histories of response quantities are the output in the discrete case (see Figure 11-9).
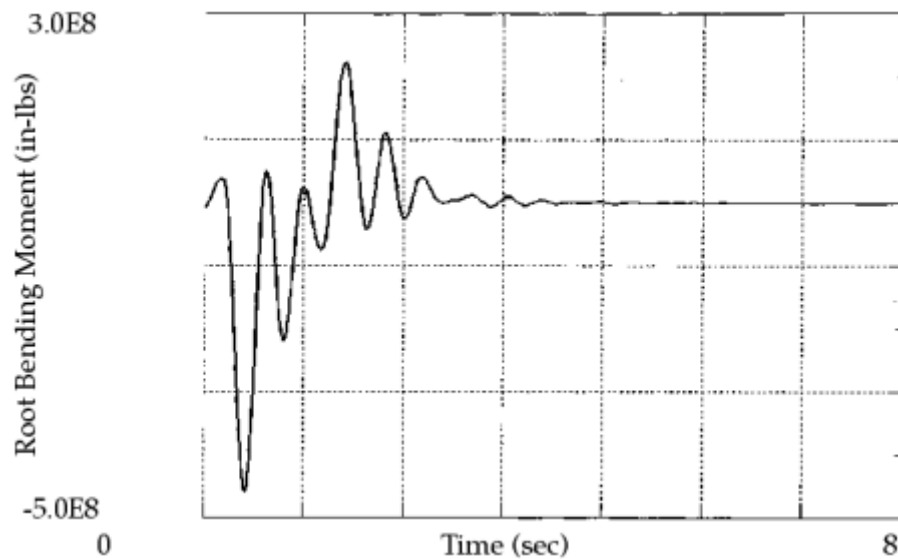
Figure 11-9. Transient Response Resulting from a Gust

11.13   DMAP

NX Nastran provides a series of solution sequences (SOL 103 for normal modes analysis, for example) written in their own language tailored to matrix manipulation. These solution sequences consist of a series of DMAP (direct matrix abstraction program) statements. NX Nastran's DMAP capability enables you to modify these solution sequences and write your own solution sequences.

DMAP is a high-level language with its own compiler and grammatical rules. DMAP statements contain data blocks and parameters and operate on them in a specified manner. For example, the DMAP statement

```
ADD  U1,U2/U3 $
```

adds matrices U1 and U2 together and calls the output U3. The DMAP statement

```
MATPRN  U3// $
```

prints the matrix U3.

Numerous DMAP Alters for dynamic analysis are provided (see the *NX Nastran Release Guide* for further information). Many of these DMAP Alters are for dynamic analysis. There are alters for model checkout, modal initial conditions, the addition of static results to transient response results, frequency-dependent impedance, and modal test-analysis correlation, among others. The *NX Nastran Advanced Dynamic Analysis User's Guide* describes a DMAP Alter for enforced motion using the Lagrange multiplier approach.

In summary, DMAP is a powerful capability that allows you to modify NX Nastran to satisfy your needs.

Appendix

A   Glossary of Terms

CMS	Component mode synthesis.
Critical Damping	The lowest value of damping for which oscillation does not occur when the structure is displaced from its rest position. Values of damping less than the critical damping value create an underdamped system for which oscillatory motion occurs.
Damping	Energy dissipation in vibrating structures.
DOF	Degree-of-freedom.
Dynamic Amplification Factor	Ratio of dynamic response to static response, which is a function of the forcing frequency, natural frequency, and damping.
Forced Vibration Analysis	Vibration response due to applied time-varying forces.
Free Vibration Analysis	Vibration response when there is no applied force. Normal modes analysis and transient response to initial conditions are examples of free vibration analysis.
Frequency Response Analysis	Computation of the steady-state response to simple harmonic excitation.
LMT	Lagrange multiplier technique.
Mechanism Mode	Stress-free, zero-frequency motions of a portion of the structure. A mechanism can be caused by an internal hinge.
Mode Shape	The deformed shape of a structure when vibrating at one of its natural frequencies.
Natural Frequency	The frequency with which a structure vibrates during free vibration response. Structures have multiple natural frequencies.
Phase Angle	The angle between the applied force and the response. In NX Nastran this angle is a phase lead, whereby the response leads the force.
Positive Definite	A matrix whose eigenvalues are all greater than zero.
Positive Semi-Definite	A matrix whose eigenvalues are greater than or equal to zero.
Repeated Roots	Two or more identical natural frequencies.
Resonance	Large-amplitude vibrations that can grow without bound. At resonance, energy is added to the system. This occurs, for example, when the harmonic excitation frequency is equal to one of the natural frequencies. The response at resonance is controlled entirely by damping.

Rigid-Body Drift	Transient displacements of an unconstrained model that grow continuously with time. This is often caused by the accumulation of small numerical errors when integrating the equations of motion.
Rigid-Body Mode	Stress-free, zero-frequency motions of the entire structure.
SDOF	Single degree-of-freedom.
Structural Damping	Damping that is proportional to displacement.
Transient Response Analysis	Computation of the response to general time-varying excitation.
Viscous Damping	Damping that is proportional to velocity.

Appendix

B   Nomenclature for Dynamic Analysis

The appendix provides nomenclature for terms commonly used in dynamic analysis.

B.1   General

Þ	Multiplication
a	Approximately
[ ]	Matrix
{ }	Vector
i	$\sqrt{-1}$
g	Acceleration of Gravity
Δt	Time Step
Δf	Frequency Step
i, j, k	Subscripts (Indices)
∞	Infinity
u	Displacement
$u(0)$	Initial Displacement
$\dot{u}$	Velocity
$\dot{u}(0)$	Initial Velocity
$\ddot{u}$	Acceleration
m	Mass
m_0	Large Mass

b	Damping
b_{cr}	Critical Damping
k	Stiffness
p	Applied Force
ω	Circular Frequency
ω_n	Circular Natural Frequency
ω^2	Eigenvalue
λ	Eigenvalue
f	Frequency
f_n	Natural Frequency
T_n	Period
ω_d	Damped Circular Natural Frequency
ζ	Damping Ratio
Q	Quality Factor
θ	Phase Angle
Δ	Logarithmic Decrement

B.2 Structural Properties

Geometry

L	Length
A	Area

Stiffness

E	Young's Modulus
G	Shear Modulus
J	Torsional Constant
ν	Poisson's Ratio
I	Area Moment of Inertia

Mass

ρ	Mass Density
ρ_m	Mass Density
ρ_w	Weight Density
I_p	Polar Moment of Inertia

Damping

G	Overall Structural Damping Coefficient
G_E	Element Structural Damping Coefficient

B.3   Multiple Degree-of-Freedom System

$\{u\}$	Displacement Vector
$\{\dot{u}\}$	Velocity Vector
$\{\ddot{u}\}$	Acceleration Vector
$\{\phi\}$	Mode Shape Vector
$\{\phi_{rig}\}$	Rigid-Body Modes
ξ_i	i-th Modal Displacement
$\dot{\xi}_i$	i-th Modal Velocity

$\ddot{\xi}_i$	i-th Modal Acceleration
m_j	j-th Generalized Mass
k_j	j-th Generalized Stiffness
λ_k	k-th Eigenvalue
$[M]$	Matrix
$[M_r]$	Rigid-Body Mass Matrix
$[K]$	Stiffness Matrix
$[K_E]$	Element Stiffness Matrix
$[B]$	Damping Matrix
$[P]$	Force Matrix

Appendix

C   The Set Notation System Used in Dynamic Analysis

The set notation system used for dynamic analysis in NX Nastran continues and expands the set notation system for static analysis. Because of the great variety of physical quantities and displacement sets used in dynamic analysis, you should understand the set notation system in NX Nastran.

C.1   Displacement Vector Sets

When you create a model in NX Nastran, equations are allocated for the purpose of assembling the necessary structural equations. These equations are rows and columns in the matrix equations that describe the structural behavior. Six equations are created per grid point, one equation is created per SPOINT or EPOINT. The model details (elements, properties, loads, etc.) are used to create the appropriate row and column entries in the matrices (e.g., stiffness coefficients). Certain data entries (i.e., SPCs, MPCs, ASETs, etc.) cause matrix operations to be performed in the various stages of the solution process. To organize the matrix operations, NX Nastran assigns each DOF to displacement sets.

Most matrix operations used in a structural analysis involve partitioning, merging, and/or transforming matrix arrays from one subset to another. All components of motion of a given type form a vector set which is distinguished from other vector sets. A given component of motion can belong to many sets.

In NX Nastran, there are two basic types of sets:

*   combined sets

*   mutually exclusive sets

Combined sets are formed through the union (i.e., combination) of two or more sets. Mutually exclusive sets are important in the solution process because if a DOF is a member of a mutually exclusive set, it cannot be a member of any of the other mutually exclusive sets. The importance of this property is demonstrated as the sets are described below.

When NX Nastran starts to assemble the structural equations, it allocates six equations for each grid point (GRID) and 1 equation for each scalar point (SPOINT). These equations are associated with a displacement set defined as the g-set. Fundamentally, the g-set represents an unconstrained set of structural equations.

The next step in the solution process is to partition the g-set into two subsets: the m-set and the n-set. The dependent DOFs of all multipoint constraint relations (MPCs, RBEs, etc.) that define the m-set are condensed into a set of independent DOFs (the n-set). The n-set represents all of the independent DOFs that remain after the dependent DOFs were removed from the active set of equations.

Using the n-set, SPCs are applied to the independent equations to further partition the equations. The degrees-of-freedom defined on SPC entries define the DOFs in the s-set. When you reduce the n-set by applying the s-set constraints, the f-set remains. The f-set is the "free" DOF of the structure. At this stage of the solution, the f-set is comprised of the remaining equations that represent a constrained structure. If the applied constraints are applied properly, the f-set equations represent a statically stable solution.

If static condensation is to be performed, the f-set is partitioned into the o-set and the a-set. The o-set degrees-of-freedom are those that are to be eliminated from the active solution through a static condensation. The remaining DOFs reside in the a-set. The a-set is termed the "analysis" set. The a-set is often the partition at which the solution is performed.

If the SUPORT entry is used, the degrees-of-freedom defined on the SUPORT entry are placed in the r-set. When the r-set is partitioned from the t-set, the l-set remains. This final set is termed the "leftover" set and is the lowest level of partitioning performed in NX Nastran static analysis. The *l*-set partition is the matrix on which the final solution is performed. Under special circumstances, the *l*-set is divided into two types of DOFs. The DOFs that are held fixed in component mode synthesis (CMS) are called b-set points, and those DOFs that are free to move in CMS are called c-set points.

C-1 demonstrates the basic partitioning operations. When a particular set above has no DOFs associated with it, it is a null set. Its partition is then applied, and the DOFs are moved to the subsequent partition.

Table C-1. Basic Partitioning Operations
g-set – m-set = n-set
n-set – s-set = f-set
f-set – o-set = a-set
a-set – r-set = *l*-set

In the above table, the m-, s-, o-, and r-sets form the mutually exclusive sets. Physically, the mutually exclusive set partitioning ensures that operations cannot be performed on DOFs that are no longer active. For example, if you apply an SPC to a DOF which is a dependent degree-of-freedom on an RBAR, a FATAL error is issued. Using an SPC entry moves a DOF to the s-set, but this cannot occur if the same DOF is already a member of the m-set. Both the m-set and s-set are mutually exclusive.

Table C-1 is a representation of all sets, set partitions, and set combinations used in NX Nastran.

A number of additional, mutually independent sets of physical displacements (namely, q-set, c-set, b-set and e-set) are used in dynamic analysis to supplement the sets used in static analysis. The q-, c-, and b-sets facilitate component mode synthesis. The e-set is used to represent control systems and other nonstructural physical variables. The p-set is created by combining the g-set with the e-set variables.

In addition to the combined sets described above, the v-set is a combined set created by combining the c-, r- and o-sets. The DOFs in these sets are the DOFs free to vibrate during component mode synthesis.

Some additional sets (sa-, k-, ps-, and pa-sets) represented in Figure C-1 are defined and used in aeroelastic analysis.

The modal coordinate set x is separated into zero frequency modes x_o and elastic (finite frequency) modes x_f. For dynamic analysis by the modal method, the extra points u_e are added to the modal coordinate set x_i to form the h-set as shown in Figure C-2.

The parameter PARAM,USETPRT can be used to print of lists of degrees-of-freedom and the sets to which they belong. Different values of the PARAM create various tables in the printed output.

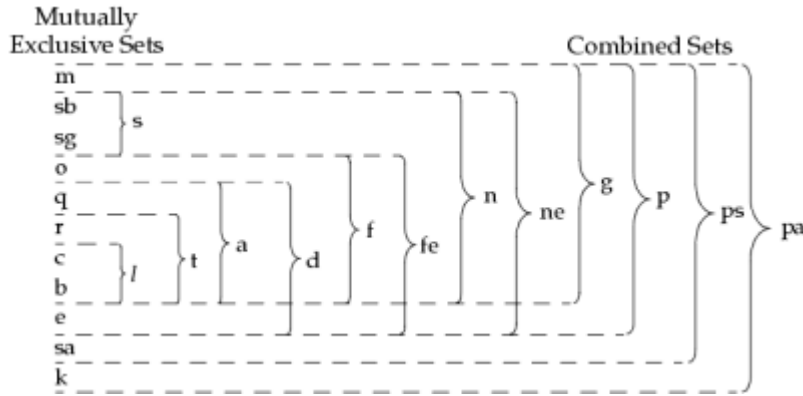The supersets formed by the union of other sets have the following definitions.

Figure C-1. Combined Sets Formed from Mutually Exclusive Sets

The set names have the definitions described in Table C-2.

Table C-2. Sets in NX Nastran	
Mutually Exclusive Sets	
Set Name	**Meaning**
m	Points eliminated by multipoint constraints.
sb[1]	Points eliminated by single-point constraints that are included in boundary condition changes and by the automatic SPC feature.
sg[1]	Points eliminated by single-point constraints that are listed on grid point Bulk Data entries.
o	Points omitted by structural matrix partitioning.
q	Generalized coordinates for dynamic reduction or component mode synthesis.
r	Reference points used to determine free body motion.
c	The free boundary set for component mode synthesis or dynamic reduction.
b	Coordinates fixed for component mode analysis or dynamic reduction.
e	Extra degrees-of-freedom introduced in dynamic analysis.
sa	Permanently constrained aerodynamic points.
k	Aerodynamic points.

[1]Strictly speaking, sb and sg are not exclusive with respect to one another. Degrees-of-freedom may exist in both sets simultaneously. These sets are exclusive, however, from the other mutually exclusive sets.

Table C-3. Sets in NX Nastran

Combined Sets (+ Indicates Union of Two Sets)	
Set Name	**Meaning**
$s = sb + sg$	All points eliminated by single-point constraints.
$l = b + c$	The structural coordinates remaining after the reference coordinates are removed (points left over).
$t = l + r$	The total set of physical boundary points for superelements.
$a = t + q$	The set assembled in superelement analysis.
$d = a + e$	The set used in dynamic analysis by the direct method.
$f = a + o$	Unconstrained (free) structural points.
$fe = f + e$	Free structural points plus extra points.
$n = f + s$	All structural points not constrained by multipoint constraints.
$ne = n + e$	All structural points not constrained by multipoint constraints plus extra points.
$g = n + m$	All structural (grid) points including scalar points.
$p = g + e$	All physical points.
$ps = p + sa$	Physical and constrained aerodynamic points.
$pa = ps + k$	Physical set for aerodynamics.
$fr = o + l$	Statically independent set minus the statically determinate supports ($o + l = f - q - r$).
$v = o + c + r$	The set free to vibrate in dynamic reduction and component mode synthesis.
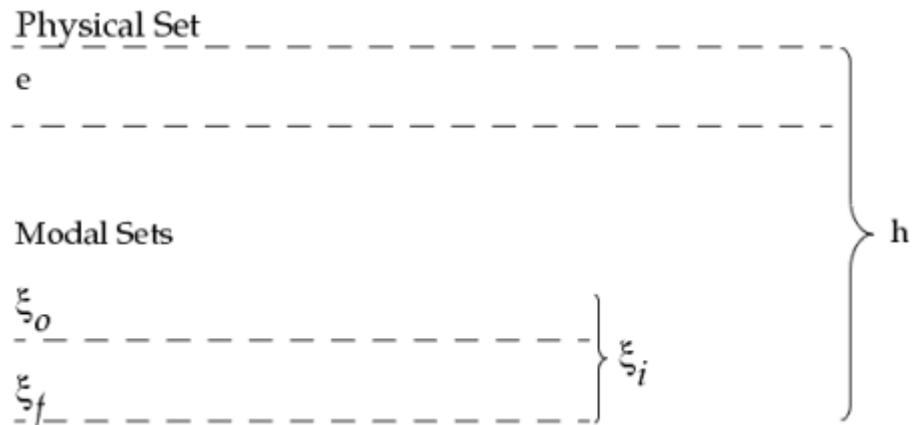
where:

x_0	=	rigid-body (zero frequency) modal coordinates
x_f	=	finite frequency modal coordinates
x_i	=	$x_0 + x_f$ = the set of all modal coordinates
u_h	=	$x_i + u_e$ = the set used in dynamic analysis by the modal method

Figure C-2. Set Notation for Physical and Modal Sets

Appendix

D   Common Commands for Dynamic Analysis

D.1   Solution Sequences for Dynamic Analysis

This section lists the solution sequences (SOLs) for the dynamic analysis types described in this guide. The SOLs are listed for the (preferred) structured solution sequences, rigid formats, and unstructured superelement solution sequences.

The NX Nastran Quick Reference Guide lists all of the solution sequences for all of the NX Nastran analysis types.

Structured Solution Sequences for Basic Dynamic Analysis

SOL Number	SOL Name	Description
103	SEMODES	Normal Modes
108	SEDFREQ	Direct Frequency Response
109	SEDTRAN	Direct Transient Response
111	SEMFREQ	Modal Frequency Response
112	SEMTRAN	Modal Transient Response

D.2   Case Control Commands for Dynamic Analysis

This section lists the Case Control commands that are often used for dynamic analysis. Commands that apply to statics, such as FORCE and STRESS, are not listed. The dynamic analysis Case Control commands are listed alphabetically. The description of each command is similar to that found in the *NX NASTRAN Quick Reference Guide*. The descriptions in this guide have been edited to apply specifically to the dynamic analysis capabilities described herein.

The *NX NASTRAN Quick Reference Guide* describes all of the Case Control commands.

The Case Control commands described in this appendix are summarized as follows:

Input Specification

B2GG	Direct Input Viscous Damping Matrix Selection
K2GG	Direct Input Stiffness Matrix Selection
K42GG	Direct Input Structural Damping Matrix Selection
M2GG	Direct Input Mass Matrix Selection

Analysis Specification

BC	Boundary Condition Identification
DLOAD	Dynamic Load Set Selection
FREQUENCY	Frequency Set Selection
IC	Transient Initial Condition Set Selection
METHOD	Real Eigenvalue Extraction Method Selection
SDAMPING	Structural Damping Selection
SUPORT1	Fictitious Support Set Selection
TSTEP	Transient Time Step Set Selection for Linear Analysis

Output Specification

ACCELERATION	Acceleration Output Request
DISPLACEMENT	Displacement Output Request
MODES	Subcase Repeater
OFREQUENCY	Output Frequency Set
OLOAD	Applied Load Output Request
OTIME	Output Time Set
SACCELERATION	Solution Set Acceleration Output Request
SDISPLACEMENT	Solution Set Displacement Output Request
SVECTOR	Solution Set Eigenvector Output Request
SVELOCITY	Solution Set Velocity Output Request
VELOCITY	Velocity Output Request

D.3   Bulk Data Entries for Dynamic Analysis

This appendix lists the Bulk Data entries that are often used for dynamic analysis. Entries that apply to generic modeling or statics, such as GRID or FORCE, are not listed. Bulk Data entries are listed alphabetically. The description of each entry is similar to that found in the NX Nastran Quick Reference Guide. The descriptions in this guide have been edited to apply specifically to the dynamic analysis capabilities described herein.

The *NX NASTRAN Quick Reference Guide* describes all of the Bulk Data entries.

The Bulk Data entries described in this appendix are summarized as follows:

Mass Properties

CMASS1	Scalar Mass Connection
CMASS2	Scalar Mass Property and Connection
CMASS3	Scalar Mass Connection to Scalar Points Only
CMASS4	Scalar Mass Property and Connection to Scalar Points Only
CONM1	Concentrated Mass Element Connection, General Form
CONM2	Concentrated Mass Element Connection, Rigid Body Form

PMASS   Scalar Mass Property

Damping Properties

CDAMP1   Scalar Damper Connection
CDAMP2   Scalar Damper Property and Connection
CDAMP3   Scalar Damper Connection to Scalar Points Only
CDAMP4   Scalar Damper Property and Connection to Scalar Points Only
CVISC   Viscous Damper Connection
PDAMP   Scalar Damper Property
PVISC   Viscous Element Property
TABDMP1   Modal Damping Table

Normal Modes

EIGR   Real Eigenvalue Extraction Data
EIGRL   Real Eigenvalue Extraction Data, Lanczos Method

Dynamic Loading

DAREA   Dynamic Load Scale Factor
DELAY   Dynamic Load Time Delay
DLOAD   Dynamic Load Combination or Superposition
DPHASE   Dynamic Load Phase Lead
LSEQ   Static Load Set Definition
TABLED1   Dynamic Load Tabular Function, Form 1
TABLED2   Dynamic Load Tabular Function, Form 2
TABLED3   Dynamic Load Tabular Function, Form 3
TABLED4   Dynamic Load Tabular Function, Form 4

Frequency Response

FREQ   Frequency List
FREQ1   Frequency List, Alternate Form 1
FREQ2   Frequency List, Alternate Form 2
FREQ3   Frequency List, Alternate Form 3
FREQ4   Frequency List, Alternate Form 4
FREQ5   Frequency List, Alternate Form 5
RLOAD1   Frequency Response Dynamic Load, Form 1
RLOAD2   Frequency Response Dynamic Load, Form 2

Transient Response

TIC	Transient Initial Condition
TLOAD1	Transient Response Dynamic Load, Form 1
TLOAD2	Transient Response Dynamic Load, Form 2
TSTEP	Transient Time Step

Miscellaneous

DMIG	Direct Matrix Input at Points
SUPORT	Fictitious Support
SUPORT1	Fictitious Support, Alternate Form

D.4   Parameters for Dynamic Analysis

This section lists some of the parameters that are often used for dynamic analysis. These parameters are listed alphabetically, beginning below. See the NX Nastran Quick Reference Guide for a description of all parameters.

ASlNG	Default = 0 ASING specifies the action to take when singularities (null rows and columns) exist in the dynamic matrices (or $[K_{ll}]$ in statics). If ASING = –1, then a User Fatal Message will result. If ASING = 0 (the default), singularities are removed by appropriate techniques depending on the type of solution being performed.
AUTOSPC	Default = YES (in SOLutions 101 through 200 except 106 and 129) Default = NO (in all other SOLutions) AUTOSPC specifies the action to take when singularities exist in the stiffness matrix ($[K_{gg}]$). AUTOSPC = YES means that singularities will be constrained automatically. AUTOSPC = NO means that singularities will not be constrained. Singularity ratios smaller than PARAM,EPPRT (default = 1.E-8) are listed as potentially singular. If PARAM,AUTOSPC has the value YES, identified singularities with a ratio smaller than PARAM,EPZERO (default = 1.E-8) will be automatically constrained with single-point constraints. If PARAM,EPPRT has the same value as PARAM,EPZERO (the default case), all singularities are listed. If PARAM,EPPRT is larger than PARAM,EPZERO, the printout of singularity ratios equal to exactly zero is suppressed. If PARAM,PRGPST is set to NO (default is YES), the printout of singularities is suppressed, except when singularities are not going to be removed. If PARAM,SPCGEN is set to 1 (default = 0), the automatically generated SPCs are placed in SPCi Bulk Data entry format on the PUNCH file. AUTOSPC provides the correct action for superelements in all contexts. It does not provide the correct action for the residual structure in SOL 129. PARAM,AUTOSPCR, not AUTOSPC, is used for the o-set (omitted set) in the residual structure in SOL 106.

BAILOUT	Default = $\begin{cases} 0 \text{ (for superelement sequences)} \\ -1 \text{ (for non-superelement sequences)} \end{cases}$ See MAXRATIO.
CB1, CB2	Default = 1.0 CB1 and CB2 specify factors for the total damping matrix. The total damping matrix is

$$[B_{jj}] \;=\; \text{CB1} \cdot [B_{jj}^{x}] + \text{CB2} \cdot [B_{jj}^{2}]$$

where $[B^2{}_{jj}]$ is selected via the Case Control command B2GG and $[B^x{}_{jj}]$ comes from CDAMPi or CVISC element Bulk Data entries. These parameters are effective only if B2GG is selected in the Case Control Section.

CK1, CK2	Default = 1.0 CK1 and CK2 specify factors for the total stiffness matrix. The total stiffness matrix (exclusive of GENEL entries) is

$$[K_{jj}^{x}] \;=\; \text{CK1} \cdot [K_{jj}^{2}] + \text{CK2} \cdot [K_{jj}^{2}]$$

where $[K^2{}_{jj}]$ is selected via the Case Control command K2GG and $[K^z{}_{jj}]$ is generated from structural element (e.g., CBAR) entries in the Bulk Data. These are effective only if K2GG is selected in the Case Control Section. Note that stresses and element forces are not factored by CK1, and must be adjusted manually.

CK41, CK42	Default =(1.0, 0.0) CK41 and CK42 specify factors for the total structural damping matrix. The total structural damping matrix is

$$\left[K_{jj}^{4,x} \right] = \text{CK41} \cdot \left[K_{jj}^{4,z} \right] + \text{CK42} \cdot \left[K_{jj}^{4,2} \right]$$

where $[K^{4,2}{}_{jj}]$ is selected via the Case Control command K42GG and $[K^{4,z}{}_{jj}]$ is generated from the stiffness of structural element (e.g., CBAR) entries in the Bulk Data times the structural damping coefficient GE on material entries (e.g. MAT1). These are effective only if K42GG is selected in Case Control. Note that stresses and element forces are not factored by CK41, and must be adjusted manually.

CM1, CM2	Default = 1.0 CM1 and CM2 specify factors for the total mass matrix. The total mass matrix is

$$[M_{jj}] \;=\; \text{CM1} \cdot [M_{jj}^{x}] + \text{CM2} \cdot [M_{jj}^{2}]$$

where $[M^2{}_{jj}]$ is selected via the Case Control command M2GG and $[M^x{}_{jj}]$ is derived from the mass element entries in the Bulk Data Section. These are effective only if M2GG is selected in the Case Control Section.

COUPMASS	Default = -1

	COUPMASS > 0 requests the generation of coupled rather than lumped mass matrices for elements with coupled mass capability. This option applies to both structural and nonstructural mass for the following elements: CBAR, CBEAM, CHEXA, CONROD, CPENTA, CQUAD4, CQUAD8, CROD, CTETRA, CTRIA3, CTRlA6, CTRIAX6, CTUBE. A negative value (the default) causes the generation of lumped mass matrices (translational components only) for all of the above elements.
CP1, CP2	Default = 1.0
	The load vectors are generated from the equation

$$\{P_j\} \;=\; \mathrm{CP1} \cdot \left\{ P_j^x \right\} + \mathrm{CP2} \cdot \left\{ P_j^2 \right\}$$

	where $\{P^2_j\}$ is selected via the Case Control command P2G, and $\{P^x_j\}$ comes from Bulk Data static load entries.
CURVPLOT	Default = -1
	PARAM,CURVPLOT,1 requests that X-Y (or curve) plots be made for displacements, loads, SPC forces, or grid point stresses or strains. The y values are response values; the x values are related to grid point locations through the parameter DOPT. PARAM,CURVPLOT,1 suppresses SORT2-type processing; in superelement dynamic analysis, SORT1 requests will be honored. To obtain stress or strain plots, set the CURV parameter to +1. DOPT controls the x spacing of curves over grid points for the CURVPLOT module. The default for DOPT is the length between grid points.
DDRMM	Default = 0
	By default, the matrix method of data recovery is used in the modal solutions. DDRMM = –1 will force calculation of complete g-set solution vectors by the mode displacement method, which is needed for SORT1 output. SORT1 output is required for deformed structure plots, grid point force balance output, the mode acceleration technique, and postprocessing with PARAM,POST,–1 or –2.
DYNSPCF	Default = NEW (Structured SOLs only)
	PARAM,DYNSPCF,NEW requests that mass and damping coupled to ground be included in the SPC Force calculations for the linear dynamic solutions: SOLs 103, 107-112, 115, 118, 145, 146, and 200. OLD neglects these effects and gives the same SPC Force results prior to the new algorithm.
EPZERO	Default = 1.E-8
	Specifies the minimum value that indicates a singularity. See AUTOSPC.
G	Default = 0.0
	G specifies the uniform structural damping coefficient in the formulation of dynamics problems. To obtain the value for the parameter G, multiply the critical damping ratio, C/Co, by 2.0. PARAM,G is not recommended for use in hydroelastic or heat transfer problems. If PARAM,G is used in transient analysis, PARAM,W3 must be greater than zero or PARAM,G will be ignored.
HFREQ	Default = 1 .+30

The parameters LFREQ and HFREQ specify the frequency range (LFREQ is the lower limit and HFREQ is the upper limit) of the modes to be used in the modal formulations. Note that the default for HFREQ will usually include all vectors computed. A related parameter is LMODES.

KDAMP   Default = 1

If KDAMP is set to –1, viscous modal damping is entered into the complex stiffness matrix as structural damping.

LFREQ   Default = 0.0

See HFREQ.

LMODES   Default = 0

LMODES is the number of lowest modes to use in a modal formulation. If LMODES = 0, the retained modes are determined by the parameters LFREQ and HFREQ.

MAXRATIO   Default = 1.E5

The ratios of terms on the diagonal of the stiffness matrix to the corresponding terms on the diagonal of the triangular factor are computed. If, for any row, this ratio is greater than MAXRATIO, the matrix will be considered to be nearly singular (having mechanisms). If any diagonal terms of the factor are negative, the stiffness matrix is considered implausible (non-positive definite). The ratios greater than MAXRATIO and less than zero and their associated external grid identities will be printed out. The program will then take appropriate action as directed by the parameter BAILOUT.

By default, in the superelement solution sequences the program will terminate processing for that superelement. A negative value for BAILOUT directs the program to continue processing the superelement. Although forcing the program to continue with near-singularities is a useful modeling checkout technique, it may lead to solutions of poor quality or fatal messages later in the run. It is recommended that the default values be used for production runs. A related parameter is ERROR.

In nonsuperelement solution sequences, the default value (–1) of BAILOUT causes the program to continue processing with near singularities and a zero value will cause the program to exit if near singularities are detected.

In SOLs 101-200 when PARAM,CHECKOUT,YES is specified, PARAM,MAXRATIO sets the tolerance for detecting multipoint constraint equations with poor linear independence.

MODACC   Default = –1

MODACC = 0 selects the mode acceleration method for data recovery in dynamic analysis. PARAM,DDRMM,–1 must also be specified.

NONCUP   Default = –1

NONCUP selects either a coupled or noncoupled solution algorithm in modal transient response analysis. By default, the noncoupled solution algorithm is selected unless the dynamic matrices KHH, MHH, or BHH have off-diagonal terms. NONCUP = 1 requests the coupled algorithm and –2 the uncoupled algorithm regardless of the existence of off-diagonal terms in the dynamic matrices. User Information Message 5222 indicates which algorithm is used in the analysis.

NOSORT1

$$\text{Default} = \begin{cases} 1 \text{ (SOL 30)} \\ -1 \text{ (SOL 31)} \end{cases}$$

NOSORT1 controls the execution of the SDR3 module, which changes SORT1 modal solutions to SORT2. In SOL 30, where SORT1 output is the default option, NOSORT1 is set to branch over this operation. In SOL 31, where SORT2 output is usually desired, PARAM,NOSORT1,1 may be used when only SORT1 output is desired.

WTMASS

Default = 1.0

The terms of the structural mass matrix are multiplied by the value of this parameter when they are generated. This parameter is not recommended for use in hydroelastic problems.

W3, W4

Default = 0.0

The damping matrix for transient analysis is assembled from the equation:

$$[B_{dd}] = [B^1_{dd}] + [B^2_{dd}] + \frac{G}{W3}[K^1_{dd}] + \frac{1}{W4}[K^4_{dd}]$$

The default values of 0.0 for W3 and W4 cause the $[K^1_{dd}]$ and $[K^4_{dd}]$ terms to be ignored in the damping matrix, regardless of the presence of the PARAM,G or $[K^4_{dd}]$. $[K^1_{dd}]$ is the stiffness. $[K^4_{dd}]$ is the structural damping and is created when GE is specified on the MATi entries.

Appendix

E File Management Section

E.1 Overview

This appendix provides an overview of how NX Nastran's database structure and File Management Section (FMS) work so that you can allocate your computer resources efficiently, especially for large models. For many problems, due to the default values, the use or knowledge of the FMS is either transparent or not required by you.

E.2 Definitions

Before presenting the details of the database description, it is helpful to define some of the basic terms that are used throughout this appendix.

DBset	Database set. This consists of an NX Nastran logical name, such as MASTER, DBALL, etc., which refers to an entity within the NX Nastran database. It consists of a collection of physical files.
Database	Collection of all DBsets assigned to a run.
Data block	Matrix or table (e.g., KAA, LAMA) that is stored in the database.
Logical name	Local internal name (log-name) used by NX Nastran for a file or DBset.
Word	For 32- and 64-bit machines, each word is equivalent to four and eight bytes, respectively.
Buffsize	Length of an NX Nastran buffer in terms of words (32- or 64-bit words). It controls the physical record size for data storage/transfer that is contained in many NX Nastran logical units. The default and maximum allowable buffsize is machine dependent. The default value is recommended except for large problems. It can be modified by using the following NASTRAN statement:

```
NASTRAN BUFFSIZE = xxxxx
```

where xxxxx is the desired buffsize.

Block	This is often referred to as a NX Nastran GINO block. Each GINO block contains one NX Nastran buffer.
{ }	A brace indicates that the quantity within this bracket is mandatory. The underlined item within { } is the default value.
[ ]	A square bracket indicates that the quantity within this bracket is optional.

E.3   NX Nastran Database

When submitting a NX Nastran job, a series of standard output files is created (e.g., F06 file). Conventions for the filenames are machine dependent. Furthermore, four permanent (MASTER, DBALL, USRSOU, USROBJ) and one scratch (SCRATCH) DBsets are created during the run.

MASTER	This is the "directory" DBset that contains a list of all the DBsets used in the job, all the physical file names assigned, and an index pointing to all the data blocks created and where they are stored. In addition, it also contains the NX Nastran Data Definition Language (NDDL) used. NDDL is the internal NX Nastran language that describes the database. You do not need to understand NDDL to use NX Nastran effectively. The default maximum size for MASTER is 5000 blocks.
DBALL	This is the DBset where the permanent data blocks are stored by default. The default maximum size is 25000 blocks.
USRSOU	This DBset stores the source file for user-created DMAP. The default maximum size is 5000 blocks.
USROBJ	This DBset stores the object file for user-created DMAP. The default maximum size is 5000 blocks.
SCRATCH	This DBset is used as the temporary workspace for NX Nastran. In general, this DBset is deleted at the end of the run. The default maximum size is 350100 blocks.

For most solutions, USRSOU and USROBJ are not needed and may be deleted or assigned as temporary for the duration of the run (see "ASSIGN").

For a typical UNIX-based workstation with an NX Nastran input file called "dyn1.dat", the following sample submittal command can be used:

```
nastran dyn1 scr=no
```

where nastran is the name of shell script for executing NX Nastran. The following four physical database files are created as a result of the above command.

```
dyn1.MASTER
dyn1.DBALL
dyn1.USROBJ
dyn1.USRSOU
```

Unless otherwise stated, the input filename is assumed to be "dyn1.dat" in this appendix.

E.4   File Management Commands

The File Management Section is intended primarily for the attachment and initialization of NX Nastran database sets (DBsets) and FORTRAN files. For many problems, due to the default values, the use of the FMS is either transparent or not required by you. At first glance the FMS may be overwhelming due in part to its flexibility and the many options it has to offer. However, in its most commonly used form, it can be quite simple. Examples in the most commonly used format are presented throughout this appendix.

There are numerous FMS statements available in NX Nastran. This section covers the commonly used FMS statements in dynamic analysis. They are presented in their most commonly used format, which in general, is a simplified format.

If an FMS statement is longer than 72 characters, it can be continued to the next line by using a comma as long as the comma is preceded by one or more blank spaces.

INIT

Purpose

Creates/initializes permanent and/or temporary DBsets. The INIT statement has two basic formats: one for all the DBsets and one specifically for the SCRATCH DBsets.

Format (Simplified) for All DBsets Except SCRATCH

```
INIT DBset-name [LOGICAL=(log-name1(max-size1)   ,
     log-name2(max-size2),...lognamei(max-sizei)   ,
     ...lognamen(max-sizen)]
```

DBset-name	Logical name of the DBset being used (e.g., DBALL).
log-namei	i-th logical name for the DBset-name referenced in this INIT statement. You can have up to 10 logical names for each DBset ($1 \le i \le 10$). The i-th physical file is assigned with the "assignment" statement.
max-sizei	Maximum allowable number of NX Nastran blocks which may be written to the i-th file.

Example

```
INIT DBALL LOGICAL=(DBALL(50000))
```

This statement creates the DBALL DBset with a logical name of DBALL and the maximum size of 50000 NX Nastran blocks instead of 25000 blocks, which is the default value for DBALL. Unless an ASSIGN statement is also used, the physical file is given the name dyn1.DBALL assuming that your input file is called "dyn1.dat".

```
INIT DBALL LOGICAL=(DB1(35000),DBTWO(60000))
```

This statement creates the DBALL DBset with logical names of DB1 and DBTWO. Two physical files, dyn1.DB1 and dyn1.DBTWO, are created with a maximum size of 35000 and 60000 NX Nastran blocks, respectively.

Format (Simplified) for the SCRATCH DBset

```
INIT SCRATCH [LOGICAL=(log-name1(max-size1)   ,
   log-name2(max-size2),...log-namei(max-sizei))   ,
   SCR300=(log-namei+1(max-sizei+1),...log-namen(max-sizen))]
```

Log-name1 through log-namei are allocated for regular scratch files as temporary workspace. This temporary workspace is not released until the end of the job. SCR300 is a special keyword which indicates that the log-names are members reserved for DMAP module internal scratch files. The space occupied by these SCR300 files is for the duration of the execution of the module. This SCR300 space is released at the end of the module execution. You can have up to a combined total of 10 logical names for the SCRATCH DBset ($1 \le i \le 10$).

Example

```
INIT SCRATCH LOGICAL=(SCR1(150000),SCR2(100000)) ,
     SCR300=(SCRA(250000),SCRB(300000))
```

This statement creates the SCRATCH DBset with logical names of SCR1, SCR2, SCRA, and SCRB. Two physical files, dyn1.SCR1 and dyn1.SCR2, are created with a maximum size of

150000 and 100000 blocks, respectively. These two files are regular scratch files. Two additional physical files, dyn1.SCRA and dyn1.SCRB, are created with a maximum size of 250000 and 300000 blocks, respectively. These two files are SCR300 type files.

ASSIGN

Purpose

Assigns physical filenames to logical filenames or special FORTRAN files that are used by other FMS statements or DMAP modules.

Format (Simplified) to Assign Logical Files

```
ASSIGN log-namei='filenamei' [TEMP DELETE]
```

log-namei	The i-th logical name for the DBset created by the INIT statement.
TEMP	Requests that filenamei be deleted at the end of the job. This is optional and is often used for USRSOU and USROBJ.
DELETE	Requests that filenamei be deleted if it exists before the start of the run. This is optional; however, if this option is not used and the FORTRAN file exists prior to the current run, then the job may fail with the following messages:

```
*** USER FATAL MESSAGE 773 (DBDEF)
THE FOLLOWING PHYSICAL FILE ALREADY EXISTS
LOGICAL NAME = XXXX
PHYSICAL FILE = YYYY
USER INFORMATION: NO ASSOCIATED DEFAULT FILES OR ASSIGNED DBSETS CAN
EXIST PRIOR TO THE DATA BASE INITIALIZATION RUN,
USER ACTION: DELETE THIS FILE AND RESUBMIT THE JOB
```

DELETE is not a suggested option if you are using RESTART since you can delete your database inadvertently. Manual deletion of unwanted databases is a safer approach.

Example

```
ASSIGN DB1='sample.DB1'
INIT DBALL LOGICAL=(DB1(50000))
```

These statements create a physical file called sample.DB1 for the logical name DB1 in the current directory. Without the ASSIGN statement, the physical file name created is called dyn1.DB1, assuming once again that your input file is called dyn1.dat.

```
ASSIGN DB1='/mydisk1/se/sample.DB1'
ASSIGN DB2='/mydisk2/sample.DB2'
INIT DBALL LOGICAL=(DB1(50000),DB2(40000))
```

Two logical names DB1 and DB2 are created for the DBset DBALL. DB1 points to a physical file called sample.DB1 that resides in the file system (directory) /mydisk1/se. DB2 points to a physical file called sample.DB2 that resides in the file system (directory) /mydisk2.

Format (Simplified) to Assign FORTRAN Files

```
ASSIGN DB1='/mydisk1/se/sample.DB1'
ASSIGN DB2='/mydisk2/sample.DB2'
INIT DBALL LOGICAL=(DB1(50000),DB2(40000))
```

log-key	This is the logical keyword for the FORTRAN file being created. The default values depend on the keyword. Acceptable keywords are
	`DBC,DBMIG,INPUTT2,INPUTT4,OUTPUT2,OUTPUT4`
	`,`
	`DBUNLOAD,DBLOAD, and USERFILE.`

You should reference the *NX Nastran Quick Reference Guide* for detailed descriptions and the defaults for these keywords.

filenamef	This is the physical name of the FORTRAN file.
STATUS	Specifies whether the FORTRAN file will be created (STATUS = new) or is an existing file (STATUS = old).
UNIT	Specifies the FORTRAN unit (e.g., UNIT = 12).
FORM	Specifies whether the file written is in ASCII (FORM = FORMATTED) or binary (FORM = UNFORMATTED) format.
DELETE	Requests that filenamef be deleted if it exists before the start of the run.

Example

```
ASSIGN OUTPUT2='sample.out',STATUS=NEW,UNIT=11,FORM=FORMATTED,
     DELETE
```

This example creates a new FORTRAN file to be used for OUTPUT2 operations. This file is in ASCII format with a physical filename of sample.out and is assigned to unit 11.

EXPAND

Purpose

Concatenates files into an existing DBset in order to increase the allowable disk space. The EXPAND statement is normally used in a restart run when you run out of disk space in your previous run.

Format

```
EXPAND DBset-name LOGICAL=(log-namei(max-sizei),... )
```

DBset-name	Logical name of the DBset to be expanded by the addition of new members to this existing DBset.
log-namei	Logical name of the i-th member of the DBset. An ASSIGN statement should be used to point this logical name to a physical file.
max-sizei	Maximum allowable number of NX Nastran blocks that may be written to the i-th member.

Example

The original run creates a database with a name dyn1.DBALL. This database is filled and the job fails with the following error messages in the F06 file:

```
*** USER FATAL MESSAGE 1012 (GALLOC)
DBSET DBALL IS FULL AND NEEDS TO BE EXPANDED.
```

For small to medium size problems, it is best to rerun the job from the beginning with a larger file allocation. For large problems, if rerunning the job is not practical, then the database can be expanded with the following statements:

```
RESTART
ASSIGN MASTER='dyn1.MASTER'
ASSIGN DBADD='morespace.DB'
EXPAND DBALL LOGICAL=(DBADD(50000))
```

These statements create an additional member (with a logical name of DBADD) to the existing DBset DBALL. This member points to a new physical file called morespace.DB, which may contain up to a maximum of 50000 NX Nastran blocks. In this case, you are restarting from "dyn1.MASTER".

RESTART

Purpose

Allows you to continue from the end of the previous run to the current run without resolving the problem from the beginning.

Format

```
RESTART [PROJECT='proj-ID', VERSION={version-ID,LAST},
{KEEP,NOKEEP}]
```

proj-ID	Project identifier used in the original run, which can have up to 40 characters. This is optional and is normally not used. The default proj-ID is blank.
version-ID	The version number you are restarting from. The default is the last version.
KEEP	If this option is used, then the version that you are restarting from is also saved at the end of the current run.
NOKEEP	If this option is used, then the version that you are restarting from is deleted at the end of the current run. This is the default.

Example

```
RESTART
```

The current run uses the last version in the database for restart. At the end of the run, this last version is deleted from the database. This statement is probably the most commonly used format for RESTART.

```
RESTART VERSION=5,KEEP
```

The current run (version 6 or higher) uses version 5 in the database for restart. At the end of the run, version 5 is also retained in the database. This format is used most often when you want to ensure that a specific version is saved in the database (i.e., a large normal modes run).

```
RESTART PROJ='xyz' VERSION=3
```

The current run uses version 3 with a proj-ID of xyz in the database for restart. At the end of the run, version 3 with a proj-ID of xyz is deleted from the database.

INCLUDE

Purpose

Inserts an external file at the location where this "include" statement is used. This is not a pure FMS statement because it can be used anywhere in the input file, not just in the FMS Section. The "include" statement must not be nested; in other words, you cannot attach a file that contains an "include" statement.

Format

```
Include 'filename'
```

filename   Physical filename of the external file to be inserted at this location.

Example

```
Sol 101
time 10
cend
include 'sub1.dat'
begin bulk
$
include 'bulk1.dat'
include 'bulk2.dat'
$
$         rest of bulk data file
$
          .
          .
enddata
```

This run reads a file called sub1.dat with all the Case Control commands contained in it. It also reads two additional files (bulk1.dat and bulk2.dat) in the Bulk Data Section. You may, for example, want to include all your grid Bulk Data entries in the file bulk1.dat and all your element connectivities in bulk2.dat. As you can see, the "include" statement can be a handy tool. For parametric studies, you can potentially save a tremendous amount of disk space by using the "include" statement instead of having multiple files with duplicate input data.

Summary

Due to the default values, very little knowledge of the NX Nastran FMS statements and database structure is required for small to medium size problems. For large problems, however, some knowledge of the FMS statements and database structure can help you to optimize your computer resources.

Appendix

F   Numerical Accuracy Considerations

F.1   Overview

NX Nastran is an advanced finite element analysis program. Because speed and accuracy are essential, NX Nastran's numerical analysis capabilities are continually enhanced to provide the highest level of each.

This appendix provides a brief overview for detecting and avoiding numerical ill-conditioning problems, especially as they relate to dynamic analysis. For more information regarding NX Nastran's numerical analysis algorithms, see the NX Nastran Numerical Methods User's Guide.

F.2   Linear Equation Solution

The basic statement of the linear equation solution is

$$[A]\{x\} = \{b\}$$

Equation F-1.

where $[A]$ is a square matrix of known coefficients (and usually symmetric for structural models), $\{b\}$ is a known vector, and $\{x\}$ is the unknown vector to be determined.

The methods used for solution in NX Nastran are based on a decomposition of $[A]$ to triangular matrices and are followed by forward-backward substitution to get $\{x\}$. The equations for this solution technique are

$$[A] = [L][U]$$

Equation F-2.

where $[L]$ is a lower-triangular matrix and $[U]$ an upper-triangular matrix, and

$$[L]\{y\} = \{b\}$$

Equation F-3.

where $\{y\}$ is an intermediate vector. Equation F-3 is called the forward pass because the solution starts with the first row where there is only one unknown due to the triangular form of $[L]$. The backward pass starts with the last row and provides the solution

$$[U]\{x\} = \{y\}$$

Equation F-4.

F.3   Eigenvalue Analysis

The general eigensolution equation is

$$[K + pB + p^2 M]\{u\} = 0$$

Equation F-5.

where *p* is the complex eigenvalue. This equation can always be transformed to a special eigenvalue problem for a matrix [*A*]

$$[A - \lambda I]\{\phi\} = 0$$

Equation F-6.

where [*I*] is the identity matrix. Equation F-6 is the basis of all the transformation methods of NX Nastran (HOU, etc.). The iterative method (SINV) work directly from EquationF-5. The Lanczos method uses both. If [*A*] is a symmetric matrix, the eigenvectors are orthogonal, and they can be normalized such that

$$[\phi^T][\phi] = [I]$$

Equation F-7.

where $[\phi]$ is a square matrix whose columns contain the eigenvectors $\{\phi_i\}$. With this normalization convention, then

$$[\phi^T][A][\phi] = [\lambda]$$

Equation F-8.

and

$$[A] = [\phi][\lambda][\phi^T]$$

Equation F-9.

where [λ] is the eigenvalue diagonal matrix.

F.4 Matrix Conditioning

Reordering the previous equations, any matrix [*I*] can be expressed as a sum of its eigenvalues multiplied by dyadic eigenvector products

$$[A] = \lambda_1\{\phi_1\}\{\phi_1\}^T + \lambda_2\{\phi_2\}\{\phi_2\}^T + \ldots + \lambda_n\{\phi_n\}\left\{\phi_n^T\right\}$$

Equation F-10.

Defining $[B_i] = \{\phi_i\}\left\{\phi_i^T\right\}$, which is a full rank 1 matrix, then

$$[A] = \sum_{k=1}^{n} \lambda_k[B_k]$$

Equation F-11.

where *n* is the dimension of [*A*]. On the average, an element of [B_k] has the same magnitude as the corresponding element of [B_{k+1}]. Let B_{max} be the magnitude of the largest coefficient of all [B_k] matrices. Then

$$\left|a_{ij}\right| \le B_{max}(\lambda_1 + \lambda_2 + \ldots + \lambda_n)$$

Equation F-12.

This equation shows that the terms of [*A*] are dominated by the largest eigenvalues. Unfortunately, the smallest eigenvalues are those of greatest interest for structural models. These small eigenvalues must be calculated by taking the differences of coefficients that are dominated by the largest eigenvalue. For this reason, the ratio $|_n/|_1$ is called a numerical conditioning number. If this number is too large, numerical truncation causes a loss of accuracy when computing the lowest eigenvalues of a system.

The assumptions that allow this simple analysis are often pessimistic in practice, that is, the bounds predicted by the error analysis are conservative. However, the effects it predicts do occur eventually so that models that produce acceptable results for one mesh size may produce unacceptable results with a finer mesh size due to the higher eigenvalues included in the larger-sized matrices occurring from the finer mesh.

F.5 Definiteness of Matrices

A matrix whose eigenvalues are all greater than zero is said to be positive definite. If some eigenvalues are zero but none are less than zero, the matrix is positive semi-definite. A stiffness matrix assembled from elements is at least positive semi-definite. If all of the structure's

rigid-body modes are constrained, the stiffness matrix is positive definite. Another category is the indefinite matrices category. These matrices have zeroes or blocks of zeroes on the diagonal.

Although definiteness is most concisely defined in terms of eigenvalues, it is not a practical test for large matrices because of the computational cost of extracting all of the eigenvalues. However, other operations, particularly linear equation solution and dynamic reduction, may detect nonpositive definite matrices and provide diagnostics using these terms as described later in this appendix.

F.6 Numerical Accuracy Issues

The numerical operations of NX Nastran are executed in a finite 64-bit floating point arithmetic. Depending on the specific computer's word structure (number of bits for mantissa versus exponent), different roundoff errors may occur. The sophistication level of the actual hardware (or software) arithmetic units also has an influence on the numerical accuracy. To attain the most numerical accuracy possible, the following strategies are used in NX Nastran.

In the decomposition of positive definite matrices, the Gaussian elimination process (which is the basis of all decomposition algorithms in NX Nastran) does not require numerical pivots. Since some of the matrices are not positive definite, sparse decomposition (both symmetric and unsymmetric) employs a numerical pivoting (row/column interchange) algorithm. These methods consider a pivot term suitable if

$$\left| a_{kk} \right| \geq 10^{\text{THRESH}} \cdot \underset{i}{\text{MAX}} \left| a_{ki} \right|$$

Equation F-13.

which means that a diagonal term is accepted as a pivot if it is greater than the maximum term in that row multiplied by 10^{THRESH} (the default for THRESH is –6).

To ensure numerical accuracy in eigenvalue calculations, most NX Nastran methods use a spectral transformation of

$$\mu = \frac{1}{\lambda - \lambda_s}$$

Equation F-14.

where λ_s is an eigenvalue shift.

This transformation is shown in Figure F-1.
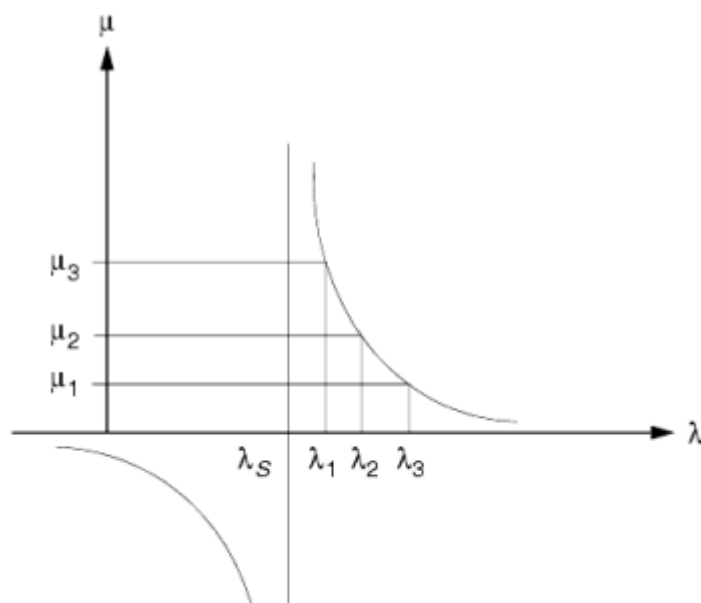
Figure F-1. Spectral Transformation

The spectral transformation ensures uniform accuracy throughout the frequency by "shifting" to the area of interest. Another effect of this transformation is the welcomed µ space separation of closely-spaced eigenvalues. When λ_s is close to an eigenvalue λ_i , the decomposition of the "shifted matrix"

$$[K - \lambda_s M] \; = \; [L][D][L^T]$$

Equation F-15.

may produce high MAXRATIO messages. Automatic logic to perturb the λ_s value in this case is implemented in NX Nastran.

F.7 Sources of Mechanisms

In all of the decomposition methods, a null row or column in [*A*] causes a fatal error message. All other causes of singularity are not distinguishable from near-singularity because of the effects of numerical truncation. Only warning messages are usually provided for these cases.

In standard decomposition

$$[A] \; = \; [L][D][L^T]$$

Equation F-16.

the process starts to compute the first term of [*D*] with the first internal degree of freedom and then processes each additional degree of freedom and its associated terms. It can be shown that when processing the k-th row, the k-th row and all rows above it are in effect free, and all rows below it are constrained to ground. The term of [*D*] at the k-th row is proportional to the amount of coupling between that degree of freedom and the degree-of-freedom with a higher value of k. If the terms of the k-th row and above are not connected to the remaining rows, the k-th term of

[D] goes to zero. Because of numerical truncation, the term may be a small positive or negative number instead. If a term of [D] is calculated to be identically zero, it is reset to a small number because of the indeterminacy of its calculation. The existence of such a small term defines a mechanism in static analysis in NX Nastran. A mechanism is a group of degrees-of-freedom that may move independently from the rest of the structure as a rigid body without causing internal loads in the structure. A hinged door, for example, is a mechanism with one rigid-body freedom. If the hinges are disconnected, the door mechanism has six rigid-body freedoms.

Mechanisms are characterized by nondimensional numbers derived by dividing the terms of [D] into the corresponding diagonal term of [A] . If these "matrix diagonal to factor diagonal" ratios are large numbers, various warning and fatal messages are produced, depending on the context.

F.8   Sources of Nonpositive Definite Matrices

A negative semidefinite element stiffness matrix, which is defined as one whose eigenvalues are all negative or zero, implies that the element has an energy source within it. NX Nastran discourages using such elements by giving fatal messages when negative element thicknesses or section properties are input. However, there are applications where such elements are useful, such as when using an element with negative stiffness in parallel with a passive element to model a damaged or thinned-down element. For this reason, negative material property coefficients and negative stiffnesses for scalar elements are allowed. Also, some incorrect modeling techniques, such as the misuse of the membrane-bending coupling term on PSHELL entries (MID4), can lead to negative eigenvalues of the element stiffness matrix.

Stiffness matrices with negative eigenvalues cause negative terms in [D]. The number of such terms is automatically output by the standard symmetric decomposition subroutine. Their existence again causes various warning and fatal messages, depending on the context. The most common cause of negative terms in [D] is true mechanisms, whose terms are small negative numbers and are actually computational zeroes.

Indefinite matrices occur when using the Lagrange multiplier method for constraint processing. In this method, the constraints are not eliminated from but are concatenated to the g-size system of equations. The constraint equations have zeroes on the diagonal, resulting in indefinite matrices. These systems can be solved using the block pivoting scheme of the sparse decomposition.

F.9   Detection and Avoidance of Numerical Problems

Static Analysis

Models used in static analysis must be constrained to ground in at least a statically determinate manner even for unloaded directions. For example, a model intended for only gravity loading must be constrained in horizontal directions as well as vertical directions. The evidence of unconstrained directions is that the entire model is a mechanism, that is, the large ratio occurs at the last grid point in the internal sequence.

Another source of high ratios arises from connecting soft elements to stiff elements. Local stiffness is a function of element thickness (moment per unit rotation) through element thickness cubed (force per unit deflection) and is inversely proportional to mesh spacing, again in linear through cubic ratios. Some relief is possible by sequencing the soft degrees-of-freedom first in the internal sequence, although this is difficult to control in the presence of automatic resequencing.

More reliable corrections are to replace the very stiff elements with rigid elements or to place the soft and stiff elements in different superelements.

A third source of high ratios is the elements omitted through oversight. The corrective action here is to start with the grid points listed in the diagnostics and track back through the elements connected to them through the upstream grid points. The missing elements may be anywhere upstream. PARAM,GPECT,1 output and undeformed structure plots all provide useful data for detecting missing elements.

At present, there are two major methods of identifying large ratios and nonpositive-definite matrices. In some solutions, the largest matrix diagonal to factor diagonal ratio greater than 105 (MAXRATIO default) is identified by its internal sequence number, and the number of negative factor diagonal terms is output. The best method to identify mechanisms here is to apply checkout loads that cause internal loads in all of the elements. Then inspect the displacement output for groups of grid points that move together with implausibly large displacements and common values of grid point rotation. The only condition that causes a fatal error is a true null column, and NASTRAN SYSTEM(69)=16 avoids this fatal error by placing a unit spring coefficient on the degrees-of-freedom with null columns. This option is recommended only for diagnostic runs because it may mask modeling errors.

In other solution sequences, all ratios greater than 10^5 are printed in a matrix format named the MECH table. The external sequence number of each large ratio is also printed, which is the grid point and degree-of-freedom number. If any such ratios exist, the action taken depends on the value of PARAM,BAILOUT. In the conventional solution sequences, its default value causes the program to continue after printing the MECH matrix. In the superelement solution sequences, a different default causes a fatal error exit after printing the MECH matrix. For both types of solution sequences, the opposite action may be requested by setting the value of PARAM,BAILOUT explicitly. Also, the criterion used for identifying large ratios may be changed using PARAM,MAXRATIO.

For static analysis, values between 10^3 and 10^6 are almost always acceptable. Values between 10^7 and 10^8 are questionable. When investigating structures after finding these values, some types of structures may be found to be properly modeled. It is still worthwhile to investigate the structures with questionable values.

The solutions with differential stiffness effects offer another method to obtain nonpositive definite stiffness matrices. For example, a column undergoing compressive gravity loading has a potential energy source in the gravity load. A lateral load that is stabilizing in the absence of gravity (i.e., a decreased load causes a decreased deflection) is destabilizing when applied in a postbuckled state.

Eigensolutions Using the Inverse Iteration and Lanczos Methods

The matrix $[K - \lambda_s M]$ is decomposed where λ_s is an eigenvalue shift and $[K]$ and $[M]$ are the stiffness and mass matrices, respectively. This condition allows the solution of models with rigid-body modes since the singularities in $[K]$ are suppressed if there are compensating terms in $[M]$. The only conditions that should cause fatal messages due to singularity are

1. The same null column in both $[K]$ and $[M]$. These columns and rows are given an uncoupled unit stiffness by the auto-omit operation if the default value (0) for PARAM,ASING is used. If this value is set to -1, a null column in both matrices is regarded as an undefined degree of freedom and causes a fatal error exit.

2. A massless mechanism. One commonly encountered example is a straight-line, pinned rod structure made from CBAR elements with no torsional stiffness (J) defined. The structural mass generated for CBAR elements does not include inertia for the torsion

degrees-of-freedom. The natural frequency for this torsion mode approaches the limit of zero torsional stiffness divided by zero mass, which is an undefined quantity. If the elements lie along a global coordinate axis, the mass term is identically zero, which leads to very large negative or positive eigenvalues and is usually beyond any reasonable search region. If the elements are skewed from the global axes, the eigenvalues may be computed at any value (including negative) because of the indeterminacy caused by numerical truncation.

The "negative terms on factor diagonal" message generally occurs for every decomposition performed in the iteration. It can be shown from Sturm sequence theory that the number of negative terms is exactly equal to the number of eigenvalues below λ_s. This condition is a means of determining if all roots in the range have truly been found.

Eigensolutions Using Transformation Methods

If the HOU method (and variations) are selected, one of the first operations performed is a Cholesky decomposition, which is used to reduce the problem to a special eigenvalue problem. The mass matrix is given this operation for the straight HOU method. Columns that are identically null are eliminated by the auto-omit operation. Poor conditioning can result from several sources. One example is a point mass input on a C0NM2 entry that uses offsets in three directions. The grid point to which it is attached has nonzero mass coefficients for all six degrees-of-freedom. However, only three independent mass degrees-of-freedom exist, not six. Another example results in a superelement analysis when most of the elements in a superelement do not have mass and all interior masses are restricted to only a few structural or mass elements. The boundary matrix produced for the superelement is generally full, no matter what its rank, i.e., regardless of the number of independent mass degrees-of-freedom that it contains.

The presence of a rank-deficient mass matrix when using the HOU method produces fatal messages due to the singularity of the mass matrix or produces solutions with poor numerical stability. Poor stability is most commonly detected when making small changes to a model and then observing large changes in the solution. Either the MHOU method or the AHOU method is the preferred methods to use when nearly singular mass matrices are expected because these methods decompose the matrix $[K - \lambda_s M]$ instead of the mass matrix. The shift parameter λ_s is automatically set to be large enough to control rigid-body modes. Better modeling practices also reduce the costs and increase the reliability for the two examples cited above. If an offset point mass is significant in any mode, it is better to attach it to an extra grid point at its center of gravity and model the offset with a rigid element. The auto-omit feature then eliminates the rotational degrees-of-freedom. Similarly, if only a few interior points of a superelement have mass, it may be more economical to convert them to exterior points, which also eliminates the singular boundary mass matrix.

It is possible to input negative terms into a mass matrix directly with DMIG terms or scalar mass elements. This class of problem causes fatal errors due to nonpositive definite mass matrices for the transformation methods, fatal errors with the Lanczos method. The complex eigenvalue methods should be used for this type of problem, which infers that modal or dynamic reduction methods may not be used.

A similar but quite different problem arises because Cholesky decomposition is used on the generalized mass matrix (named MI in the diagnostics) when orthogonalizing the eigenvectors with respect to the mass matrix. The existence of negative terms here indicates that poor eigenvectors were computed. The row number where the negative term occurs is printed in the diagnostic. This row number does not refer to a physical degree of freedom but refers instead to an eigenvector number. The usual cause is computing eigenvectors for computational infinite roots in the modified transformation methods. This problem can be avoided after the fact by setting ND on the EIGR entry to a value less than the row number that appears in the diagnostics or before the fact by setting F2 on the EIGR entry to a realistic value instead.

Frequency Response Analysis

Negative terms on the factor and high factor to matrix diagonal ratios can be expected when using coupled methods and may often be safely ignored. These messages are merely an indication that the excitation frequency is above one or some of the natural frequencies of the system (negative terms), or is near a natural frequency (high ratio).

Transient Response Analysis

A well-conditioned model should have neither negative terms nor high ratios on its factor terms. The causes of such messages include all of the effects described above. Negative mass terms can be detected by rapidly diverging oscillations in the transient solution.

The Large Mass Method

One of the methods for enforcing motion is the large mass method. It may be used with direct methods as well as all of the reduction methods. The basis of the method is to attach artificial masses to the structure at the degrees-of-freedom where the motion is to be enforced. These large masses should be orders of magnitude larger than the total mass or the moment of inertia of the structure. A survey of the literature shows recommendations for mass ratios ranging from 10^3 to 10^8 with a value of 10^6 as the most common recommendation.

The mass ratio affects both the accuracy and numerical conditioning, and must be adjusted in a compromise that meets both criteria. With regard to load accuracy, the error in the approximation is inversely proportional to the ratio. A ratio of 10^3 causes an error of ten percent at resonance for a mode with one-half percent of damping, which represents an extreme case. Off-resonance excitation or higher damping ratios result in lower errors. Numerical conditioning problems are much more difficult to predict.

The Lagrange Multiplier Method

Another method to enforce motion is the Lagrange multiplier method. In this method, the input motion function is described by a constraint equation. This method provides better accuracy than the large mass method where the numerical error introduced is proportional to the large mass.

The formulation introduces an indefinite system of linear equations where some numerical problems may arise from the fact that the system matrix contains terms that have dimensions of stiffness as well as nondimensional (constraint) terms. The decomposition of this matrix with the sparse decomposition methods pivoting strategy is stable.

You may control the pivoting strategy with the THRESH DMAP parameter mentioned earlier. The default value (–6) is adequate for most Lagrange multiplier solutions. In fact, higher (ranging up to –2) values provide better accuracy, while increasing the number of pivots may result in performance degradation.

Appendix

G   Grid Point Weight Generator

G.1   Overview

The grid point weight generator (GPWG) calculates the masses, centers of gravity, and inertias of the mathematical model of the structure. The data are extracted from the mass matrix by using a rigid-body transformation calculation. Computing the mass properties is somewhat complex because a finite element model may have directional mass properties, that is, the mass may differ in each of the three coordinate directions. From a mathematical point of view, the NX Nastran mass may have tensor properties similar to the inertia tensor. This complexity is reflected in the GPWG output. All of the transformations used in calculating the mass properties are shown for the general case. Since most models have the same mass in each of the three coordinate directions, the GPWG output provides more information than you generally need.

To avoid unnecessary confusion and at the same time provide the necessary information for the advanced user, the discussion of the GPWG is separated into two sections. In Commonly Used Features, a basic discussion is given that should satisfy most users. If you need additional information, read Example with Direction Dependent Masses.

In both sections, a simple model consisting of four concentrated masses is used to demonstrate the GPWG output. In the first section, the mass is the same in each direction. For the second section, the mass is different in each of the three directions.

G.2   Commonly Used Features

To demonstrate the typical output generated by the GPWG, a small model consisting of four concentrated masses as shown in Figure G-1 is used. This model is typical of most models because the mass is the same in each coordinate direction. The number of masses has been kept small so you can better understand the physics.

Concentrated masses are located at four different grid points. The displacement coordinate system for each of the grid points is the basic coordinate system.
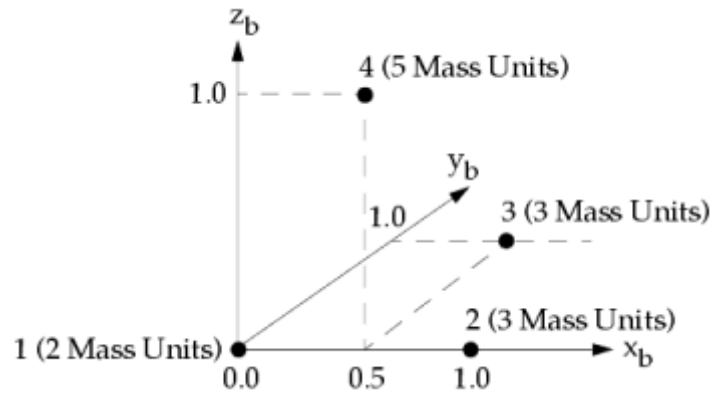
Figure G-1. Four Concentrated Mass Model

To request the GPWG output, you must add parameter GRDPNT in either the Bulk Data Section or the Case Control Section as follows:

        PARAM,GRDPNT,x

If

x = –1   GPWG is skipped (default).

x = 0   The mass properties are computed relative to the origin of the basic coordinate system.

x > 0   The mass properties are computed relative to grid point x. If grid point x does not exist, the properties are computed relative to the basic coordinate system.

For the four masses shown in Figure G-1, the resulting GPWG output is given in Figure G-2.

```
O U T P U T F R O M G R I D P O I N T W E I G H T G E N E R A T O R
REFERENCE POINT = 0
M O
* 1.300000E+01 0.000000E+00 0.000000E+00 0.000000E+00 5.000000E+00 -3.000000E+00 *
* 0.000000E+00 1.300000E+01 0.000000E+00 -5.000000E+00 0.000000E+00 7.000000E+00 *
* 0.000000E+00 0.000000E+00 1.300000E+01 3.000000E+00 -7.000000E+00 0.000000E+00 *
* 0.000000E+00 -5.000000E+00 3.000000E+00 8.000000E+00 -1.500000E+00 -2.500000E+00 *
* 5.000000E+00 0.000000E+00 -7.000000E+00 -1.500000E+00 1.000000E+01 0.000000E+00 *
* -3.000000E+00 7.000000E+00 0.000000E+00 -2.500000E+00 0.000000E+00 8.000000E+00 *
S
* 1.000000E+00 0.000000E+00 0.000000E+00 *
* 0.000000E+00 1.000000E+00 0.000000E+00 *
* 0.000000E+00 0.000000E+00 1.000000E+00 *
DIRECTION
MASS AXIS SYSTEM (S) MASS X-C.G. Y-C.G. Z-C.G.
X 1.300000E+01 0.000000E+00 2.307692E-01 3.846154E-01
Y 1.300000E+01 5.384616E-01 0.000000E+00 3.846154E-01
Z 1.300000E+01 5.384616E-01 2.307692E-01 0.000000E+00
I(S)
* 5.384615E+00 -1.153847E-01 -1.923079E-01 *
* -1.153847E-01 4.307692E+00 -1.153846E+00 *
* -1.923079E-01 -1.153846E+00 3.538461E+00 *
I(Q)
* 5.503882E+00 *
* 5.023013E+00 *
* 2.703873E+00 *
Q
* 8.702303E-01 4.915230E-01 3.323378E-02 *
* 3.829170E-01 -7.173043E-01 5.821075E-01 *
* 3.099580E-01 -4.938418E-01 -8.124324E-01 *
```

Figure G-2. GPWG Output for the Four Concentrated Mass Model

The [*MO*] matrix represents the rigid-body mass properties of the structure and is generally not needed for model checkout. This matrix represents an intermediate step in computing the inertia properties of the structure. The [*S*] matrix should always be equal to the identity matrix

when the mass is the same in each coordinate direction, which is the typical case. If this matrix is not the identity matrix, inspect the model for inconsistent masses.

Following the [*S*] matrix are the mass and center of gravity locations. These are the most commonly used information of the GPWG output. Because the mass may be different in the three translational directions, the mass is printed for every coordinate direction. For the same reason, the center of gravity location is given for each of the three translational masses. If the mass is the same in all directions, a unique center of gravity exists and is located at the x-component of the y (or z) mass, the y-component of the x (or z) mass, and the z-component of the x (or y) mass. If the mass is not the same in all three directions, it is likely due to the CONM1, CMASSi, or DMIG input. If the reference point is specified with PARAM,GRDPNT,0 and [*S*] is the identity matrix, then the center of gravity location is given in the basic coordinate system. For the example, the mass of the structure is 13.0 and the center of gravity location is (0.5384, 0.2307, 0.3846) in the basic coordinate system.

If a grid point ID is used for the reference point, and [*S*] is an identity matrix, then the center of gravity location is in a coordinate system parallel to the basic coordinate system with an origin located at the grid point.

If the [*S*] matrix is equal to the identity matrix, then the [*I(S)*] matrix represents the inertia matrix of structure for the center of gravity with respect to the basic coordinate system, the [*I(Q)*] matrix is the corresponding principal moments of inertia matrix, and [*Q*] represents the transformation from the principal directions to the basic coordinate system.

The following additional comments on the GPWG should be noted.

- The scale factor entered with parameter WTMASS is applied to the assembled element mass before the GPWG. The GPWG module, however, converts mass back to the original input units that existed prior to the scaling effect of the parameter WTMASS. (Note that the parameter WTMASS is not applied to M2GG or M2PP input, but the M2GG mass is assembled into the mass matrix prior to GPWG. Therefore, for GPWG output only, the M2GG mass is scaled by the same parameter as the element mass. M2GG input may be scaled independently using the CM2 parameter.)

- The GPWG is performed on the g-size mass matrix, which is the mass matrix prior to the processing of the rigid elements, MPCs, and SPCs.

- The mass at scalar points and fluid-related masses are not included in the GPWG calculation.

- The GPWG for a superelement does not include the mass from upstream superelements. Therefore, the GPWG for the residual structure includes only the mass on the residual points. The center of gravity location is also based on the mass of the current superelement only.

- If a large mass is used for enforced motion, the large mass dominates the structural mass. For model checkout, it is recommended to remove the large mass and constrain the driving point. A static analysis is a convenient way to generate a mass matrix and obtain output from the GPWG.

- The output from the GPWG is for information purposes only and is not used in the analysis.

G.3 Example with Direction Dependent Masses

In the previous section, the mass was the same in each of the three coordinate directions thereby producing a unique center of gravity location. However, if scalar masses are used, the total mass may have different properties in each direction, and the center of gravity may not be a unique

location. This effect is shown in the output by providing the direction and center of gravity for each of the three mass principal components.

When using directional mass, the axes about which the inertia matrix $i[S]$ is calculated are referred to as the principal mass axes. The principal mass axes may not necessarily intersect. However, these axes provide uncoupled rotation and translation mass properties. If the structural model is constructed using only real masses, the three principal mass values printed out are equal, the center of gravity is unique, and the principal mass axes intersect at the center of gravity.

To demonstrate all of the features of the GPWG module, the four-mass sample problem discussed in the previous section is modified so that the mass is not equal in each of the three translational directions (see Figure G-3). Furthermore, different displacement coordinate systems are used for the grid points. The displacement coordinate system for grid point 1 is the local rectangular system 1, which is oriented at an angle of 45 degrees (about the z_b axis). The displacement coordinate system for grid point 3 is the local rectangular system 3, which is oriented at an angle of 60 degrees (about the z_b axis). The grid point locations and masses are summarized in Table G-1.
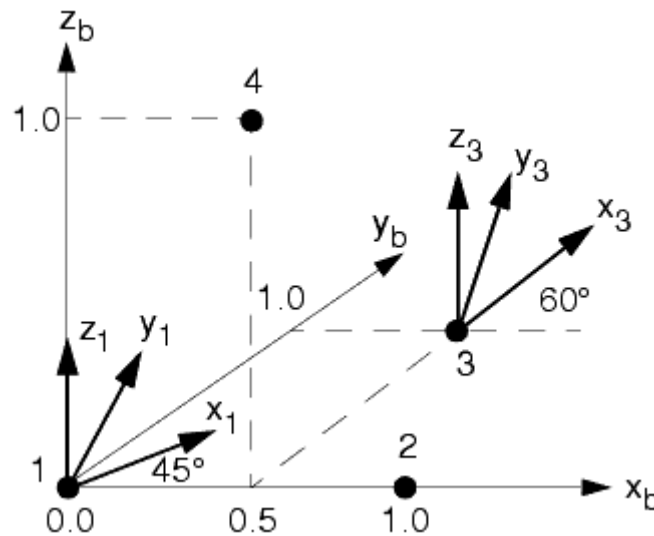

Figure G-3. Four Concentrated Mass Model

Table G-1. Location and Size of Masses

Grid ID	Location Basic System (CP Fields)			Mass Global System (CD Fields)		
	x_b	y_b	z_b	x_{CD}	y_{CD}	z_{CD}
1	0	0	0	2	3	5
2	1	0	0	2	3	5
3	0.5	1	0	2	3	5
4	0.5	0	1	2	3	5

The GPWG output for the four mass model is shown in G-2.

Table G-2. Output from the Grid Point Weight Generator

```*** USER WARNING MESSAGE 3042 MODULE = GPWG``` ```INCONSISTENT SCALAR MASSES HAVE BEEN USED. EPSILON/DELTA = 3.8054429E-02```	

``` REFERENCE POINT =   1```
```   M O```
```* 9.250000E+00 -9.330128E-01 0.000000E+00 0.000000E+00 2.000000E+00 -2.966506E+00 *```
```* -9.330128E-01 1.075000E+01 0.000000E+00 -3.000000E+00 0.000000E+00 6.058013E+00 *```
```* 0.000000E+00 0.000000E+00 2.000000E+01 5.000000E+00 -1.000000E+01 0.000000E+00 *```
```* 0.000000E+00 -3.000000E+00 5.000000E+00 8.000000E+00 -2.500000E+00 -1.500000E+00 *```
```* 2.000000E+00 0.000000E+00 -1.000000E+01 -2.500000E+00 9.500000E+00 0.000000E+00 *```
```* -2.966506E+00 6.058013E+00 0.000000E+00 -1.500000E+00 0.000000E+00 7.495513E+00 *```
	Rigid-Body Mass Properties Matrix for the Reference Point

``` S```
```   * 4.321332E-01 9.018098E-01 0.000000E+00 *```
```   * -9.018098E-01 4.321332E-01 0.000000E+00 *```
```   * 0.000000E+00 0.000000E+00 1.000000E+00 *```
	Transformation from the Principal Mass to the Basic Direction

``` DIRECTION```
```   MASS AXIS SYSTEM (S)   MASS   X-C.G.   Y-C.G.   Z-C.G.```
```   X   1.119709E+01   3.480388E-02 6.023980E-01 2.512494E-01```
```   Y   8.802916E+00   -6.515555E-03 -4.426965E-02 2.484108E-01```
```   Z   2.000000E+01   -9.385824E-03 5.589382E-01 0.000000E+00```
	Center of Gravity Relative to the Reference Point in the Principal Mass Axes System

``` I(S)```
```   * 4.376953E+00 -8.768300E-01 6.624477E-01 *```
```   * -8.768300E-01 5.623007E+00 -3.419880E-01 *```
```   * 6.624477E-01 -3.419880E-01 3.431904E+00 *```
	Moments of Inertia with Respect to Principal Mass Axes for the Center of Gravity

``` I(Q)```
```   * 4.463246E+00   *```
```   *   6.075616E+00   *```
```   *   2.893001E+00 *```
	Principal Moments of Inertia

```   Q```
```   * 7.201511E-01 4.586845E-01 5.205678E-01 *```
```   * -3.717336E-01 8.885992E-01 -2.687111E-01 *```
```   * -5.858298E-01 0.000000E+00 8.104341E-01 *```
	Transformation from the Principal Direction of the Momental Ellipsoid to the Principal Mass Axes

Before showing how each of the matrices are computed, a few items should be noted for this model:

- User Warning Message 3042 is printed to inform you that inconsistent scalar masses were used. This message occurs because there are different scalar masses in the three components. In general, if you are using structural mass and/or CONM2s, you should not get this message.

- The rigid-body mass matrix [*MO*] is computed with respect to the reference grid point1 in the basic coordinate system. Grid point 1 is used for this example because PARAM,GNDPNT,1 is entered in the Bulk Data Section.

- The mass and center of gravity location shown are not in the basic coordinate system for this example. The mass and center of gravity are computed and printed in the principal mass coordinate system. This principal mass coordinate system should not be confused with the principal axes discussed in most text books. The principal mass axes in NX Nastran are the axes that have no coupling terms between the translational masses (diagonal translational mass matrix). Also, the NX Nastran principal mass axes are not the axes of the inertia ellipsoid. The [*S*] matrix is the transformation from the principal mass direction to the basic coordinate system.

  This additional step may sound confusing, but it is necessary. In real structures, the mass of structure is generally the same in all directions, so there is no coupling between the translational mass terms. Since text books are written to solve real structural problems, there is no need to discuss the principal mass axes, as they are called in NX Nastran. However, with NX Nastran you are not restricted to the same mass in each coordinate direction—a situation that may not be physically realizable but still is quite useful for certain modeling situations. Therefore, the additional step of computing principal mass axes is necessary.

- If your model has the same mass in all coordinate directions, then the [*S*] matrix is the identity matrix indicating that the principal mass axes is the basic coordinate system. Always check the [*S*] matrix. If it is not the identity matrix, verify that the mass distribution is correct. Do not use the directional mass and center of gravity location blindly. Remember, these quantities are in the principal mass axes.

To fully understand how the GPWG module works, it is useful to trace the steps NX Nastran follows to generate the output shown in Figure G-2. The following shows the step-by-step procedure for the four mass example.

1. The GPWG module uses the global mass matrix, which is the mass matrix before any constraints are applied. In this example, there are four grids, each with six degrees-of-freedom, resulting in a total of 24 degrees-of-freedom in the mass [*MJJ*] matrix. The matrix is shown in Figure G-4.

$$[MJJ] =
\begin{bmatrix}
2 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 3 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 5 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & \textbf{Grid 1} & & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 2 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 3 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 5 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & \textbf{Grid 2} & & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 2 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 3 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 5 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & \textbf{Grid 3} & & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 2 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 3 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 5 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & \textbf{Grid 4} & & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0
\end{bmatrix}$$

Row labels (right): 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 21 22 23 24

Column labels (below): 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 21 22 23 24

Figure G-4. Global Mass Matrix

The [*MJJ*] matrix shows the mass contribution for each of the four grid points. Note that the coordinate system associated with rows and columns 1 through 6 (grid point 1) is coordinate system 1, the coordinate system associated with rows and columns 13 through 18 (grid point 3) is coordinate system 3. The remaining rows and columns are in the basic coordinate system.

2. To generate the 6x6 rigid-body mass matrix [*MO*] for the structure, it is necessary to compute the mass matrix in the basic coordinate system relative to the reference point. This computation requires the transformation matrix [*D*] that relates the rigid-body displacements in the global system u_g to the six unit displacements in the basic coordinate system located at the reference grid point (u_o) as shown in Equation G-1.

$$\{u_g\} = [D]\{u_o\}$$

Equation G-1.

(Reference Point)

The transformation matrix $[D]$ is assembled from the individual transformation matrices $[d_i]$ computed for each grid point. Each individual transformation matrix $[d_i]$ consists of two transformations: $[Tr]_i$, which relates the location of the grid point to the reference grid point in the basic coordinate system, and $[Ti]_i$, which relates the global coordinate system at the grid point to the basic coordinate system.

The $[Tr]_i$ transformation matrices are first computed by constructing the location vectors in the basic coordinate system r_i for each grid point in the model relative to the reference point as shown in Equation G-2.

$$\{r\}_i = \{R\}_i - \{R\}_0 = \begin{Bmatrix} r_1 \\ r_2 \\ r_3 \end{Bmatrix}$$

Equation G-2.

The location vectors for the example are as follows:

$$\{r\}_1 = \begin{Bmatrix} 0 \\ 0 \\ 0 \end{Bmatrix}; \{r\}_2 = \begin{Bmatrix} 1 \\ 0 \\ 0 \end{Bmatrix}; \{r\}_3 = \begin{Bmatrix} .5 \\ 1 \\ 0 \end{Bmatrix}; \{r\}_4 = \begin{Bmatrix} .5 \\ 0 \\ 1 \end{Bmatrix}$$

Equation G-3.

Using the location vectors, the grid point transformation matrix $[Tr]_i$ is computed for each grid point by expanding the location vectors to a 3x3 matrix as shown in Equation G-4.

$$[T_r]_i = \begin{bmatrix} 0 & r_3 & -r_2 \\ -r_3 & 0 & r_1 \\ r_2 & -r_1 & 0 \end{bmatrix}$$

Equation G-4.

For the example problem, the grid point transformation matrices are

$$[Tr]_1 = \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix}; [Tr]_2 = \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & 1 \\ 0 & -1 & 0 \end{bmatrix}; [Tr]_3 = \begin{bmatrix} 0 & 0 & -1 \\ 0 & 0 & .5 \\ 1 & -.5 & 0 \end{bmatrix}; [Tr]_1 = \begin{bmatrix} 0 & 1 & 0 \\ -1 & 0 & .5 \\ 0 & -.5 & 0 \end{bmatrix}$$

Equation G-5.

The coordinate system transformation matrices from the global coordinate system to the basic system coordinates are given by the direction cosine matrices as follows:

$$[Ti]_1 = \begin{bmatrix} \cos(45°) & -\sin(45°) & 0 \\ \sin(45°) & \cos(45°) & 0 \\ 0 & 0 & 1 \end{bmatrix}; \quad [Ti]_2 = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix};$$

$$[Ti]_3 = \begin{bmatrix} \cos(60°) & -\sin(60°) & 0 \\ \sin(60°) & \cos(60°) & 0 \\ 0 & 0 & 1 \end{bmatrix}; \quad [Ti]_4 = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

Equation G-6.

The grid point transformation $[Tr]_i$ and the coordinate system transformation $[Ti]_i$ are combined to form the individual grid point transformation matrix $[d]_i$ for each grid point using Equation G-7.

$$[d]_i = \begin{bmatrix} Ti^T & Ti^T \cdot T_r \\ 0 & Ti^T \end{bmatrix}$$

Equation G-7.

The rows of each $[d]_i$ form the columns of the global transformation matrix $[D]^T$ as shown in Equation G-8.

$$[D]^T = \begin{bmatrix} d_1^T & d_2^T & \dots & d_n^T \end{bmatrix}$$

Equation G-8.

Using Equation G-9, the global transformation matrix for the example is

$$[D]^T = \begin{bmatrix} .707 & -.707 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & .5 & -.866 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 \\ .707 & .707 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & .866 & .5 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & .707 & -.707 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 1 & .5 & -.866 & 0 & 0 & -1 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & .707 & .707 & 0 & 0 & 0 & -1 & 0 & 1 & 0 & 0 & 0 & -.5 & .866 & .5 & 0 & 1 & 0 & -.5 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & 1 & 0 & 0 & 0 & 1 & -.067 & 1.116 & 0 & 0 & 0 & 1 & 0 & .5 & 0 & 0 & 0 & 1 \end{bmatrix}$$

| Grid 1 | Grid 2 | Grid 3 | Grid 4 |

Equation G-9.

Using the global transformation matrix $[D]$, the rigid-body mass matrix about the reference point in the basic coordinate system $[MO]$ is obtained by Equation G-10.

$$[M0] = [D]^T[MJJ][D]$$

Equation G-10.

For the example, $[MO]$ is determined to be

$$[M0] = \begin{bmatrix} 9.25 & -.933 & 0. & 0. & 2. & -2.967 \\ -.933 & 10.75 & 0. & -3. & 0. & 6.058 \\ 0. & 0 & 20. & 5. & -10. & 0. \\ 0. & -3 & 5. & 8. & -2.5 & -1.5 \\ 2. & 0 & -10. & -2.5 & 9.5 & 0. \\ -2.967 & 6.058 & 0. & -1.5 & 0. & 7.496 \end{bmatrix}$$

Equation G-11.

Comparing the results shown in Figure G-11 to $[MO]$ generated by the GPWG module (Figure G-2) shows the matrices to be numerically the same.

3. The next step is to inspect the $[MO]$ to determine whether the basic coordinate system can be used as the principal mass directions. The principal mass axes are axes that have no coupling between the translational mass components. For real structures, there is no coupling in the translational mass terms in the inertia matrix. However, with inconsistent scalar masses (CONM1, CMASSi, or DMIG), you may define any type of mass matrix you desire.

    To determine whether coupling exists between the translational mass terms, $[MO]$ is partitioned into four 3x3 matrices as shown in Equation G-12.

$$[M0] = \left[\begin{array}{c|c} \overline{M}^t & \overline{M}^{tr} \\ \hline \overline{M}^{rt} & \overline{M}^r \end{array}\right]$$

Equation G-12.

where the superscripts t and r refer to translation and rotation, respectively.

For this example, the translational mass partition is given by

$$[\overline{M}^t] = \begin{bmatrix} 9.25 & -.933 & 0 \\ -.933 & 10.75 & 0 \\ 0 & 0 & 20. \end{bmatrix}$$

Equation G-13.

A check is made for coupling as follows:

$$\delta = \sqrt{\sum (\overline{M}^t_{ij})^2} \qquad i = j$$

$$\varepsilon = \sqrt{\sum (\overline{M}^t_{ij})^2} \qquad i > j$$

Equation G-14.

If ϵ/d is greater than .001, then excessive coupling exists preventing the basic coordinate system from being used for the principal mass directions and User Warning Message 3042 is printed. For this problem, $\delta = (9.25^2 + 10.75^2 + 20^2)^{.5}$ and $\varepsilon = (.933^2 + 0.^2 + 0.^2)^{.5}$. The ratio $\epsilon/d = .038$ agrees with the System Warning Message 3042 shown in Figure G-2.

If needed, the principal mass directions are computed by performing an eigensolution with the translational mass components. The eigenvectors of $[\overline{M}^t]$ are the columns of the transformation matrix $[S]$

$$[S] = [\{e_1\}, \{e_2\}, \{e_3\}]$$

Equation G-15.

Using this eigenvector matrix, the partitions of the rigid-body mass matrix with respect to the principal mass direction are computed by Equation G-16.

$$[M^t] = [S]^T [\bar{M}^t][S]$$

$$[M^{tr}] = [S]^T [\bar{M}^{tr}][S]$$

$$[M^r] = [S]^T [\bar{M}^r][S]$$

Equation G-16.

The $[S]$ matrix for the example is given by

$$[S] = \begin{bmatrix} .432 & .902 & 0. \\ -.902 & .432 & 0. \\ 0. & 0. & 1. \end{bmatrix}$$

Equation G-17.

The [S] matrix is printed after the [M0] matrix as shown in Figure G-2. This represents the transformation relating the basic coordinate system to the principal mass axes. Again, if there is no coupling between the translational mass component, which is the case for most problems, the eigensolution is not required, and the [S] matrix is set equal to the identity matrix. This example was selected to demonstrate all of the features of the GPWG module, but it is not a typical problem.

4. The next step is to determine the principal masses and the center of gravity location in the principal mass axes system as shown in Equations G-18.

$$M_x = M'_{11}$$

$$M_y = M'_{22}$$

$$M_z = M'_{33}$$

The "centers of gravity" are

$$X_x = \frac{M^{tr}_{11}}{M_x}, \quad Y_x = \frac{-M^{tr}_{13}}{M_x}, \quad Z_x = \frac{M^{tr}_{12}}{M_x},$$

$$X_y = \frac{M^{tr}_{23}}{M_y}, \quad Y_y = \frac{M^{tr}_{22}}{M_y}, \quad Z_y = \frac{-M^{tr}_{21}}{M_y},$$

$$X_z = \frac{-M^{tr}_{32}}{M_z}, \quad Y_z = \frac{M^{tr}_{31}}{M_z}, \quad Z_z = \frac{M^{tr}_{33}}{M_z}$$

Equation G-18.

As can be seen, the center of gravity location is not a unique location. The center of gravity location is computed separately for the for x-, y-, and z-directions relative to the principal mass axes. Only if the mass is the same in each direction (which is typical) is there a unique center of gravity location, which is relative to the reference point in the basic coordinate system.

For this example problem, the center of gravity locations are determined to be

Table G-3. Mass Center of Gravity Locations

Mass Component	Center of Gravity Location in Principal Mass Axes		
	X	Y	L
x - mass	.035	–0.602	0.251
y - mass	–0.007	–0.044	0.248
z - mass	–0.009	–0.559	0

The center of gravity location given in Figure G-3 is the same as shown in Figure G-2.

5. Following the center of gravity calculation is the calculation to determine the moments of inertia for the center of gravity with respect to the principal mass axes $[I(S)]$ as shown in Figure G-19.

$$I_{11}^{(S)} = M_{11}^r - M_y Z_y^2 - M_z Y_z^2$$

$$I_{12}^{(S)} = I_{21}^{(S)} = -M_{12}^r - M_z X_z Y_z$$

$$I_{13}^{(S)} = I_{31}^{(S)} = -M_{13}^r - M_y X_y Z_y$$

$$I_{22}^{(S)} = M_{22}^r - M_z X_z^2 - M_x Z_x^2$$

$$I_{32}^{(S)} = I_{23}^{(S)} = -M_{23}^r - M_x Y_x Z_x$$

$$I_{33}^{(S)} = M_{33}^r - M_x Y_x^2 - M_y X_y^2$$

Equation G-19.

For the example, the inertia matrix $[I(S)]$ is given by

$$[I(S)] = \begin{bmatrix} 4.377 & -.877 & .662 \\ -.877 & 5.623 & -.342 \\ .662 & -.342 & 3.432 \end{bmatrix}$$

Equation G-20.

6. The final step is to compute the principal moments of inertia and the principal directions of the momental ellipsoid (commonly referred to as the principal axes in text books). An intermediate inertia matrix $[\bar{I}]$ is generated by reversing the sign on the off diagonal terms of $[I(S)]$. For the inertia matrix given in Equation G-20, $[\bar{I}]$ is given by

$$[\bar{I}] = \begin{bmatrix} 4.377 & .877 & -.662 \\ .877 & 5.623 & .342 \\ -.662 & .342 & 3.432 \end{bmatrix}$$

Equation G-21.

An eigensolution is performed on the $[\bar{I}]$ matrix to determine the principal directions. The resulting eigenvalues are the principal moments of inertia, which are assumed to be the diagonal terms of the principal inertia matrix $[I(Q)]$. The eigenvectors form the columns of the matrix [Q], which is the transformation relating the intermediate inertia matrix $[\bar{I}]$ to the principal inertia matrix $[I(Q)]$ as shown in Equation G-22.

$$[I(Q)] = [Q]^T [\bar{I}][Q]$$

Equation G-22.

For the example, [Q] and [I (Q)] are given by

$$[I(Q)] = \begin{bmatrix} 4.463 & 0. & 0. \\ 0. & 6.076 & 0. \\ 0. & 0. & 2.893 \end{bmatrix}$$

$$[Q] = \begin{bmatrix} .720 & .459 & .521 \\ -.372 & .889 & -.269 \\ -.586 & 0 & .810 \end{bmatrix}$$

Equation G-23.

The matrices [S] and [Q] matrices are the coordinate rotation matrices, which when taken together, relate the principal directions of the momental ellipsoid to the basic coordinate system. The matrices given in Equation G-23 are in agreement with those given in Equation G-2. The example for the four mass model is now complete.

Appendix

H   Diagnostic Messages for Dynamic Analysis

This appendix lists common diagnostic messages for dynamic analysis. The text for each message is given in uppercase letters and is followed by additional explanatory material, including suggestions for remedial action..

The messages in this section have the following format:

*** (SYSTEM/USER) (FATAL/WARNING/INFORMATION) MESSAGE ID, text

where "ID" is a unique message identification number and "text" is the message as indicated in capital letters for each of the diagnostic messages. Four asterisks (****) in the message text indicates information that is filled in for a specific use of the message, such as the number of a grid point or the name of a Bulk Data entry. Some of the messages are followed by additional explanatory material, including suggestions for remedial action.

Fatal messages cause the termination of the execution following the printing of the message text. These messages always appear at the end of the NX Nastran output. Warning and information messages appear at various places in the output stream. Such messages only convey warnings or information to the user. Consequently, the execution continues in a normal manner following the printing of the message text.

As an example, consider message number 2025, which appears in the printed output as follows:

*** USER FATAL MESSAGE 2025, UNDEFINED COORDINATE SYSTEM 102

The three leading asterisks (***) are always present in numbered user and system diagnostic messages. The word USER indicates that this is a user problem rather than a system problem. The word FATAL indicates that this is a fatal message rather than a warning or information message. The number 2025 is the identification number for this message. The text of the message follows the comma. The number 102 replaces the asterisks (****) in the general message text and indicates that 102 is the identification number of the undefined coordinate system. The abbreviation UFM refers to User Fatal Message, UWM refers to User Warning Message, UIM refers to User Information Message, and SFM refers to System Fatal Message.

UFM 2066	*** USER FATAL MESSAGE 2066, UNDEFINED GRID POINT **** ON DAREA CARD.
	A dynamic loading entry references an undefined grid point.
UFM 2069	*** USER FATAL MESSAGE 2069, UNDEFINED GRID POINT **** IN TRANSIENT INITIAL CONDITION SET ****.
	An attempt has been made to specify initial conditions for an undefined grid point. All degrees-of-freedom with initial conditions must be in the analysis set.

UFM 2071	*** USER FATAL MESSAGE 2071, DYNAMIC LOAD SET **** REFERENCES UNDEFINED TABLE ****.

A referenced dynamic load table was not present in the Bulk Data.

UFM 2074	*** USER FATAL MESSAGE 2074, UNDEFINED TRANSFER FUNCTION SET ****.

A transfer function set was selected in the Case Control but was not present in the Bulk Data.

UFM 2079	*** USER FATAL MESSAGE 2079, GRID OR SCALAR POINT **** HAS AN UNDEFINED COORDINATE REFERENCED ON A DAREA, DELAY, DPHASE CARD.

The "C" or component value for scalar-type points must be zero or one.

UFM 2088	*** USER FATAL MESSAGE 2088, DUPLICATE TABLE ID ****.

All tables must have unique numbers. Check for uniqueness.

UFM 2101A	*** USER FATAL MESSAGE 2101A, GRID POINT **** COMPONENT *** ILLEGALLY DEFINED IN SETS ****.

The above grid point and component are defined in each of the above independent subsets. A point may belong to a maximum of one independent subset. This error occurs when a DOF is defined as belonging to two mutually exclusive sets (see *NX Nastran User's Guide*).

A common example of this occurs when a DOF is defined as dependent on an MPC (M-set) as well as being constrained (s-set) on an SPC entry. The message for this states that the component is illegally defined in the um (user-defined m-set) and us (user-defined s-set) sets. These two sets are mutually exclusive because all MPC equations are processed before the SPCs are applied and the m-set DOFs are removed from the matrix. When the program attempts to apply the SPC, the DOF is no longer available, and the fatal message is issued. The normal correction for this is to modify the MPC so that the DOF in question is independent (n-set). Then there is no conflict.

UFM 2107	*** USER FATAL MESSAGE 2107, EIGR-CARD FROM SET **** REFERENCES DEPENDENT COORDINATE OR GRID POINT ****.

When the point option is used on an EIGR entry, the referenced point and component must be in the analysis set (a-set) for use in normalization.

UFM 2109	*** USER FATAL MESSAGE 2109, NO GRID, SCALAR OR EXTRA POINTS DEFINED.

Dynamics problems must have at least one grid, scalar, or extra point.

UFM 2133	*** USER FATAL MESSAGE 2133, INITIAL CONDITION IN SET **** SPECIFIED FOR POINT NOT IN ANALYSIS SET.

Initial conditions can only be specified for analysis set points. Therefore the point/component mentioned on TIC entries must belong to the analysis set.

UFM 2135	*** USER FATAL MESSAGE 2135, DLOAD CARD *** HAS A DUPLICATE SET ID FOR SET ID ***.

The Li Set IDs on a DLOAD entry are not unique. See the DLOAD Bulk Data description.

UFM 2136   *** USER FATAL MESSAGE 2136, DUPLICATE DLOAD, RLOAD, OR
TLOAD SET ID NUMBER = ****** HAS BEEN ENCOUNTERED FOR
DLOAD SET = ******.

Dynamic loads may not be defined by giving multiple data entries with the
same ID. Use unique IDs.

UIM 2141   *** USER INFORMATION MESSAGE 2141, TIME ESTIMATE IS ****
SECONDS. PROBLEM SIZE IS ****, SPILL WILL OCCUR FOR THIS CORE
AT A PROBLEM SIZE OF ****.

The time estimate includes the time of the tridiagonalization and eigenvalue
calculation when the HOU method is used. If ND is given on the EIGR entry,
it also includes the time of the eigenvector generation. If F1 and F2 are
used instead, the eigenvector times are not estimated. This condition can
underestimate the time when the range includes many eigenvectors.

UFM 2200   *** USER FATAL MESSAGE 2200, INCONSISTENT RIGID BODY SYSTEM.

This message occurs if a SUPORT is used and the rigid-body mass matrix is
not positive definite. Possible causes are unconstrained mechanisms or input
of negative mass terms. A diagnostic method is to remove all SUPORT entries
and inspect the resulting eigenvectors for implausible behavior.

UFM 3031   *** USER FATAL MESSAGE 3031, UNABLE TO FIND SELECTED SET
(****) IN TABLE (****) IN SUBROUTINE (****).

A particular set used in the problem was not included in the data. Good
examples are loads, initial conditions, or frequency sets. Include the required
data or change the Case Control commands to select data already in the
problem. Set zero (0) has a special meaning. A set selection was required, but
none was made. For example, no METHOD was selected for an eigenvalue
extraction problem, or no FREQ was selected for frequency response.

This message can also indicate that a LOAD entry references another LOAD
entry, which is not permitted.

This message can also occur if a DLOAD entry references a nonexisting
LOAD entry, e.g., RLOAD1.

UWM 3034   *** USER WARNING MESSAGE 3034, ORTHOGONALITY TEST FAILED.
LARGEST TERM = ****, NUMBER FAILED = ****, PAIR = ****, ****,
EPSILON = ****.

This indicates that eigenvector accuracy is in doubt. This message is printed
only when the off-diagonal terms of the modal mass matrix are larger
than 1.0E-10. The eigenvectors are not orthogonal to this extent. This
nonorthogonality is especially important if a modal formulation is used. The
pair of eigenvectors listed exhibit the worst behavior. The number failed
is the number of pairs above the criteria. You can improve the numerical
conditioning of the problem by reducing the range of mass ratios, stiffness
ratios, and eigenvalue range.

UIM 3035   *** USER INFORMATION MESSAGE 3035, FOR DATA BLOCKS ****. SUPORT PT. NO. EPSILON STRAIN ENERGY EPSILONS LARGER THAN .001 ARE FLAGGED WITH ASTERISKS.

One line of output is printed for each component on a SUPORT entry. Large values of either EPSILON or STRAIN ENERGY indicate errors in the constraints, MPCs, or offsets.

UWM 3045   *** USER WARNING MESSAGE 3045, INSUFFICIENT TIME TO COMPLETE THE REMAINING ** SOLUTION(S) IN MODULE ***.

The estimated time for completion of the module is less than the time remaining, as specified on the Executive Control TIME statement. The module computes one solution (for example, one excitation frequency in frequency response analysis) and then processes all output requests. The remaining frequencies can be obtained on restart by adding or changing a FREQ command.

UFM 3046   *** USER FATAL MESSAGE 3046, YOUR SELECTED LOADING CONDITION, INITIAL CONDITION, AND NONLINEAR FORCES ARE NULL. A ZERO SOLUTION WILL RESULT.

Transient solutions must have one of the above nonzero loading condition, initial condition, or nonlinear forces. Also, make sure that LOADSET is spelled correctly.

UFM 3047   *** USER FATAL MESSAGE 3047, NO MODES WITHIN RANGE AND LMODES=0. A MODAL FORMULATION CANNOT BE MADE.

The modes used for a modal formulation must be selected by a PARAM entry. Set LFREQ, HFREQ, or LMODES to request modes.

UFM 3051   *** USER FATAL MESSAGE 3051, INITIAL CONDITION SET **** WAS SELECTED FOR A MODAL TRANSIENT PROBLEM. INITIAL CONDITIONS ARE NOT ALLOWED IN SUCH A PROBLEM.

The IC command is not allowed for modal transient response problems.

UWM 3053   *** USER WARNING MESSAGE 3053, THE ACCURACY OF EIGENVALUE **** IS IN DOUBT. HOU QR FAILED TO CONVERGE IN **** ITERATIONS.

Each eigenvalue is computed to the precision limits of each machine consistent with the maximum number of iterations allowed.

UFM 3057   *** USER FATAL MESSAGE 3057, MATRIX **** IS NOT POSITIVE DEFINITE.

A Cholesky decomposition was attempted on the above matrix, but a diagonal term of the factor was imaginary or equal to zero such that the decomposition failed. This message is from the regular (as opposed to sparse) decomposition method.

This message may be produced because of constraint problems. Check the output for UWM 4698 for large factor diagonal ratios and constrain appropriately.

UWM 4193   *** USER WARNING MESSAGE 4193, A GRID AND COMPONENT SPECIFICATION ON A (DPHASE/DELAY) SID = **, DOES NOT APPEAR ON A DAREA CARD.

The area specification is set to zero.

SFM 4276   *** SYSTEM FATAL MESSAGE 4276, **** ERROR CODE ****

This message occurs when NX Nastran encounters errors that are not otherwise trapped (including system errors). There are various error codes (EC), each of which has a different meaning.

In nearly every case, the log file contains further information about the problem, so you should look there for further clues. In addition, because these are errors that most likely should be caught in another manner (and with a more explicit error message), you should look at the recent Error Reports (search for "4276").

In many cases, increasing memory, BUFFSIZE, or disk space resolves the problem, especially when attempting to run large models on workstations.

This error often indicates machine underflow or overflow, although it is impossible to list specific reasons for this error code. Observed problems include:

- Modeling problems. This condition is usually accompanied by an "arithmetic fault, floating overflow" type message in the log file on some machines. Perform a static analysis and verify that the value for the maximum factor diagonal ratio is acceptable.

- When SFM 4276 is followed by an "access violation", then it is often due to a lack of memory or disk space or due to a coding error in NX Nastran.

UWM 4312   *** USER WARNING MESSAGE 4312, CONM2 **** HAS NONPOSITIVE-DEFINITE INERTIA MATRIX.

Most dynamic analysis methods require positive-definite mass matrices. Inserting inertia data into the wrong fields can result in nonpositive-definite systems, which are not physically realistic for normal modeling practices. Reduce the size of the off-diagonal terms to provide a positive-definite determinant.

UFM 4346   *** USER FATAL MESSAGE 4346, FREQUENCY RESPONSE SET, ID = *** IS UNDEFINED.

Define the set of frequencies to be used for the analysis.

UFM 4391   *** USER FATAL MESSAGE 4391, NONUNIQUE DAREA SET ***** HAS BEEN SPECIFIED FOR LSEQ DEFINED VECTOR *****.

Each LSEQ Bulk Data entry must define a unique DAREA set specification.

UFM 4392   *** USER FATAL MESSAGE 4392, CONTINUATION CARD ERRORS. EXPLANATIONS FOLLOW LIST OF CARDS IN ERROR.

ERROR NUMBER .1..2..3... *** Input echo

EXPLANATION OF ERROR CODES ABOVE FOLLOWS...

1. FIELD 1 IS NOT UNIQUE.

2. MORE THAN ONE CARD HAS FIELD 10 WHICH IS THE SAME AS FIELD 1 OF THIS CARD.

3. CARD IS AN ORPHAN (I.E., NO PARENT CARD EXISTS).

Continuation mnemonics in field 10 of a parent entry and field 1 of its continuation entry must be unique. Each continuation entry must have a parent entry. Check all continuation mnemonics.

UFM 4405   *** USER FATAL MESSAGE 4405, NO EIGENVECTORS COMPUTED FOR COMPONENT MODE SYNTHESIS OR SYSTEM SOLUTION.

The eigenvectors computed in component mode synthesis (CMS) are used to approximate the motion of the component. Some vectors must be present to perform this reduction. For the system solution, an exit is taken if eigenvalues are requested but not eigenvectors. Reset the "range of frequency" and/or the "number desired" on the EIGR or EIGRL entry. If the superelement does not have any eigenvalues in the range of interest, remove the CMS request for that particular superelement.

This message is also issued when the eigenvectors calculated with a transformation method do not pass internal orthogonality checks. This is indicative of a modeling error.

This message can also be issued if insufficient memory is available for the Lanczos method with sparse decomposition.

This could occur with UFM 5401 and be related to UWM 5411.

UFM 4407   *** USER FATAL MESSAGE 4407, MR MATRIX HAS NULL DIAGONAL TERM.

The MR matrix contains the rigid-body mass matrix of the structure as measured at the degrees-of-freedom listed on the SUPORT entry. If any of these degrees-of-freedom have null mass, they result in invalid eigenvectors. Specify enough masses to define all rigid-body modes and check attachments between the SUPORT and the rest of the model.

UIM 4415   *** USER INFORMATION MESSAGE 4415, THE FOLLOWING A-SET DEGREES-OF-FREEDOM HAVE EITHER NULL MASSES OR NULL MASSES AND STIFFNESSES.

If the listed degrees-of-freedom have null mass for the HOU or MHOU methods, they are automatically omitted. For direct frequency or direct transient response, the null degrees-of-freedom are given a very small mass or stiffness. Inspect the listed degrees-of-freedom to ensure that masses or stiffnesses are not left out inadvertently.

UFM 4416   *** USER FATAL MESSAGE 4416, NO DYNAMIC LOAD TABLE AVAILABLE.

A frequency response or transient response analysis was requested, but no dynamic load data is available. Include dynamic load data (DLOAD, RLOADi, TLOADi) in model.

UFM 4417   *** USER FATAL MESSAGE 4417, NO TRANSIENT RESPONSE LIST AVAILABLE.

A transient response dynamic analysis was requested, but no transient response list is available. Include a TSTEP entry in the Bulk Data Section.

UFM 4418   *** USER FATAL MESSAGE 4418, NO EIGENVALUE EXTRACTION DATA IS AVAILABLE.

A dynamic analysis was requested, but no eigenvalue extraction data was available. Include eigenvalue extraction data EIGR or EIGRL in the Bulk Data.

Possible causes are

- The METHOD command in the Case Control Section but no EIGR or EIGRL entry in the Bulk Data Section.

- No correspondence between Set IDs on the METHOD command and the EIGR/EIGRL entries.

UWM 4420   *** USER WARNING MESSAGE 4420, THE FOLLOWING DEGREES-OF-FREEDOM ARE POTENTIALLY SINGULAR.

During decomposition, the degrees-of-freedom listed had pivot ratios greater than MAXRATIO. Verify that the degrees-of-freedom are not part of a mechanism and that elements do not have excessive stiffness. In SOLs 100 and higher, this condition causes run termination. PARAM,BAILOUT may be used to continue the run. See the *NX Nastran Numerical Methods User's Guide*.

UFM 4421   *** USER FATAL MESSAGE 4421, NO FREQUENCY RESPONSE LIST AVAILABLE.

A frequency response dynamic analysis was requested, but no frequency data is available. Include frequency data (FREQ, FREQ1, FREQ2) in the Bulk Data.

UFM 4501   *** USER FATAL MESSAGE 4501, RLOADi CARD SELECTED IN TRANSIENT ANALYSIS. USE TLOADi.

RLOADi entries are used in frequency response analysis. These entries have no meaning in transient analysis. Replace RLOADi with TLOADi entries.

UWM 4561   *** USER WARNING MESSAGE 4561, INSUFFICIENT MEMORY FOR MODE ORTHOGONALITY CHECKS.

The amount of memory needed for eigenvector orthogonalization is $1/2 \cdot$ [number of eigenvectors $\cdot$ (number of eigenvectors $+1$)] $+ 2 \cdot$ BUFFSIZE + number of eigenvalues. If this equation is not met, the modes are orthogonalized, but the checking function is not performed. However, all outputs from the module are provided. If the check is desired, you should either increase memory or decrease the number of eigenvectors to satisfy the above equation.

UFM 4562   *** USER FATAL MESSAGE 4562, TSTEP (TIME STEPS) DATA IS MISSING.

Transient analysis requires the time step data. Add a TSTEP Bulk Data entry, and select it with a TSTEP Case Control command.

UWM 4582   *** USER WARNING MESSAGE 4582, LSEQ CARD SID = *** REFERS TO A NONEXISTENT STATIC LOAD MATRIX COLUMN *** (NCOLS = ***).

The most likely cause occurs when changing an LSEQ entry on a restart without regenerating and assembling the static load matrix.

UFM 4603	*** USER FATAL MESSAGE 4603, THE LSEQ SET ID **** IS NOT UNIQUE WITH RESPECT TO OTHER STATIC LOAD IDS.

LSEQ set IDs must be unique with respect to all other static load set IDs.

UFM 4645	*** USER FATAL MESSAGE 4645, THE SHIFTED STIFFNESS MATRIX IS NOT POSITIVE DEFINITE.

The matrix sum $K + \lambda \cdot M$ is decomposed by the Cholesky method at the start of the MGIV method of eigensolution. This decomposition requires that the matrix be positive-definite. A condition that prevents this is a massless mechanism (for example, a point mass on an offset with no rotational stiffness).

UFM 4646	*** USER FATAL MESSAGE 4646, THE MASS MATRIX IS NOT POSITIVE DEFINITE, USING THE HOU METHOD. USE MHOU INSTEAD.

The reduced mass matrix has columns that are not linearly independent. Common causes are rotation degrees-of-freedom whose only inertia terms result from point masses on offsets. Use the MHOU method instead since it does not require a positive definite mass matrix.

UFM 4647	*** USER FATAL MESSAGE 4647, INSUFFICIENT TIME TO COMPLETE *****.

CPU ESTIMATE = **** SEC CPU REMAINING = **** SEC I/O ESTIMATE = **** SEC I/O REMAINING = **** SEC

CPU and I/O limits are supplied on the Executive Control statement TIME (in minutes). The module where the terminated program is listed. If the time to completion appears reasonable, you should increase the estimates on the TIME statement and resubmit the run. For large models, an increase in the system memory request should also be considered.

UWM 4648	*** USER WARNING MESSAGE 4648, THE MODAL MASS MATRIX IS NOT POSITIVE DEFINITE.

The modal mass matrix cannot be decomposed by the Cholesky algorithm after merging elastic and free-body modes. (Cholesky decomposition is used to orthogonalize the eigenvectors with respect to the mass matrix.) The causes include the input of negative masses and the calculation of eigenvectors for eigenvalues approaching machine infinity. Inspect the model or ask for fewer eigenvectors using the F2 option. When this condition occurs, the eigenvectors are not orthogonalized or normalized. The second parameter of the READ or REIGL module is given a negative sign. This parameter is used in the solution sequences to branch to an error exit after printing the real eigenvalue table. You may use a DMAP Alter to print these eigenvectors if the cause of the problem is not apparent in the eigenvalues. The solution can be forced to completion by changing the sign of this parameter. You should be aware that a poor-quality solution is provided for this case. This poor solution may be useful for diagnosing the problem but should not be used for other purposes.

A possible cause of this error is when large offsets (large relative to the element length) are used for the BEAM element and coupled mass is selected.

UFM 4671	*** USER FATAL MESSAGE 4671, LOAD COMBINATION REQUESTED BUT LSEQ CARDS DO NOT EXIST FOR SID = ****.

Check the LSEQ entries.

UFM 4683 *** USER FATAL MESSAGE 4683, MASS/STIFFNESS MATRIX NEEDED FOR EIGENVALUE ANALYSIS.

The eigensolution module was given a purged (that is, nonexistent) mass or stiffness matrix. Common causes include the deletion of mass density input on MATi entries, user restart errors in the superelement solution sequences. It therefore sets the number of generalized coordinates to zero. This condition can be detected from UIM 4181. Provide mass matrix generating data by any of several means including a mass density entry on material entries, concentrated masses, and g-type DMIG entries.

Possibly no mass matrix is defined. Check for the following:

- RHO entry on MATi

- NSM entry on element properties (i.e., PSHELL, PBAR)

- CONMi or CMASSi

NX Nastran needs at least one of the above to compute the mass matrix. Incorrect cross-sectional properties may also lead to this error but typically show up as another error.

UWM 4698 *** USER WARNING MESSAGE 4698, STATISTICS FOR DECOMPOSITION OF MATRIX ****. THE FOLLOWING DEGREES-OF-FREEDOM HAVE FACTOR DIAGONAL RATIOS GREATER THAN ****, OR HAVE NEGATIVE TERMS ON THE FACTOR DIAGONAL.

During decomposition, the degrees-of-freedom listed have pivot ratios that are greater than maxratio or are negative. Verify that the degrees-of-freedom are not part of a mechanism and that elements do not have excessive stiffness. In SOLs 100 and higher this condition causes run termination. PARAM,BAILOUT may be used to continue the run to obtain messages issued by subsequent modules. See the *NX Nastran Numerical Methods User's Guide*.

UIM 5010 *** USER INFORMATION MESSAGE 5010, STURM SEQUENCE DATA FOR EIGENVALUE EXTRACTION. TRIAL EIGENVALUE = (real), CYCLES = (real), NUMBER OF EIGENVALUES BELOW THIS VALUE = (integer).

This message is automatic output during eigenvalue extraction using the Lanczos and SINV methods. This message can be used, along with the list of eigenvalues, to identify the modes found. See the *NX Nastran Numerical Methods User's Guide*.

UFM 5025 *** USER FATAL MESSAGE 5025, LAMA PURGED. DSTA MODULE TERMINATED.

The LAMA data block contains a list of natural frequencies and may be purged because no eigenvalues were computed or the data block was not properly recovered on restart.

UIM 5218	*** USER INFORMATION MESSAGE 5218, EIGENVALUE APPROACHING INFINITY AT **** TH MODE. EIGENVECTORS WILL NOT BE COMPUTED BEYOND THIS POINT.

The MHOU and AHOU methods substitute a very large number for eigenvalues that approach machine infinity. If eigenvectors are computed for these artificial values, they may be numerical noise, or they may cause overflows. Eigenvector computation is halted at the first machine infinity instead even if you requested eigenvectors in this range.

UIM 5222	*** USER INFORMATION MESSAGE 5222, COUPLED/UNCOUPLED SOLUTION ALGORITHM USED.

The modal methods use uncoupled solution algorithms, if possible. The uncoupled algorithms are considerably more economical than the coupled algorithms. Coupled algorithms are required when any of the following effects are present: transfer functions, DMIG requests of the p-type, element damping, and PARAM,G.

Consider the use of modal damping (TABDMP1 entry) to reduce the cost of your analysis in modal solutions.

UFM 5225	*** USER FATAL MESSAGE 5225, ATTEMPT TO OPERATE ON THE SINGULAR MATRIX **** IN SUBROUTINE DCMP.

This message is preceded by the listing of the grid point ID and degrees-of-freedom for any null columns.

UIM 5236	*** USER INFORMATION MESSAGE 5236, THE FREQUENCY RANGE HAS BEEN SPLIT INTO **** SUBREGIONS.

The overall frequency range for eigenanalysis is split into several smaller ranges when using the SINV option to calculate modes and frequencies.

UFM 5238	*** USER FATAL MESSAGE 5238, THE NUMBER OF ROOTS IN THE DEFINED FREQUENCY RANGE IS GREATER THAN 600.

More than 600 roots are in the desired frequency range, which is greater than the maximum allowed using SINV. Decrease the size of the frequency range.

UIM 5239	*** USER INFORMATION MESSAGE 5239, BISECTIONING IN THE ***-*** INTERVAL.

The frequency subregion encompassing eigenvalues xx-yy is cut in half in order to find the remaining roots.

UIM 5240	*** USER INFORMATION MESSAGE 5240, THE BISECTION VALUE IS: ****

The selected value is midway between the lowest and highest frequencies in the frequency subregion.

UIM 5241	*** USER INFORMATION MESSAGE 5241, MISSING ROOT(S) IN THE ***-*** INTERVAL.

The Sturm sequence check has indicated that roots are missing in the frequency range, and they cannot be found by further bisectioning.

If the run terminates with missing roots, decrease the frequency range.

UIM 5242   *** USER INFORMATION MESSAGE 5242, THE ROOT FOUND IS NOT THE LOWEST ONE ABOVE FMIN.

The Sturm sequence check indicates that at least one unfounded root exists between FMIN and the lowest frequency root found. Set FMAX close to the lowest frequency found, so that the lower roots can be found.

UIM 5274   *** USER INFORMATION MESSAGE 5274, THE ACTUAL TIME OF (****) TRIDIAGONALIZATION IS: ****, THE ACTUAL TIME OF EIGENVALUE ITERATION IS: ****, THE ACTUAL TIME OF EIGENVECTOR GENERATION IS: ****.

The time spent in the major operations of the real eigensolution module are output. Note that the number of eigenvectors requested has a large effect on solution cost.

UFM 5288   *** USER FATAL MESSAGE 5288, NO ROOT EXISTS ABOVE FMIN.

This message occurs when the eigenproblem of finding any number of roots above FMIN is expected, but the Sturm number indicates that there is no root above FMIN. Reduce FMIN in order to attempt to find a root.

SFM 5299   *** SYSTEM FATAL MESSAGE 5299 (This text varies depending on the reason for termination; see the description given below.)

1. Insufficient storage for the Lanczos method.

2. Factorization error on three consecutive shifts.

3. Stack overflow in the Lanczos method.

4. Unrecoverable termination from the Lanczos method.

5. Insufficient working storage.

6. Finite interval analysis error (see UIM 6361).

This message can also occur for models with two or more widely separated groups of repeated roots. An avoidance is to search each group separately. This error may also be caused by a massless mechanism, which can be confirmed by performing a static analysis.

See the *NX Nastran Numerical Methods User's Guide* for more information.

UFM 5400   *** USER FATAL MESSAGE 5400, INCORRECT RELATIONSHIP BETWEEN FREQUENCY LIMITS.

You have incorrectly specified V1 > V2. Check V1, V2 specified on the EIGRL Bulk Data entry.

SFM 5401   *** SYSTEM FATAL MESSAGE 5401 (REIGL), LANCZOS METHOD
                IS UNABLE TO FIND ALL EIGENVALUES IN RANGE. ACCEPTED
                EIGENVALUES AND ADDITIONAL ERROR MESSAGES MAY BE
                LISTED ABOVE. USER ACTION: RERUN WITH ANOTHER METHOD OR
                ANOTHER SETTING ON EIGRL ENTRY.

                This message can be issued if insufficient memory is available for Lanczos
                with sparse decomposition. It can also be issued if UFM 5299 occurs. See the
                NX Nastran Numerical Methods User's Guide.

                This condition can be related to the occurrence of UWM 5411.

UWM 5402   *** USER WARNING MESSAGE 5402, —THE PROBLEM HAS NO
                STIFFNESS MATRIX.

                The problem requires a stiffness matrix. Verify that property entries are
                specified correctly.

UIM 5403   *** USER INFORMATION MESSAGE 5403, CPU TIME AT START OF
                LANCZOS ITERATION ****.

                Since several Lanczos iterations may be executed during one application of
                the Lanczos method (each shift is followed by at least one iteration), this
                information is given to measure the time required for the individual iterations.

UWM 5404   *** USER WARNING MESSAGE 5404, NEGATIVE MODAL MASS
                TERM, IS ENCOUNTERED DURING INVERSE ITERATION. PROCESS
                ABORTED.

                The modal mass matrix should have unit diagonal terms (for mass
                normalization). Negative terms may indicate negative eigenvalues. If these
                negative terms are computational zeroes (rigid-body modes, for example),
                then the negative terms are acceptable. If the negative terms are finite values,
                there may be a modeling problem.

UWM 5405   *** USER WARNING MESSAGE 5405, ERROR OCCURRED DURING
                ITERATION. ERROR NUMBER IS : Y (SEE DESCRIPTION FOR VALUES
                OF Y AND USER ACTION.)

                This message marks the breakdown of the inverse iteration process in the
                Lanczos method. See the *NX Nastran Numerical Methods User's Guide* for
                additional values and actions.

	Y Value	User Action
-11	File open error in interface; see GINO error message.	This error should not occur; report error to UGS.
-12	File open error in postprocessing; see GINO error message.	This error should not occur; report error to UGS.
-13	File read error; see GINO error message.	This error should not occur; report error to UGS.
-21	Insufficient space for blocksize = 1.	Increase memory.

-22	Three consecutive factorizations failed at a shift.	Possible ill-conditioning; check model.
-23	Lanczos internal table overflow due to enormous number of shifts.	Specify smaller internal; may be necessary to have several runs.
-31	Internal error in Lanczos (REIGL) module.	This error should not occur; report error to UGS.
-32	No convergence in solving the tridiagonal problem.	Possible ill-conditioning; check model.
-33	Too many eigenvalues were found; inconsistency between the roots found and Sturm number.	Check the orthogonality of the eigenvectors; if it is good, then ignore this warning.

UWM 5406   *** USER WARNING MESSAGE 5406 NO CONVERGENCE IN SOLVING THE TRIDIAGONAL PROBLEM.

This message signals eigensolution problems in the Lanczos method. There is possible ill-conditioning; check your model.

UWM 5407   *** USER WARNING MESSAGE 5407, INERTIA (STURM SEQUENCE) COUNT DISAGREES WITH THE NUMBER OF MODES ACTUALLY COMPUTED IN AN (SUB) INTERVAL

This message shows a serious problem. Spurious modes were found in the Lanczos method. Check the multiplicity of the roots given in the interval. See the *NX Nastran Numerical Methods User's Guide*.

UWM 5408   *** USER WARNING MESSAGE 5408, FACTORIZATION FAILED. SHIFT CHANGED TO ****.

No user action to be taken. This message occurs only for the Lanczos method.

UWM 5411   *** USER WARNING MESSAGE 5411, NEGATIVE TERM ON DIAGONAL OF MASS MATRIX (VIBRATION) OR STIFFNESS (BUCKLING), ROW ****, VALUE = ****

The message is given from the REIGL module which performs a necessary (but not sufficient) check on the positive semi-definiteness of the indicated matrix. Look for evidence of negative mass, such as minus signs on input. Negative terms on the factor of the indicated matrix must be removed for correct answers.

Something has caused a negative term on the diagonal of the mass or stiffness matrix. Look for explicitly defined negative mass and/or stiffness terms. Also, check the continuation entries on the PBEAM entry. An incorrect entry for the SO field may lead to improper mass definition. For example, if SO is set to NO at a particular X/XB location, the continuation entry for defining four stress locations on the cross section (C, D, E, F) is not used. If SO is NO but the C, D, E, and F points are entered in error, negative mass terms can result if either E1 or E2 entries are entered. The offending DOF can be traced using the USET tables.

The Lanczos method gives wrong answers for indefinite matrices. The existence of negative diagonal terms indicates a subclass of indefinite matrix.

See the *NX Nastran Numerical Methods User's Guide* for more information.

UIM 5458	*** USER INFORMATION MESSAGE 5458, (****) METHOD IS SELECTED. or (****) METHOD IS (****)

The exact text of this message depends on the METHOD field on the selected EIGR Bulk Data entry. This message indicates the eigensolution status (all eigenvalues found, not all found, etc.).

UFM 6133	*** USER FATAL MESSAGE 6133 (DFMSDD), SINGULAR MATRIX IN SPARSE DECOMPOSITION.

USER ACTION: CHECK MODEL.

This message is often followed by UFM 4645, UFM 4646, or UWM 4648.

UFM 6134	*** USER FATAL MESSAGE 6134 (DFMSDD), MATRIX IS NOT POSITIVE DEFINITE IN SPARSE DECOMPOSITION.

USER ACTION: CHECK MODEL.

This message is often followed by UFM 4645, UFM 4646, or UWM 4648.

SFM 6135	*** SYSTEM FATAL MESSAGE 6135, ERROR IN READING SYMBOLIC FACTOR IN SPARSE FBS.

This message may be issued if the FBS module is using a sparse method to solve factors which are not decomposed by the sparse method. This message can also be caused by a compatibility or database integrity problem.

UFM 6136	*** USER FATAL MESSAGE 6136 (****), INSUFFICIENT CORE FOR (SYMBOLIC/NUMERIC) PHASE OF SPARSE DECOMPOSITION.

USER ACTION: INCREASE CORE BY **** WORDS.

USER INFORMATION: !!! NOW REVERTING BACK TO ACTIVE COLUMN DECOMPOSITION UPON USER REQUEST !!!

If this message is issued in the symbolic phase, the memory estimate is not necessarily conservative and even more memory may be required (although this estimate is fairly accurate for Version 68). Also, the memory increase required is only for the symbolic phase. It is not unusual for the decomposition phase to require more memory than the symbolic phase. To increase the chances for a successful run, increase the memory even more than the amount indicated in this message. After the run is complete, determine the amount of memory actually used and use this as a guideline for similar runs in the future.

The user information message is written if SYSTEM(166) = 1 (that is, if there is not enough memory for sparse decomposition, and you should switch to regular decomposition).

UWM 6137	*** USER WARNING MESSAGE 6137 (DFMSDD), INPUT MATRIX IS RANK DEFICIENT, RANK = ****.

USER ACTION: CHECK MODEL.

One of your matrices is singular. See the *NX Nastran Numerical Methods User's Guide* for a discussion of singularity.

UFM 6138   *** USER FATAL MESSAGE 6138 (DFMSB), INSUFFICIENT CORE FOR SPARSE FBS.

USER ACTION: INCREASE CORE BY **** WORDS.

See UFM 6136.

UIM 6214   *** USER INFORMATION MESSAGE 6214, FEWER THAN REQUESTED VECTORS CALCULATED, DUE TO INSUFFICIENT TIME.

This information message occurs in the READ module when there is insufficient time to compute eigenvectors. Resubmit the job with an increased time limit (TIME).

UWM 6243   *** USER WARNING MESSAGE 6243 (READ) —- THE DEGREE OF FREEDOM (D.O.F) REQUESTED FOR POINT NORMALIZATION HAS NOT BEEN SPECIFIED ON THE EIGR OR EIGB ENTRY.

USER INFORMATION: THE D.O.F PRECEDING THE REQUESTED D.O.F. IN THE INTERNAL SEQUENCE LIST WILL BE USED

The point requested was not in the a-set, so another point was chosen.

UIM 6361   *** USER INFORMATION MESSAGE 6361-LANCZOS MODULE DIAGNOSTICS

This message prints various levels of diagnostics for the Lanczos method. The amount of print depends on the message level set on the EIGRL entry. See the *NX Nastran Numerical Methods User's Guide* for more information.

UIM 6480   *** USER INFORMATION MESSAGE 6480 (REIGLA) — EXTERNAL IDENTIFICATION TABLE FOR DECOMPOSITION MESSAGES FOR MATRIX **** ROW NUMBER **** = GRID ID **** + COMPONENT ****

This message is output from the REIGL module when using sparse decomposition to convert the internal (row number oriented) diagnostic messages to external (grid and component) form.

Appendix

I   References and Bibliography

I.1   Overview

This appendix includes references of interest in the field of dynamic analysis. Two categories are included. The first category, General References, lists books that cover the general range of structural dynamic analysis. The second category, Bibliography, is an excerpt from the dynamic analysis section of the NX Nastran Bibliography.

I.2   General References

1.   Paz, M., *Structural Dynamics: Theory and Computation*, Van Nostrand Reinhold, New York, N.Y., 1985.

2.   Bathe, K. J. and Wilson, E. L., *Numerical Methods in Finite Element Analysis*, Prentice-Hall, Englewood Cliffs, N.J., 1976.

3.   Harris, C. M. and Crede, C. E., *Shock and Vibration Handbook*, McGraw-Hill, New York, N.Y., 1976.

4.   Clough, R. W. and Penzien, J., *Dynamics of Structures*, McGraw-Hill, New York, N.Y., 1975.

5.   Timoshenko, S., Young, D. H., and Weaver Jr., W., *Vibration Problems in Engineering*, John Wiley and Sons, New York, N.Y., 1974.

6.   Hurty, W. C. and Rubinstein, M. F., *Dynamics of Structures*, Prentice-Hall, Englewood Cliffs, N.J., 1964.

I.3   Bibliography

DYNAMICS – GENERAL

Abdallah, Ayman A.; Barnett, Alan R.; Widrick, Timothy W.; Manella, Richard T.; Miller, Robert P. *Stiffness-Generated Rigid-Body Mode Shapes for Lanczos Eigensolution with Support DOF Via a MSC/NASTRAN DMAP Alter*, MSC 1994 World Users' Conf. Proc., Paper No. 10, June, 1994.

Anderson, William J.; Kim, Ki-Ook; Zhi, Bingchen; Bernitsas, Michael M.; Hoff, Curtis; Cho, Kyu-Nam. *Nonlinear Perturbation Methods in Dynamic Redesign*, MSC/NASTRAN Users' Conf. Proc., Paper No. 16, March, 1983.

Barber, Pam; Arden, Kevin. *Dynamic Design Analysis Method (DDAM) Using MSC/NASTRAN*, MSC 1994 World Users' Conf. Proc., Paper No. 31, June, 1994.

Bedrossian, Herand; Veikos, Nicholas. *Rotor-Disk System Gyroscopic Effect in MSC/NASTRAN Dynamic Solutions*, MSC/NASTRAN Users' Conf. Proc., Paper No. 12, March, 1982.

Bernstein, Murray; Mason, Philip W.; Zalesak, Joseph; Gregory, David J.; Levy, Alvin. *NASTRAN Analysis of the 1/8-Scale Space Shuttle Dynamic Model*, NASTRAN: Users' Exper., pp. 169-242, September, 1973, (NASA TM X-2893).

Berthelon, T.; Capitaine, A. *Improvements for Interpretation of Structural Dynamics Calculation Using Effective Parameters for Substructures*, Proc. of the 18th MSC Eur. Users' Conf., Paper No. 9, June, 1991.

Birkholz, E. *Dynamic Investigation of Automobile Body Parts*, Proc. of the 15th MSC/NASTRAN Eur. Users' Conf., October, 1988.

Bishop, N. W. M.; Lack, L. W.; Li, T.; Kerr, S. C. *Analytical Fatigue Life Assessment of Vibration Induced Fatigue Damage*, MSC 1995 World Users' Conf. Proc., Paper No. 18, May, 1995.

Blakely, Ken; Howard, G. E.; Walton, W. B.; Johnson, B. A.; Chitty, D. E. *Pipe Damping Studies and Nonlinear Pipe Benchmarks from Snapback Tests at the Heissdampfreaktor*, NUREG/CR-3180, March, 1983.

Blakely, Ken. *Dynamic Analysis: Application and Modeling Considerations*, J. of Engineering Computing and Applications, Fall, 1987.

Bramante, A.; Paolozzi, A; Peroni, I. *Effective Mass Sensitivity: A DMAP Procedure*, MSC 1995 World Users' Conf. Proc., Paper No. 39, May, 1995.

Brutti, C.; Conte, M.; Linari, M. *Reduction of Dynamic Environment to Equivalent Static Loads by a NASTRAN DMAP Procedure*, MSC 1995 European Users' Conf. Proc., Italian Session, September, 1995.

Butler, Thomas G. *Dynamic Structural Responses to Rigid Base Acceleration*, Proc. of the Conf. on Finite Element Methods and Technology, Paper No. 8, March, 1981.

Butler, Thomas G. *Telescoping Robot Arms*, MSC/NASTRAN Users' Conf. Proc., Paper No. 10, March, 1984.

Butler, T. G. *Experience with Free Bodies*, Thirteenth NASTRAN Users' Colloq., pp. 378-388, May, 1985, (NASA CP-2373).

Butler, Thomas G. *Mass Modeling for Bars*, Fifteenth NASTRAN Users' Colloq., pp. 136-165, August, 1987, (NASA CP-2481).

Butler, T. G. *Coupled Mass for Prismatical Bars*, Sixteenth NASTRAN Users' Colloq., pp. 44-63, April, 1988, (NASA CP-2505).

Caldwell, Steve P.; Wang, B. P. *An Improved Approximate Method for Computing Eigenvector Derivatives in MSC/NASTRAN*, The MSC 1992 World Users' Conf. Proc., Vol. I, Paper No. 22, May, 1992.

Case, William R. *Dynamic Substructure Analysis of the International Ultraviolet Explorer (IUE) Spacecraft*, NASTRAN: Users' Exper., pp. 221-248, September, 1975, (NASA TM X-3278).

Chang, H. T.; Cao, Tim; Hua, Tuyen. *SSF Flexible Multi-Body Control/Structure Interaction Simulation*, The MSC 1993 World Users' Conf. Proc., Paper No. 15, May, 1993.

Chang, W. M.; Lai, J. S.; Chyuan, S. W.; *Application of the MSC/NASTRAN Design Optimization Capability to Identify Joint Dynamic Properties of Structure*, The Sixth Annual MSC Taiwan Users' Conf. Proc., Paper No. 1, November, 1994.

Chargin, M.; Miura, H.; Clifford, Gregory A. *Dynamic Response Optimization Using MSC/NASTRAN*, The MSC 1987 World Users Conf. Proc., Vol. I, Paper No. 14, March, 1987.

Chen, J. T.; Chyuan, S. W.; You, D. W.; Wong, H. T. A *New Method for Determining the Modal Participation Factor in Support Motion Problems Using MSC/NASTRAN*, The Seventh Annual MSC/NASTRAN Users' Conf. Proc., Taiwan, 1995.

Chen, J. T.; Wong, H. T. *Applications of Modal Reaction Method in Support Motion Problems*, Techniques in Civil Engineering, Vol. 4, pp 17 - 30, March, 1996, in Chinese.

Chen, J. T.; Hong, H. K.; Chyuan, S. W.; Yeh, C. S. *A Note on the Application of Large Mass and Large Stiffness Techniques for Multi-Support Motion*, The Fifth Annual MSC Taiwan Users' Conf. Proc., November, 1993.

Chen, Yohchia. *Improved Free-Field Analysis for Dynamic Medium-Structure Interaction Problems*, The MSC 1992 World Users' Conf. Proc., Vol. I, Paper No. 13, May, 1992.

Chen, Yohchia. *Dynamic Response of Reinforced Concrete Box-Type Structures*, The MSC 1992 World Users' Conf. Proc., Vol. I, Paper No. 24, May, 1992.

Chiu, Chi-Wai. *Spacecraft Dynamics During Solar Array Panel Deployment Motion*, The Fifth Annual MSC Taiwan Users' Conf. Proc., November, 1993.

Chung, Y. T.; Kahre, L. L. *A General Procedure for Finite Element Model Check and Model Identification*, MSC 1995 World Users' Conf. Proc., Paper No. 38, May, 1995.

Cicia, C. Static, *Thermal and Dynamic Analysis of the Liquid Argon Cryostat for the ICARUS Experiment*, Proc. of the 15th MSC/NASTRAN Eur. Users' Conf., October, 1988.

Cifuentes, Arturo O. *Dynamic Analysis of Railway Bridges Using MSC/NASTRAN*, The MSC 1988 World Users Conf. Proc., Vol. II, Paper No. 44, March, 1988.

Cifuentes, A. O. *Dynamic Response of a Beam Excited by a Moving Mass*, Finite Elements in Analysis and Design, Vol. 5, pp. 237-246, 1989.

Citerley, R. L.; Woytowitz, P. J. Ritz *Procedure for COSMIC/ NASTRAN*, Thirteenth NASTRAN Users' Colloq., pp. 225-233, May, 1985, (NASA CP-2373).

Ciuti, Gianluca. *Avionic Equipment Dynamic Analysis*, MSC 1995 European Users' Conf. Proc., Italian Session, September, 1995.

Coates, Dr. Tim; Matthews, Peter. *Transient Response in Dynamic and Thermal Behaviour*, The Second Australasian MSC Users Conf. Proc., Paper No. 4, November, 1988.

Coppolino, Robert N.; Bella, David F. *Employment of MSC/STI-VAMP for Dynamic Response Post-Processing*, The MSC 1987 World Users Conf. Proc., Vol. I, Paper No. 12, March, 1987.

Corder, P. R.; Persh, R. *Castigliano and Symbolic Programming in Finite Element Analysis*, Proceedings of the 16th Annual Energy - Sources Technology Conference and Exhibition, Houston, 1993.

Coyette, J. P.; Wijker, J. J. *The Combined Use of MSC/NASTRAN and Sysnoise for Evaluating the Dynamic Behavior of Solar Array Panels*, Proc. of the 20th MSC European Users' Conf., Paper No. 16, September, 1993.

Curti, G.; Chiandussi, G.; Scarpa, F. *Calculation of Eigenvalue Derivatives of Acousto-Structural Systems with a Numerical Comparison*, MSC 23rd European Users' Conf. Proc., Italian Session, September, 1996.

Defosse, H.; Sergent, A. *Vibro-Acoustic Modal Response Analysis of Aerospace Structures*, Proc. of the MSC/NASTRAN Eur. Users' Conf., April, 1985.

Deloo, Ph.; Dunne, L.; Klein, M. *Alter DMAPS for the Generation, Assembly and Recovery of Craig-Bampton Models in Dynamic Analyses*, Actes de la 2ème Confèrence Française Utilisateurs des Logiciels MSC, Toulouse, France, September, 1995.

Denver, Richard E.; Menichello, Joseph M. *Alternate Approaches to Vibration and Shock Analysis Using NASTRAN*, Sixth NASTRAN Users' Colloq., pp. 199-212, October, 1977, (NASA CP-2018).

Detroux, P.; Geraets, L. H. *Instability at Restart or Change of Time Step with NASTRAN in the Presence of Nonlinear Loads*, Proc. of the MSC/NASTRAN Eur. Users' Conf., June, 1983.

Deuermeyer, D. W.; Clifford, G. A.; Petesch, D. J. *Traditional Finite Element Analysis: Opportunities for Parallelism?*, Computing Systems in Engineering, Vol. 2, No. 2-3, pp. 157-165, 1991.

Dirschmid, Dr. W.; Nolte, Dr. F.; Dunne, L. W. *Application of an FRF-Based Update Method to the Model Parameter Tuning of an Hydraulic Engine Mounting*, Proc. of the 18th MSC Eur. Users' Conf., Paper No. 10, June, 1991.

Drago, Raymond J.; Margasahayam, Ravi N. *Resonant Response of Helicopter Gears Using 3-D Finite Element Analysis*, The MSC 1988 World Users Conf. Proc., Vol. I, Paper No. 20, March, 1988.

Elchuri, V.; Smith, G. C. C.; Gallo, A. Michael. *An Alternative Method of Analysis for Base Accelerated Dynamic Response in NASTRAN*, Eleventh NASTRAN Users' Colloq., pp. 89-112, May, 1983.

Everstine, Gordon C.; Schroeder, Erwin A. *The Dynamic Analysis of Submerged Structures, NASTRAN: Users' Exper.*, pp. 419-430, September, 1975, (NASA TM X-3278).

Everstine, Gordon C. *Structural Analogies for Scalar Field Problems*, Int. J. for Numerical Methods in Engineering, Vol. 17, No. 3, pp. 471-476, March, 1981.

Everstine, G. C. *Dynamic Analysis of Fluid-Filled Piping Systems Using Finite Element Techniques*, J. of Pressure Vessel Technology, Vol. 108, pp. 57-61, February, 1986.

Flanigan, Christopher C. *Accurate and Efficient Mode Acceleration Data Recovery for Superelement Models*, The MSC 1988 World Users Conf. Proc., Vol. I, Paper No. 38, March, 1988.

Flanigan, Christopher C.; Manella, Richard T. *Advanced Coupled Loads Analysis Using MSC/NASTRAN*, The MSC 1991 World Users' Conf. Proc., Vol. I, Paper No. 14, March, 1991.

Fox, Gary L. *Solution of Enforced Boundary Motion in Direct Transient and Harmonic Problems*, Ninth NASTRAN Users' Colloq., pp. 96-105, October, 1980, (NASA CP-2151).

Geyer, A.; Schweiger, W. *Multiple Support Excitation for NASTRAN Piping Analysis*, Proc. of the MSC/NASTRAN Eur. Users' Conf., April, 1982.

Geyer, A.; Schweiger, W. *Aeroelastic and Stress Analysis of the CHIWEC Chinese Wind Energy Converter Using MSC/NASTRAN*, Proc. of the MSC/NASTRAN Eur. Users' Conf., April, 1985.

Ghofranian, S.; Dimmagio, O. D. *Space Station Dynamic Analysis with Active Control Systems Using MSC/NASTRAN*, The MSC 1988 World Users Conf. Proc., Vol. I, Paper No. 17, March, 1988.

Gibson, Warren C. *Experiences with Optimization Using ASD/NASOPT and MSC/NASTRAN for Structural Dynamics*, The MSC 1987 World Users Conf. Proc., Vol. I, Paper No. 13, March, 1987.

Gibson, Warren C.; Austin, Eric. *Analysis and Design of Damped Structures Using MSC/NASTRAN*, The MSC 1992 World Users' Conf. Proc., Vol. I, Paper No. 25, May, 1992.

Gielen, L.; Brughmans, M.; Petellat, C. *A Stepwise Approach for Fatigue Evaluation of Engine Accessories Prior to Prototyping Using Hybrid Modelling Technology*, MSC 1996 World Users' Conf. Proc., Vol. III, Paper No. 29, June, 1996.

Go, James Chi-Dian. *Structural Dynamic and Thermal Stress Analysis of Nuclear Reactor Vessel Support System, NASTRAN: Users' Exper.*, pp. 465-476, September, 1972, (NASA TM X-2637).

Grasso, A.; Tomaselli, L. *Whirling Speed Analysis of Multispool Systems*, Proc. of the MSC/NASTRAN Eur. Users' Conf., May, 1984.

Grimes, Roger G.; Lewis, John G.; Simon, Horst D.; Komzsik, Louis; Scott, David S. *Shifted Block Lanczos Algorithm in MSC/NASTRAN*, MSC/NASTRAN Users' Conf. Proc., Paper No. 12, March, 1985.

Harn, Wen-Ren; Lin, Shyang-Kuang; Chen, Jeng-Tzong. *Localization of Dynamic Model Modification Based on Constrained Minimization Method*, The 2nd Annual MSC Taiwan Users Conf., Paper No. 14, October, 1990.

Herting, David N.; Bella, David F.; Kimbrough, Patty A. *Finite Element Simulation of Coupled Automobile Engine Dynamics*, The MSC 1987 World Users Conf. Proc., Vol. I, Paper No. 10, March, 1987.

High, Gerald D. *An Iterative Method for Eigenvector Derivatives*, The MSC 1990 World Users Conf. Proc., Vol. I, Paper No. 17, March, 1990.

Hill, R. G. *Transient Analysis of an IVHM Grapple Impact Test*, NASTRAN: Users' Exper., pp. 161-178, September, 1972, (NASA TM X-2637).

Howells, R. W.; Sciarra, J. J. *Finite Element Analysis Using NASTRAN Applied to Helicopter Transmission Vibration/Noise Reduction*, NASTRAN: Users' Exper., pp. 321-340, September, 1975, (NASA TM X-3278).

Howlett, James T. *Applications of NASTRAN to Coupled Structural and Hydrodynamic Responses in Aircraft Hydraulic Systems*, NASTRAN: Users' Exper., pp. 407-420, September, 1971, (NASA TM X-2378).

Huang, S. L.; Rubin, H. *Static and Dynamic Analysis, F-14A Boron Horizontal Stabilizer*, NASTRAN: Users' Exper., pp. 251-264, September, 1971, (NASA TM X-2378).

Hurwitz, Myles M. *New Large Deflection Analysis for NASTRAN*, Sixth NASTRAN Users' Colloq., pp. 235-256, October, 1977, (NASA CP-2018).

Hussain, M. A.; Pu, S. L.; Lorensen, W. E. *Singular Plastic Element: NASTRAN Implementation and Application*, Sixth NASTRAN Users' Colloq., pp. 257-274, October, 1977, (NASA CP-2018).

Ishikawa, Masanori; Iwahara, Mitsuo; Nagamatsu, Akio. *Dynamic Optimization Applied to Engine Structure*, The MSC 1990 World Users Conf. Proc., Vol. I, Paper No. 31, March, 1990.

Iwahara, Mitsuo. *Dynamic Optimization Using Quasi Least Square Method*, The Fifth MSC/NASTRAN User's Conf. in Japan, October, 1987, in Japanese.

Jakovich, John; Van Benschoten, John. *SDRC SUPERTAB Interactive Graphics as a Front-End to MSC/NASTRAN Dynamic Analysis*, Proc. of the MSC/NASTRAN Users' Conf., March, 1979.

Jones, Gary K. *The Response of Shells to Distributed Random Loads Using NASTRAN*, NASTRAN: Users' Exper., pp. 393-406, September, 1971, (NASA TM X-2378).

Kalinowski, Anthony J. *Steady Solutions to Dynamically Loaded Periodic Structures*, Eighth NASTRAN Users' Colloq., pp. 131-164, October, 1979, (NASA CP-2131).

Kalinowski, A. J. *Solution Sensitivity and Accuracy Study of NASTRAN for Large Dynamic Problems Involving Structural Damping*, Ninth NASTRAN Users' Colloq., pp. 49-62, October, 1980, (NASA CP-2151).

Kasai, Manabu. *Real Eigenvalue Analysis by Modal Synthesis Method Taking Differential Stiffness into Account*, The First MSC/NASTRAN User's Conf. in Japan, October, 1983, in Japanese.

Kasai, Manabu. *Recovery Method for Components by DMAP of Constrained Modal Type*, The Fifth MSC/NASTRAN User's Conf. in Japan, October, 1987, in Japanese.

Kasai, Manabu. *DMAP Program for Modal Mass and Momentum*, The Sixth MSC/NASTRAN User's Conf. in Japan, October, 1988, in Japanese.

Kienholz, Dave K.; Johnson, Conor D.; Parekh, Jatin C. *Design Methods for Viscoelastically Damped Sandwich Plates*, AIAA/ASME/ASCE/AHS 24th Structures, Structural Dynamics and Materials Conf., Part 2, pp. 334-343, May, 1983.

Lambert, Nancy; Tucchio, Michael. *Ring Element Dynamic Stresses*, Ninth NASTRAN Users' Colloq., pp. 63-78, October, 1980, (NASA CP-2151).

Lee, Jyh-Chiang. *Investigation for the Large Stiffness Method*, The Fifth Annual MSC Taiwan Users' Conf. Proc., November, 1993.

Lee, Ting-Yuan; Lee, Jyh-Chiang. *Modal Analysis and Structural Modification for a Harpoon Launcher*, The Fifth Annual MSC Taiwan Users' Conf. Proc., November, 1993.

LeMaster, R. A.; Runyan, R. B. *Dynamic Certification of a Thrust-Measuring System for Large Solid Rocket Motors*, Eleventh NASTRAN Users' Colloq., pp. 207-225, May, 1983.

Lewis, J.; Komzsik, L. *Symmetric Generalized Eigenproblems in Structural Engineering*, SIAM Conf. on Applied Numerical Analysis, 1985.

Lin, Chih-Kai; Harn, Wen-Ren; Lin, Shyang-Kuang. *The Dynamic Response of Bridge Due to Passing of Vehicle*, The 2nd Annual MSC Taiwan Users Conf., Paper No. 6, October, 1990.

Lin, S. L.; Yang, T. W.; Chen, J. T. *MSC/NASTRAN Application in Inertia Relief*, The 1st MSC Taiwan Users' Conf., Paper No. 13, October, 1989, in Chinese.

Lin, Shan. *Time-Dependent Restrained Boundary Condition Simulation*, The MSC 1988 World Users Conf. Proc., Vol. I, Paper No. 9, March, 1988.

Lu, Ming-Ying; Yang, Joe-Ming. *Analysis of Static and Dynamic Responses on Ship Structures Under Wave Loadings*, The Sixth Annual MSC Taiwan Users' Conf. Proc., Paper No. 14, November, 1994.

Magari, P. J.; Shultz, L. A.; Murthy, V. R. *Dynamics of Helicopter Rotor Blades, Computers and Structures*, Vol. 29, No. 5, pp. 763-776, 1988.

Malcolm, D. J. *Dynamic Response of a Darrieus Rotor Wind Turbine Subject to Turbulent Flow*, Engineering Structures, Vol. 10, No. 2, pp. 125-134, April, 1988.

Maritan, M.; Micelli, D. *Dynamic Behaviour of a High-Speed Crankshaft*, MSC 23rd European Users' Conf. Proc., Italian Session, September, 1996.

Masters, Steven G. *Plant Troubleshooting with MSC/NASTRAN*, Proc. of the Conf. on Finite Element Methods and Technology, Paper No. 12, March, 1981.

Mastrorocco, David T. *Predicting Dynamic Environments for Space Structure Appendages*, The MSC 1992 World Users' Conf. Proc., Vol. II, Paper No. 56, May, 1992.

Mayer, Lee S.; Zeischka, Johann; Scherens, Marc; Maessen, Frank. *Analysis of Flexible Rotating Crankshaft with Flexible Engine Block* Using MSC/NASTRAN and DADS, MSC 1995 World Users' Conf. Proc., Paper No. 35, May, 1995.

McLaughlin, A. *Finite Element Dynamic Analysis of Production Aircraft*, 4th Eur. Rotorcraft and Powered Lift Aircraft Forum, Assoc. Ital di Aeronaut ed Astronaut, pp. 20.1-20.7, September, 1978.

Melli, R.; Rispoli, F.; Sciubba, E.; Tavani, F. *Structural and Thermal Analysis of Avionic Instruments for an Advanced Concept Helicopter*, Proc. of the 15th MSC/NASTRAN Eur. Users' Conf., October, 1988.

Mikami, Kouichi. *Dynamic Stress Analysis System for Ship's Hull Structure Under Wave Loads*, The Second MSC/NASTRAN User's Conf. in Japan, October, 1984, in Japanese.

Moharir, M. M. *NASTRAN Nonlinear Capabilities in Dynamic Solutions*, MSC/NASTRAN Users' Conf. Proc., Paper No. 9, March, 1985.

Moore, Gregory J.; Nagendra, Gopal K. *Dynamic Response Sensitivities in MSC/NASTRAN*, The MSC 1991 World Users' Conf. Proc., Vol. I, Paper No. 4, March, 1991.

Mulcahy, T. M.; Turula, P.; Chung, H.; Jendrzejczyk, A. *Analytical and Experimental Study of Two Concentric Cylinders Coupled by a Fluid Gap*, NASTRAN: Users' Exper., pp. 249-258, September, 1975, (NASA TM X-3278).

Murthy, P. L. N.; Chamis, C. C. *Dynamic Stress Analysis of Smooth and Notched Fiber Composite Flexural Specimens*, National Aeronautics and Space Administration, April, 1984, (NASA TM-83694).

Murthy, P. L. N.; Chamis, C. C. *Dynamic Stress Analysis of Smooth and Notched Fiber Composite Flexural Specimens*, Composite Materials: Testing and Design (Seventh Conf.), ASTM, pp. 368-391, 1986, (ASTM STP 893).

Neal, M. *Vibration Analysis of a Printed Wiring Board Assembly*, Proc. of the MSC/NASTRAN Eur. Users' Conf., May, 1984.

Nefske, D. J.; Sung, S. H. *Power Flow Finite Element Analysis of Dynamic Systems: Basic Theory and Application to Beams, American Soc. of Mechanical Engineers, Noise Control and Acoustics Division*, Vol. 3, pp. 47-54, December, 1987.

Nowak, Bill. *The Analysis of Structural Dynamic Effects on Image Motion in Laser Printers Using MSC/NASTRAN*, The MSC 1988 World Users Conf. Proc., Vol. I, Paper No. 10, March, 1988.

Nowak, Bill. *Structural Dynamics Analysis of Laser Printers, Sound and Vibration*, Vol. 23, No. 1, pp. 22-26, January, 1989.

Nowak, William J. *Dynamic Analysis of Optical Scan Systems Using MSC/NASTRAN*, Proc. of the Conf. on Finite Element Methods and Technology, Paper No. 10, March, 1981.

Nowak, William; James, Courtney. *Dynamic Modeling and Analysis of Spinning Polygon Assemblies Using MSC/NASTRAN*, The MSC 1993 World Users' Conf. Proc., Paper No. 66, May, 1993.

Oei, T. H.; Broerse, G. *Reduction of Forced Vibration Levels on Ro-Ro Car Ferry-Type Ships by Means of Minor Changes of the Inner Aft-Body Construction*, Proc. of the MSC/NASTRAN Eur. Users' Conf., April, 1982.

Ojalvo, I. U. *Extensions of MSC/NASTRAN to Solve Flexible Rotor Problems*, MSC/NASTRAN Users' Conf. Proc., Paper No. 13, March, 1982.

Palmieri, F. *Nonlinear Dynamic Analysis of STS Main Engine Heat Exchanger*, Proc. of the 15th MSC/NASTRAN Eur. Users' Conf., October, 1988.

Palmieri, F. W. *Analyzing Deployment of Spacecraft Appendages Using MSC/NASTRAN*, The MSC 1990 World Users Conf. Proc., Vol. I, Paper No. 6, March, 1990.

Pamidi, P. R.; Brown, W. K. *On Eigenvectors of Multiple Eigenvalues Obtained in NASTRAN*, NASTRAN: Users' Exper., pp. 285-300, September, 1975, (NASA TM X-3278).

Paolozzi, A. *Structural Dynamics Modification with MSC/NASTRAN*, Proc. of the 19th MSC European Users' Conf., Paper No. 14, September, 1992.

Parthasarathy, Alwar. *Force-Sum Method for Dynamic Stresses in MSC/NASTRAN Aeroelastic Analysis*, The MSC 1991 World Users' Conf. Proc., Vol. I, Paper No. 8, March, 1991.

Parthasarathy, Alwar; Elzeki, Mohamed; Abramovici, Vivianne. *PSDTOOL-A DMAP Enhancement to Harmonic/Random Response Analysis in MSC/NASTRAN*, The MSC 1993 World Users' Conf. Proc., Paper No. 36, May, 1993.

Patel, Jayant S.; Seltzer, S. M. *Complex Eigenvalue Solution to a Spinning Skylab Problem*, NASTRAN: Users' Exper., pp. 439-450, September, 1971, (NASA TM X-2378).

Patel, Jayant S.; Seltzer, S. M. *Complex Eigenvalue Analysis of Rotating Structures*, NASTRAN: Users' Exper., pp. 197-234, September, 1972, (NASA TM X-2637).

Patel, Kirit V. *Stress Analysis of Hybrid Pins in a Warped Printed Wiring Board Using MSC/NASTRAN*, MSC 1995 World Users' Conf. Proc., Paper No. 20, May, 1995.

Paxson, Ernest B., Jr. *Simulation of Small Structures- Optics-Controls Systems with MSC/NASTRAN*, The 1989 MSC World Users Conf. Proc., Vol. II, Paper No. 39, March, 1989.

Pinnament, Murthy. *Mode Acceleration Data Recovery in MSC/NASTRAN Dynamic Analysis with Generalized Dynamic Reduction*, MSC/NASTRAN Users' Conf. Proc., Paper No. 24, March, 1985.

Raney, John P.; Kaszubowski, M.; Ayers, J. Kirk. *Analysis of Space Station Dynamics Using MSC/NASTRAN*, The MSC 1987 World Users Conf. Proc., Vol. I, Paper No. 11, March, 1987.

Reyer, H. *A Crash-Down Calculated with NASTRAN*, Proc. of the MSC/NASTRAN Eur. Users' Conf., April, 1982.

Rose, Ted L. *Using Superelements to Identify the Dynamic Properties of a Structure*, The MSC 1988 World Users Conf. Proc., Vol. I, Paper No. 41, March, 1988.

Rose, Ted L. *Creation of and Use of 'Craig-Bampton' Models Using MSC/NASTRAN*, The MSC 1990 World Users Conf. Proc., Vol. II, Paper No. 51, March, 1990.

Rose, Ted. *Using Residual Vectors in MSC/NASTRAN Dynamic Analysis to Improve Accuracy*, The MSC 1991 World Users' Conf. Proc., Vol. I, Paper No. 12, March, 1991.

Rose, Ted. *DMAP Alters to Apply Modal Damping and Obtain Dynamic Loading Output for Superelements*, The MSC 1993 World Users' Conf. Proc., Paper No. 24, May, 1993.

Rose, Ted.; McNamee, Martin. *A DMAP Alter to Allow Amplitude-Dependent Modal Damping in a Transient Solution*, MSC 1996 World Users' Conf. Proc., Vol. V, Paper No. 50, June, 1996.

Ross, Robert W. *Prediction and Elimination of Resonance in Structural Steel Frames*, The MSC 1988 World Users Conf. Proc., Vol. II, Paper No. 45, March, 1988.

Russo, A.; Mocchetti, R. *Dynamic Analysis of Loaded Structures in the Helicopter Field*, Proc. of the MSC/NASTRAN Eur. Users' Conf., May, 1984.

Salus, W. L.; Jones, R. E.; Ice, M. W. *Dynamic Analysis of a Long Span, Cable-Stayed Freeway Bridge Using NASTRAN*, NASTRAN: Users' Exper., pp. 143-168, September, 1973, (NASA TM X-2893).

Sauer, G.; Wolf, M. *Gyroscopic Effects in the Dynamic Response of Rotating Structures*, Proc. of the MSC/NASTRAN Eur. Users' Conf., Paper No. 11, May, 1986.

Schips, C. *Aero-Engine Turbine Dynamic Analysis*, Proc. of the 18th MSC Eur. Users' Conf., Paper No. 8, June, 1991.

Schmitz, Ronald P. *Structural Dynamic Analysis of Electronic Assemblies Using NASTRAN Restart/Format Change Capability*, NASTRAN: Users' Exper., pp. 363-392, September, 1971, (NASA TM X-2378).

Schweiger, W.; de Bruyne, F.; Dirschmid, W. *Fluid Structure Interaction of Car Fuel Tanks*, Proc. of the MSC/NASTRAN Eur. Users' Conf., May, 1984.

Shein, Shya-Ling. *Generation of the Space Station Freedom On-Orbit Dynamic Loads Analysis Model Using MSC/NASTRAN V66A Superelements*, The 2nd Annual MSC Taiwan Users Conf., Paper No. 7, October, 1990.

Shiraki, K.; Hashimoto, H.; Sato, N.; Nasu, S.; Kinno, M. *Japanese Experiment Module (JEM): On-Orbit Structural Dynamic Analysis*, 1993 MSC Japan's 11th User's Conf. Proc., Paper No. 10.

Shivaji, M.; Raju, V. S. N. *Dynamic Analysis of R. C. C. Chimneys*, MSC 1995 World Users' Conf. Proc., Paper No. 34, May, 1995.

Singh, Ashok K.; Nichols, Christian W. *Derivation of an Equivalent Beam Model From a Structural Finite Element Model*, The MSC 1988 World Users Conf. Proc., Vol. I, Paper No. 14, March, 1988.

Singh, Sudeep K.; Engelhardt, Charlie. *Dynamic Analysis of a Large Space Structure Using External and Internal Superelements*, The MSC 1991 World Users' Conf. Proc., Vol. I, Paper No. 27, March, 1991.

Skattum, Knut S. *Modeling Techniques of Thin-Walled Beams with Open Cross Sections*, NASTRAN: Users' Exper., pp. 179-196, September, 1972, (NASA TM X-2637).

Smith, Michael R.; Rangacharyulu, M.; Wang, Bo P.; Chang, Y. K. *Application of Optimization Techniques to Helicopter Structural Dynamics*, AIAA/ASME/ASCE/AHS/ASC 32nd Structures, Structural Dynamics, and Materials Conf., Part 1, Paper No. 91-0924, pp. 227-237, April, 1991.

Stockwell, Alan E.; Perez, Sharon E.; Pappa, Richard S. *Integrated Modeling and Analysis of a Space-Truss Test Article*, The MSC 1990 World Users Conf. Proc., Vol. I, Paper No. 16, March, 1990.

Struschka, M.; Goldstein, H. *Approximation of Frequency Dependant Nonlinearities in Linear FE-Models*, Proc. of the 15th MSC/NASTRAN Eur. Users' Conf., October, 1988.

Subrahmanyam, K. B.; Kaza, K. R. V.; Brown, G. V.; Lawrence, C. *Nonlinear Bending-Torsional Vibration and Stability of Rotating, Pre-Twisted, Preconed Blades Including Coriolis Effects*, National Aeronautics and Space Administration, January, 1986, (NASA TM-87207).

Tecco, T. C. *Analyzing Frequency Dependent Stiffness and Damping with MSC/NASTRAN*, MSC/NASTRAN Users' Conf. Proc., Paper No. 25, March, 1985.

Thornton, E. A. *Application of NASTRAN to a Space Shuttle Dynamics Model*, NASTRAN: Users' Exper., pp. 421-438, September, 1971, (NASA TM X-2378).

Ting, Tienko. *Test/Analysis Correlation for Multiple Configurations*, The MSC 1993 World Users' Conf. Proc., Paper No. 74, May, 1993.

Tinti, Francesco Carlo; Scaffidi, Costantino. *Structural Dynamics and Acoustic Design of Engine Component in View of Exterior Noise Reduction Using Numerical Techniques*, MSC 1995 European Users' Conf. Proc., Italian Session, September, 1995.

Turner, Patrick Ryan. *Integrating Finite Element Analysis with Quasi-Static Loadings from a Large Displacement Dynamic Analysis*, The 1989 MSC World Users Conf. Proc., Vol. II, Paper No. 37, March, 1989.

Tzong, George T. J.; Sikes, Gregory D.; Dodd, Alan J. *Large Order Modal Analysis Module in the Aeroelastic Design Optimization Program (ADOP)*, The MSC 1991 World Users' Conf. Proc., Vol. II, Paper No. 36, March, 1991.

Unger, B.; Eichlseder, Wilfried; Schuch, F. *Predicting the Lifetime of Dynamically Stressed Components*, Proc. of the 20th MSC European Users' Conf., Paper No. 36, September, 1993.

Vance, Judy; Bernard, James E. *Approximating Eigenvectors and Eigenvalues Across a Wide Range of Design*, The MSC 1992 World Users' Conf. Proc., Vol. II, Paper No. 46, May, 1992.

Visintainer, Randal H.; Aslani, Farhang. *Shake Test Simulation Using MSC/NASTRAN*, MSC 1994 World Users' Conf. Proc., Paper No. 32, June, 1994.

Vitiello, P.; Quaranta, V. *SEA Investigation Via a FEM Based Substructuring Technique*, MSC 1995 European Users' Conf. Proc., Italian Session, September, 1995.

Walton, William B.; Blakely, Ken. *Modeling of Nonlinear Elastic Structures Using MSC/NASTRAN*, MSC/NASTRAN Users' Conf. Proc., Paper No. 11, March, 1983.

Wamsler, M.; Blanck, N.; Kern, G. *On the Enforced Relative Motion Inside a Structure*, Proc. of the 20th MSC European Users' Conf., September, 1993.

Wang, B. P.; Caldwell, S. P.; Smith, C. M. *Improved Eigensolution Reanalysis Procedures in Structural Dynamics*, The MSC 1990 World Users Conf. Proc., Vol. II, Paper No. 46, March, 1990.

Wang, B. P.; Chang, Y. K.; Lawrence, K. L.; Chen, T. Y. *Optimum Design of Structures with Multiple Configurations with Frequency and Displacement Constraints*, 31st AIAA/ ASME/ASCE/AHS/ASC Structures, Structural Dynamics and Materials Conf., Part 1, pp. 378-384, April, 1990.

Wang, B. P.; Caldwell, S. P. *Reducing Truncation Error in Structural Dynamic Modification*, The MSC 1991 World Users' Conf. Proc., Vol. I, Paper No. 11, March, 1991.

Wang, B. P.; Caldwell, S. P. *Improved Approximate Method for Computting Eigenvector Derivatives*, Finite Elements in Analysis and Design v 14 n 4 Nov 1993.

Watanabe, Masaaki. *Computation of Virtual Mass to Rigid Body Structure by MSC/NASTRAN*, The First MSC/NASTRAN User's Conf. in Japan, October, 1983, in Japanese.

Wijker, J. J. *Differential Stiffness in Conjunction with Dynamics*, NASTRAN User's Conf., June, 1981.

Wijker, J. J. *Acoustic Effects on the Dynamic Behaviour of Lightweight Structures*, Proc. of the MSC/NASTRAN Eur. Users' Conf., Paper No. 3, April, 1985.

Wilhelmy, Dr. Viktor. Dynamic Analysis with Gaps, The 1989 MSC World Users Conf. Proc., Vol. II, Paper No. 40, March, 1989.

Woytowitz, P. J.; Jiang, K. C.; Bhat, K. P. *Dynamic Analysis of Optical Beam Pointing*, The MSC 1988 World Users Conf. Proc., Vol. I, Paper No. 11, March, 1988.

Yang, Jackson C. S.; Frederick, Diana L. *Application of NASTRAN in Nonlinear Analysis of a Cartridge Case Neck Separation Malfunction*, NASTRAN: Users' Exper., pp. 389-396, September, 1975, (NASA TM X-3278).

Young, K. J.; Mitchell, L. D. *On the Performance of Various Kinds of Rod and Beam Mass Matrices on a Plane-Frame Structure*, Proc. of IMAC-IX, Vol. I, pp. 1057-1065, April, 1991.

DYNAMICS – ANALYSIS / TEST CORRELATION

Allen, James J.; Martinez, David R. *Techniques for Implementing Structural Model Identification Using Test Data*, Sandia National Laboratories, June, 1990, (SAND90-1185).

Anker, J. C. *Checks that Pay*, Proc. of the MSC/NASTRAN Eur. Users' Conf., May, 1984.

Blakely, Ken; Howard, G. E.; Walton, W. B.; Johnson, B. A.; Chitty, D. E. *Comparison of a Nonlinear Dynamic Model of a Piping System to Test Data*, 7th Int. Conf. on Struct. Mech. in Reactor Tech., August, 1983.

Blakely, Ken; Walton, W. B. *Selection of Measurement and Parameter Uncertainties for Finite Element Model Revision*, 2nd Int. Modal Analysis Conf., February, 1984.

Blakely, Ken. *Updating MSC/NASTRAN Models to Match Test Data*, The MSC 1991 World Users' Conf. Proc., Vol. II, Paper No. 50, March, 1991.

Blakely, Ken. *Revising MSC/NASTRAN Models to Match Test Data*, Proc. of the 9th Int. Modal Analysis Conf., April, 1991.

Blakely, Ken. *Get the Model Right, then Run the Analysis*, Machine Design, October 24, 1991.

Blakely, Ken; Rose, Ted. *Cross-Orthogonality Calculations for Pre-Test Planning and Model Verification*, The MSC 1993 World Users' Conf. Proc., Paper No. 72, May, 1993.

Blakely, Ken; Bush, Richard. *Using MSC/NASTRAN to Match Dynamic Test Data*, Proc. of the Int. Conf. on Structural Dynamics Modelling, July, 1993.

Blakely, Ken; Rose, Ted. *Cross-Orthogonality Calculations for Pre-Test Planning and Model Verification*, Proc. of the 20th MSC European Users' Conf., September, 1993.

Blakely, Ken. *Matching Frequency Response Test Data with MSC/NASTRAN*, MSC 1994 World Users' Conf. Proc., Paper No. 17, June 1994.

Blakely, Ken. *Matching Frequency Response Test Data with MSC/NASTRAN*, Proc. of the 21st MSC European User's Conf., Italian Session, September, 1994.

Brillhart, Ralph; Hunt, David L.; Kammer, Daniel C.; Jensen, Brent M.; Mason, Donald R. *Modal Survey and Test-Analysis Correlation of the Space Shuttle SRM*, Proc. of the 6th Int. Modal Analysis Conf., pp. 863-870, February, 1988.

Brughmans, Marc; Leuridan, Jan; Blauwkamp, Kevin. *The Application of FEM-EMA Correlation and Validation Techniques on a Body-in-White*, The MSC 1993 World Users' Conf. Proc., Paper No. 6, May, 1993.

Brughmans, M.; Lembregts, PhD. F.; Furini, PhD. F.; Storrer, O. *Modal Test on the Pininfarina Concept Car Body "ETHOS 1"*, Actes de la 2ème Confêrence Française Utilisateurs des Logiciels MSC, Toulouse, France, September, 1995.

Brughmans, M.; Lembregts, F, Ph.D.; Furini, F., Ph.D. *Modal Test on the Pininfarina Concept Car Body "ETHOS 1"*, MSC 1995 World Users' Conf. Proc., Paper No. 5, May, 1995.

Budynas, R.; Kolhatkar, S. *Modal Analysis of a Robot Arm Using Finite Element Analysis and Modal Testing*, Proc. of the 8th Int. Modal Analysis Conf., Vol. I, pp. 67-70, January, 1990.

Budynas, R. G.; Krebs, D. *Modal Correlation of Test and Finite Element Results Using Cross Orthogonality with a Reduced Mass Matrix Obtained by Modal Reduction and NASTRAN's Generalized Dynamic Reduction Solution*, Proc. of the 9th Int. Modal Analysis Conf., Vol. I, pp. 549-554, April, 1991.

Butler, Thomas G. *Test vs. Analysis: A Discussion of Methods*, Fourteenth NASTRAN Users' Colloq., pp. 173-186, May, 1986, (NASA CP-2419).

Call, V.; Mason, D. *Space Shuttle Redesigned Solid Rocket Booster Structural Dynamic Predictions and Correlations of Liftoff*, AIAA/SAE/ASME/ASEE 26th Joint Propulsion Conf., Paper No. AIAA 90-2081, July, 1990.

Chung, Y. T. *Model Reduction and Model Correlation Using MSC/NASTRAN*, MSC 1995 World Users' Conf. Proc., Paper No. 8, May, 1995.

Coladonato, Robert J. *Development of Structural Dynamic Test Evninroments for Subsystems and Components*, Seventh NASTRAN Users' Colloq., pp. 85-110, October, 1978, (NASA CP-2062).

Coppolino, Robert N. *Integrated Dynamic Test/Analysis Processor Overview*, MSC/NASTRAN Users' Conf. Proc., Paper No. 5, March, 1986.

Cronkhite, J. D. Development, *Documentation and Correlation of a NASTRAN Vibration Model of the AH-1G Helicopter Airframe*, NASTRAN: Users' Exper., pp. 273-294, October, 1976, (NASA TM X-3428).

Dascotte, E.; Von Estorff, O.; Wandinger, J. *Validation and Updating of MSC/NASTRAN Finite Element Models Using Experimental Modal Data*, Proc. of the 20th MSC European Users' Conf., Paper No. 10, September, 1993.

de Boer, A.; Kooi, B. W. *A DMAP for Updating Dynamic Mathematical Models with Respect to Measured Data*, Proc. of the MSC/NASTRAN Eur. Users' Conf., May, 1986.

Deger, Yasar. *Modal Analysis of a Concrete Gravity Dam - Linking FE Analysis and Test Results*, Proc. of the 20th MSC European Users' Conf., September, 1993.

Deutschel, Brian W.; Katnik, Richard B.; Bijlani, Mohan; Cherukuri, Ravi. *Improving Vehicle Response to Engine and Road Excitation Using Interactive Graphics and Modal Reanalysis Methods*, SAE Trans., Paper No. 900817, September, 1991.

Dirschmid, W.; Nolte, F.; Dunne, L. W. *Mathematical Model Updating Using Experimentally Determined Real Eigenvectors*, Proc. of the 17th MSC Eur. Users' Conf., Paper No. 4, September, 1990.

Drago, Raymond J.; Margasahayam, Ravi. *Stress Analysis of Planet Gears with Integral Bearings; 3-D Finite Element Model Development and Test Validation*, The MSC 1987 World Users Conf. Proc., Vol. I, Paper No. 4, March, 1987.

Ferg, D.; Foote, L.; Korkosz, G.; Straub, F.; Toossi, M.; Weisenburger, R. *Plan, Execute, and Discuss Vibration Measurements, and Correlations to Evaluate a NASTRAN Finite Element Model of the AH-64 Helicopter Airframe*, National Aeronautics and Space Administration, January, 1990, (NASA CR-181973).

Graves, Roger W. Interfacing MSC/NASTRAN with SDRC-IDEAS to Perform Component Mode Synthesis Combining Test, Analytical, and F. E. Data, The MSC 1988 World Users Conf. Proc., Vol. II, Paper No. 58, March, 1988.

Hehta, Pravin K. *Correlation of a NASTRAN Analysis with Test Measurements for HEAO-2 Optics*, MSC/NASTRAN Users' Conf. Proc., Paper No. 17, March, 1984.

Herbert, Andrew A.; Currie, A. O.; Wilson, W. *Analysis of Automotive Axle Carrier Assembly and Comparison with Test Data*, The MSC 1987 World Users Conf. Proc., Vol. I, Paper No. 6, March, 1987.

Herting, D. N. *Parameter Estimation Using Frequency Response Tests*, MSC 1994 World Users' Conf. Proc., Paper No. 18, June, 1994.

Jiang, K. C. *Finite Element Model Updates Using Modal Test Data*, The 1989 MSC World Users Conf. Proc., Vol. II, Paper No. 48, March, 1989.

Kabe, Alvar M. *Mode Shape Identification and Orthogonalization*, AIAA/ASME/ASCE/AHS/ASC 29th Structures, Structural Dynamics and Materials Conf., Paper No. 88-2354, 1988.

Kammer, Daniel C.; Jensen, Brent M.; Mason, Donald R. *Test-Analysis Correlation of the Space Shuttle Solid Rocket Motor Center* Segment, J. of Spacecraft, Vol. 26, No. 4, pp. 266-273, March, 1988.

Kelley, William R.; Isley, L. D. *Using MSC/NASTRAN for the Correlation of Experimental Modal Models for Automotive Powertrain Structures*, The MSC 1993 World Users' Conf. Proc., Paper No. 8, May, 1993.

Kelley, William R.; Isley, L. Dean; Foster, Thomas J. *Dynamic Correlation Study Transfer Case Housings*, MSC 1996 World Users' Conf. Proc., Vol. II, Paper No. 15, June, 1996.

Kientzy, Donald; Richardson, Mark; Blakely, Ken. *Using Finite Element Data to Set Up Modal Tests, Sound and Vibration*, June, 1989.

Lammens, Stefan; Brughmans, Marc; Leuridan, Jan; Sas, Paul. *Application of a FRF Based Model Updating Technique for the Validation of Finite Element Models of Components of the Automotive Industry*, MSC 1995 World Users' Conf. Proc., Paper No. 7, May, 1995.

Lee, John M.; Parker, Grant R. *Application of Design Sensitivity Analysis to Improve Correlations Between Analytical and Test Modes*, The 1989 MSC World Users Conf. Proc., Vol. I, Paper No. 21, March, 1989.

Linari, M.; Mancino, E. *Application of the MSC/NASTRAN Program to the Study of a Simple Reinforced Concrete Structure in Nonlinear Material Field*, Proc. of the 15th MSC/NASTRAN Eur. Users' Conf., October, 1988.

Lowrey, Richard D. *Calculating Final Mesh Size Before Mesh Completion*, The MSC 1990 World Users Conf. Proc., Vol. II, Paper No. 44, March, 1990.

Marlow, Jill M.; Lindell, Michael C. *NASSTAR: An Instructional Link Between MSC/NASTRAN and STAR*, Proceedings of the 11th International Modal Analysis Conference, Florida, 1993.

Masse, Barnard; Pastorel, Henri. *Stress Calculation for the Sandia 34-Meter Wind Turbine Using the Local Circulation Method and Turbulent Wind*, The MSC 1991 World Users' Conf. Proc., Vol. II, Paper No. 53, March, 1991.

Mindle, Wayne L.; Torvik, Peter J. *A Comparison of NASTRAN (COSMIC) and Experimental Results for the Vibration of Thick Open Cylindrical Cantilevered Shells*, Fourteenth NASTRAN Users' Colloq., pp. 187-204, May, 1986, (NASA CP-2419).

Morton, Mark H. *Application of MSC/NASTRAN for Assurance of Flight Safety and Mission Effectiveness with Regard to Vibration upon Installation of the Stinger Missile on the AH-64A*, The MSC 1991 World Users' Conf. Proc., Vol. II, Paper No. 52, March, 1991.

Neads, M. A.; Eustace, K. I. *The Solution of Complex Structural Systems by NASTRAN within the Building Block Approach*, NASTRAN User's Conf., May, 1979.

Nowak, William. *Electro-Mechanical Response Simulation of Electrostatic Voltmeters Using MSC/NASTRAN*, The MSC 1993 World Users' Conf. Proc., Paper No. 65, May, 1993.

O'Callahan, Dr. John; Avitabile, Peter; Reimer, Robert. *An Application of New Techniques for Integrating Analytical and Experimental Structural Dynamic Models*, The 1989 MSC World Users Conf. Proc., Vol. II, Paper No. 47, March, 1989.

Ott, Walter; Kaiser, Hans-Jurgen; Meyer, Jurgen. *Finite Element Analysis of the Dynamic Behaviour of an Engine Block and Comparison with Experimental Modal Test Results*, The MSC 1990 World Users Conf. Proc., Vol. I, Paper No. 14, March, 1990.

Paolozzi, A. *Structural Dynamics Modification with MSC/NASTRAN*, Proc. of the 19th MSC European Users' Conf., Paper No. 14, September, 1992.

Park, H. B.; Suh, J. K.; Cho, H. G.; Jung, G. S. *A Study on Idle Vibration Analysis Technique Using Total Vehicle Model*, MSC 1995 World Users' Conf. Proc., Paper No. 6, May, 1995.

Parker, Grant R.; Rose, Ted L.; Brown, John J. *Kinetic Energy Calculation as an Aid to Instrumentation Location in Modal Testing*, The MSC 1990 World Users Conf. Proc., Vol. II, Paper No. 47, March, 1990.

Preve, A.; Meneguzzo, M.; Merlo, A.; Zimmer, H. *Simulation of Vehicles' Structural Noise: Numerical/Experimental Correlation in the Acoustic Simulation of the Internal Noise*, Proc. of the 21st MSC European Users' Conf., Italian Session, September, 1994.

Rabani, Hadi. *Static and Dynamic FEM/Test Correlation of an Automobile Body*, The 1989 MSC World Users Conf. Proc., Vol. II, Paper No. 49, March, 1989.

Rainer, I. G. *MSC/NASTRAN as a Key Tool to Satisfy Increasing Demand for Numerical Simulation Techniques*, Proc. of the 20th MSC European Users' Conf., September, 1993.

Ray, William F. *The Use of MSC/NASTRAN and Empirical Data to Verify a Design*, MSC/NASTRAN Users' Conf. Proc., Paper No. 11, March, 1984.

Scapinello, F.; Colombo, E. *An Approach for Detailed Analysis of Complex Structures Avoiding Complete Models*, Proc. of the MSC/NASTRAN Eur. Users' Conf., May, 1987.

Sok-chu, Park; Ishii, Tetsu; Honda, Shigeki; Nagamatsu, Akio. *Vibration Analysis and Optimum Design of Press Machines*, 1994 MSC Japan Users' Conf. Proc.

Stack, Charles P.; Cunningham, Timothy J. *Design and Analysis of Coriolis Mass Flowmeters Using MSC/NASTRAN*, The MSC 1993 World Users' Conf. Proc., Paper No. 54, May, 1993.

Su, Hong. *Structural Analysis of Ka-BAND Gimbaled Antennas for a Communications Satellite System*, MSC 1996 World Users' Conf. Proc., Vol. IV, Paper No. 33, June, 1996.

Tawekal, Ricky; Budiyanto, M. Agus. *Finite Element Model Correlation for Structures*, The MSC 1993 World Users' Conf. Proc., Paper No. 73, May, 1993.

Ting, T.; Ojalvo, I. U. *Dynamic Structural Correlation via Nonlinear Programming Techniques*, The MSC 1988 World Users Conf. Proc., Vol. II, Paper No. 57, March, 1988.

Ting, Tienko; Chen, Timothy L. C. *FE Model Refinement with Actual Forced Responses of Aerospace Structures*, The MSC 1991 World Users' Conf. Proc., Vol. II, Paper No. 51, March, 1991.

Ting, Tienko. *Test/Analysis Correlation for Multiple Configurations*, The MSC 1993 World Users' Conf. Proc., Paper No. 74, May, 1993.

Ujihara, B. H.; Dosoky, M. M.; Tong, E. T. *Improving a NASTRAN Dynamic Model with Test Data Using Linwood*, Tenth NASTRAN Users' Colloq., pp. 74-86, May, 1982, (NASA CP-2249).

Zeischka, H. *LMS/Link Correlating and Validating F.E.A. for Dynamic Structure Behaviour with Experimental Modal Analysis*, Proc. of the MSC/NASTRAN Eur. Users' Conf., May, 1987.

DYNAMICS – COMPONENT MODE SYNTHESIS

Barnett, Alan R.; Ibrahim, Omar M.; Sullivan, Timothy L.; Goodnight, Thomas W. T*ransient Analysis Mode Participation for Modal Survey Target Mode Selection Using MSC/NASTRAN DMAP*, MSC 1994 World Users' Conf. Proc., Paper No. 8, June, 1994.

Bedrossian, Herand; Rose, Ted. *DMAP Alters for Nonlinear Craig-Bampton Component Modal Synthesis*, The MSC 1993 World Users' Conf. Proc., Paper No. 25, May, 1993.

Brillhart, Ralph; Hunt, David L.; Kammer, Daniel C.; Jensen, Brent M.; Mason, Donald R. *Modal Survey and Test-Analysis Correlation of the Space Shuttle SRM*, Proc. of the 6th Int. Modal Analysis Conf., pp. 863-870, February, 1988.

Brown, J. J.; Lee, J. M.; Parker, G. R.; Zuhuruddin, K. *Procedure for Processing and Displaying Entire Physical Modes Based on Results Generated Through Component Mode Synthesis*, The MSC 1988 World Users Conf. Proc., Vol. I, Paper No. 15, March, 1988.

Call, V.; Mason, D. *Space Shuttle Redesigned Solid Rocket Booster Structural Dynamic Predictions and Correlations of Liftoff*, AIAA/SAE/ASME/ASEE 26th Joint Propulsion Conf., Paper No. AIAA 90-2081, July, 1990.

Cifuentes, A. O.; Herting, D. N. *Transient Response of a Beam to a Moving Mass Using a Finite Element Approach, Innovative Numerical Methods in Engineering*, Proc. of the Fourth Int. Symp. on Numerical Methods in Engineering, Springer-Verlag, pp. 533-539, March, 1986.

Cifuentes, A. O. *Non-Linear Dynamic Problems Using a Combined Finite Element-Finite Difference Technique*, Proc. of the 6th Conf. on the Mathematics of Finite Elements and Application, April/May, 1987.

Carney, Kelly S.; Abdallah, Ayma A.; Hucklebridge, Arthur A. *Implementation of the Block-Krylov Boundary Flexibility Method of Component Synthesis*, The MSC 1993 World Users' Conf. Proc., Paper No. 26, May, 1993.

Del Basso, Steve; Singh, Sudeep; Lindenmoyer, Alan J. *Component Mode Synthesis of Space Station Freedom Using MSC/NASTRAN Superelement Architecture*, The MSC 1990 World Users Conf. Proc., Vol. II, Paper No. 48, March, 1990.

Duncan, Alan E. *Application of Modal Modeling and Mount System Optimization to Light Duty Truck Ride Analysis*, 4th Int. Conf. on Veh. Struct. Mech., pp. 113-128, November, 1981, (SAE #811313).

Flanigan, Christopher C.; Abdallah, Ayman; Manella, Richard. *Implementation of the Benfield-Hruda Modal Synthesis Method in MSC/NASTRAN*, The MSC 1992 World Users' Conf. Proc., Vol. I, Paper No. 11, May, 1992.

Garnek, Michael. *Large Space Structure Analysis Using Substructure Modal Test Data, AIAA/ASME/ASCE/AHS 25th Structures*, Structural Dynamics, and Materials Conf., AIAA Paper 84-0942-CP, May, 1984.

Ghosh, Tarun. *MSC/NASTRAN Based Component Mode Synthesis Analysis Without the Use of DMAPS*, MSC 1996 World Users' Conf. Proc., Vol. II, Paper No. 18, June, 1996.

Gieseke, R. K. *Analysis of Nonlinear Structures via Mode Synthesis*, NASTRAN: Users' Exper., pp. 341-360, September, 1975, (NASA TM X-3278).

Graves, Roger W. *Interfacing MSC/NASTRAN with SDRC-IDEAS to Perform Component Mode Synthesis Combining Test, Analytical, and F. E. Data*, The MSC 1988 World Users Conf. Proc., Vol. II, Paper No. 58, March, 1988.

Halcomb, J. R. *Application of Component Modes to the Analysis of a Helicopter*, Proc. of the MSC/NASTRAN Users' Conf., March, 1979.

Hambric, Stephen A. *Power Flow and Mechanical Intensity Calculations in Structural Finite Element Analysis*, ASME J. of Vibration and Acoustics, Vol. 112, pp. 542-549, October, 1990.

Herting, David N.; Hoesly, R. L. *Development of an Automated Multi-Stage Modal Synthesis System for NASTRAN*, Sixth NASTRAN Users' Colloq., pp. 435-448, October, 1977, (NASA CP-2018).

Herting, David N. *Accuracy of Results with NASTRAN Modal Synthesis*, Seventh NASTRAN Users' Colloq., pp. 389-404, October, 1978, (NASA CP-2062).

Herting, D. N. *A General Purpose, Multi-Stage, Component Modal Synthesis Method*, Finite Elements in Analysis and Design, Vol. 1, No. 2, 1985.

Hill, R. G.; Merckx, K. R. *Seismic Response Evaluation of a Reactor Core Using Modal Synthesis*, Proc. of the 3rd Int. Modal Analysis Conf., Vol. 2, pp. 996-1000, June, 1985.

Hodgetts, P. A.; Maitimo, F. M.; Wijker, J. J. *Dynamic Analysis of the Polar Platform Solar Array Using a Multilevel Component Mode Synthesis Technique*, Proc. of the 19th MSC European Users' Conf., Paper No. 15, September, 1992.

Ichikawa, Tetsuji; Hagiwara, Ichiro. *Frequency Response Analysis of Large-Scale Damped Structures Using Component Mode Synthesis*, Nippon Kikai Gakkai Ronbunshu, C Hen/Transactions of the Japan Society of Mechanical Engineers, Part C v 60 n 569 Jan 1994.

Jasuja, S. C.; Borowski, V. J.; Anderson, D. H. *Finite Element Modeling Techniques for the Simulation of Automotive Powertrain Dynamics*, Proc. of the 5th Int. Modal Analysis Conf., Vol. II, pp. 1520-1530, April, 1987.

Jay, Andrew; Lewis, Bryan; Stakolich, Ed. *Effect of Time Dependent Flight Loads on Turbofan Engine Performance Deterioration*, ASME J. of Engineering for Power, Vol. 104, No. 3, July, 1982.

Kammer, Daniel C.; Jensen, Brent M.; Mason, Donald R. *Test- Analysis Correlation of the Space Shuttle Solid Rocket Motor Center Segment*, J. of Spacecraft, Vol. 26, No. 4, pp. 266-273, March, 1988.

Kasai, Manabu. *Generalized CMS Employing External Superelements*, The Fourth MSC/NASTRAN User's Conf. in Japan, October, 1986, in Japanese.

Kasai, Manabu. *Representation of DMAP by Generalized CMS in System Identification*, The Fourth MSC/NASTRAN User's Conf. in Japan, October, 1986, in Japanese.

Kasai, Manabu. *Approach to CMS Subjected to the Boundary Constrained at Single Point*, The Sixth MSC/NASTRAN User's Conf. in Japan, October, 1988, in Japanese.

Kim, Hyoung M.; Bartkowicz, Theodoore J.; Van Horn, David A. *Data Recovery and Model Reduction Methods for Large Structures*, The MSC 1993 World Users' Conf. Proc., Paper No. 23, May, 1993.

Kubota, Minoru. *Efficient Use of Component Mode Sysnthesis Using Image Superelements Applied to Dynamic Analysis of Crankshaft*, MSC/NASTRAN Users' Conf. Proc., Paper No. 22, March, 1986.

Lee, W. M. *Substructure Mode Synthesis with External Superelement*, The 2nd Annual MSC Taiwan User's Conf., Paper No. 16, October, 1990, in Chinese.

MacNeal, R. H. *A Hybrid Method of Component Mode Synthesis, Computers and Structures*, Vol. 1, No. 4, pp. 581-601, 1971.

Martinez, D. R.; Gregory, D. L. *A Comparison of Free Component Mode Synthesis Techniques Using MSC/NASTRAN*, MSC/NASTRAN Users' Conf. Proc., Paper No. 18, March, 1983.

Martinez, David R.; Gregory, Danny L. *A Comparison of Free Component Mode Synthesis Techniques Using MSC/NASTRAN*, Sandia National Laboratories, June, 1984, (SAND83-0025).

Murakawa, Osamu. *Hull Vibration Analysis by Modal Synthesis Method*, The First MSC/NASTRAN User's Conf. in Japan, October, 1983, in Japanese.

Murakawa, Osamu; Iwahashi, Yoshio; Sakato, Tsuneo. *Ship Vibration Analysis Using Modal Synthesis Technique*, MSC/NASTRAN Users' Conf. Proc., Paper No. 23, March, 1984.

Nasu, Syouichi. *Modal Synthesis of Experimental Vibration Characteristics Using MSC/NASTRAN Results as the Reference Model*, The Sixth MSC/NASTRAN User's Conf. in Japan, October, 1988, in Japanese.

Nefske, D. J.; Sung, S. H.; Duncan, A. E. *Applications of Finite Element Methods to Vehicle Interior Acoustic Design*, Proc. of the 1984 Noise and Vibration Conf., Paper No. 840743, 1984.

Ookuma, Masaaki; Nagamatsu, Akio. *Comparison of Component Mode Synthesis Method with MSC-NASTRAN*, Nippon Kikai Gakkai Ronbunshu, C Hen, Vol. 49, No. 446, pp. 1883-1889, October, 1983, in Japanese.

Ookuma, Masaaki; Nagamatsu, Akio. *Comparison of Component Mode Synthesis Method with MSC-NASTRAN*, Bulletin of the JSME, Vol. 27, No. 228, pp. 1294-1298, June, 1984.

Parekh, Jatin C.; Harris, Steve G. *The Application of the Ritz Procedure to Damping Prediction Using a Modal Strain Energy Approach*, Damping '89, Paper No. CCB, November, 1989.

Philippopoulos, V. G. *Dynamic Analysis of an Engine-Transmission Assembly-Superelement and Component Mode Synthesis*, Proc. of the Conf. on Finite Element Methods and Technology, Paper No. 3, March, 1981.

Reyer, H. *Modal Synthesis with External Superelements in MSC/NASTRAN*, Proc. of the MSC/NASTRAN Eur. Users' Conf., May, 1984.

Sabahi, Dara; Rose, Ted. *Special Applications of Global-Local Analysis*, The MSC 1990 World Users Conf. Proc., Vol. II, Paper No. 49, March, 1990.

Sabahi, Dara; Rose, Ted. *MSC/NASTRAN Superelement Analysis of the NASA/AMES Pressurized Wind Tunnel*, The MSC 1990 World Users Conf. Proc., Vol. II, Paper No. 50, March, 1990.

Shein, Shya-Ling; Marquette, Brian; Rose, Ted. *Superelement Technology Application and Development in Dynamic Analysis of Large Space Structures*, The MSC 1991 World Users' Conf. Proc., Vol. I, Paper No. 26, March, 1991.

Suzukiri, Yoshihiro. *Component Mode Synthesis Application of MSC/NASTRAN V66*, The 2nd Annual MSC Taiwan Users Conf., Paper No. 10, October, 1990.

Suzukiri, Yoshihiro. *Component Mode Synthesis Application of MSC/NASTRAN V66*, Proc. of the First MSC/NASTRAN Users' Conf. in Korea, Paper No. 17, October, 1990.

Tong, Edward T.; Chang, Craig C. J. *An Efficient Procedure for Data Recovery of a Craig-Bampton Component*, MSC 1994 World Users' Conf. Proc., Paper No. 26, June 1994.

Wamsler, M.; Komzsik, L.; Rose, T. *Combination of Quasi-Static and Dynamic System Mode Shapes*, Proc. of the 19th MSC European Users' Conf., Paper No. 13, September, 1992.

Wang, Bo Ping. *Synthesis of Structures with Multiple Frequency Constraints*, AIAA/ASME/ASCE/AHS 27th Structures, Structural Dynamics and Materials Conf., Part 1, pp. 394-397, May, 1986.

DYNAMICS – DAMPING

El Maddah, M.; Imbert, J. F. *A Comparison of Damping Synthesis Methods for Space Vehicle Dynamic Analysis*, NASTRAN User's Conf., May, 1979.

Everstine, Gordon C.; Marcus, Melvyn S. *Finite Element Prediction of Loss Factors for Structures with Frequency- Dependent Damping Treatments*, Thirteenth NASTRAN Users' Colloq., pp. 419-430, May, 1985, (NASA CP-2373).

Gibson, W. C.; Johnson, C. D. *Optimization Methods for Design of Viscoelastic Damping Treatments*, ASME Design Engineering Division Publication, Vol. 5, pp. 279-286, September, 1987.

Gibson, Warren C.; Austin, Eric. *Analysis and Design of Damped Structures Using MSC/NASTRAN*, The MSC 1992 World Users' Conf. Proc., Vol. I, Paper No. 25, May, 1992.

Johnson, Conor D.; Keinholz, David A. *Prediction of Damping in Structures with Viscoelastic Materials Using MSC/NASTRAN*, MSC/NASTRAN Users' Conf. Proc., Paper No. 17, March, 1983.

Kalinowski, A. J. Modeling *Structural Damping for Solids Having Distinct Shear and Dilational Loss Factors*, Seventh NASTRAN Users' Colloq., pp. 193-206, October, 1978, (NASA CP-2062).

Kalinowski, A. J. *Solution Sensitivity and Accuracy Study of NASTRAN for Large Dynamic Problems Involving Structural Damping*, Ninth NASTRAN Users' Colloq., pp. 49-62, October, 1980, (NASA CP-2151).

Kienholz, Dave K.; Johnson, Conor D.; Parekh, Jatin C. *Design Methods for Viscoelastically Damped Sandwich Plates*, AIAA/ASME/ASCE/AHS 24th Structures, Structural Dynamics and Materials Conf., Part 2, pp. 334-343, May, 1983.

Li, Tsung-hsiun; Bernard, James. *Optimization of Damped Structures in the Frequency Domain*, The MSC 1993 World Users' Conf. Proc., Paper No. 28, May, 1993.

Lu, Y. P.; Everstine, G. C. *More on Finite Element Modeling of Damped Composite Systems*, J. of Sound and Vibration, Vol. 69, No. 2, pp. 199-205, 1980.

Mace, M. *Damping of Beam Vibrations by Means of a Thin Constrained Viscoelastic Layer: Evaluation of a New Theory*, Journal of Sound and Vibration v 172 n 5 May 19 1994.

Merchant, D. H.; Gates, R. M.; Ice, M. W.; Van Derlinden, J. W. *The Effects of Localized Damping on Structural Response*, NASTRAN: Users' Exper., pp. 301-320, September, 1975, (NASA TM X-3278).

Parekh, Jatin C.; Harris, Steve G. *The Application of the Ritz Procedure to Damping Prediction Using a Modal Strain Energy Approach*, Damping '89, Paper No. CCB, November, 1989.

Rose, Ted. *DMAP Alters to Apply Modal Damping and Obtain Dynamic Loading Output for Superelements*, The MSC 1993 World Users' Conf. Proc., Paper No. 24, May, 1993.

Rose, Ted.; McNamee, Martin. *A DMAP Alter to Allow Amplitude-Dependent Modal Damping in a Transient Solution*, MSC 1996 World Users' Conf. Proc., Vol. V, Paper No. 50, June, 1996.

Shieh, Rong C. *A Superefficient, MSC/NASTRAN-Interfaced Computer Code System for Dynamic Response Analysis of Nonproportionally Damped Elastic Systems*, The MSC 1993 World Users' Conf. Proc., Paper No. 14, May, 1993.

Tonin, Renzo. *Vibration Isolation of Impacts in High-Rise Structures*, The Second Australasian MSC Users Conf., Paper No. 11, November, 1988.

DYNAMICS – FREQUENCY RESPONSE

Balasubramanian, B.; Wamsler, M. *Identification of Contributing Modes in MSC/NASTRAN Modal Frequency Response Analyses*, Proc. of the MSC/NASTRAN Eur. Users' Conf., May, 1987.

Barnett, Alan R.; Widrick, Timothy W.; Ludwiczak, Damian R. *Combining Acceleration and Displacement Dependent Modal Frequency Responses Using An MSC/NASTRAN DMAP Alter*, MSC 1996 World Users' Conf. Proc., Vol. II, Paper No. 17, June, 1996.

Bellinger, Dean. *Dynamic Analysis by the Fourier Transform Method with MSC/NASTRAN*, MSC 1995 World Users' Conf. Proc., Paper No. 10, May, 1995.

Bianchini, Emanuele; Marulo, Francesco; Sorrentino, Assunta. *MSC/NASTRAN Solution of Structural Dynamic Problems Using Anelastic Displacement Fields*, Proceedings of the 36th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference and AIAA/ASME Adpative Structures Forum, Part 5 (of 5), New Orleans, 1995.

Blakely, Ken. *Matching Frequency Response Test Data with MSC/NASTRAN*, MSC 1994 World Users' Conf. Proc., Paper No. 17, June 1994.

Carlson, David L.; Shipley, S. A.; Yantis, T. F. *Procedure for FRF Model Tuning in MSC/NASTRAN*, The MSC 1993 World Users' Conf. Proc., Paper No. 71, May, 1993.

Charron, Francois; Donato, Louis; Fontaine, Mark. *Exact Calculation of Minimum Margin of Safety for Frequency Response Analysis Stress Results Using Yielding or Failure Theories*, The MSC 1993 World Users' Conf. Proc., Paper No. 5, May, 1993.

de la Fuente, E.; San Millán, J. *Calculation within MSC/NASTRAN of the Forces Transmitted by Multipoint Constraints (MPC) and the Forces Generated in Support Constraint*s, MSC 1996 World Users' Conf. Proc., Vol. II, Paper No. 20, June, 1996.

Herbert, S.; Janavicius, P. *MSC/NASTRAN Frequency Response Analysis of the Raven Army Communication Shelter*, The Third Australasian MSC Users Conf. Proc., Paper No. 14, November, 1989.

Herting, D. N. *Parameter Estimation Using Frequency Response Tests*, MSC 1994 World Users' Conf. Proc., Paper No. 18, June, 1994.

Kajiwara, Itsurou; Nagamatsu, Akio; Seto, Kazuto. *New Theory for Elimination of Resonance Peak and Optimum Design of Optical Servosystem*, 1994 MSC Japan Users' Conf. Proc., Paper No. 1.

Liew, K. M.; Jiang, L.; Lim, M. K.; Low, S. C. *Numerical Evaluation of Frequency Responses for Delaminated Honeycomb Structures*, Computers and Structures v 55 n 2 Apr 17 1995.

Parker, G. R.; Brown, J. J. *Evaluating Modal Contributors in a NASTRAN Frequency Response Analysis*, MSC/NASTRAN Users' Conf. Proc., Paper No. 14, March, 1983.

Rose, Ted. *Using Optimization in MSC/NASTRAN to Minimize Response to a Rotating Imbalance*, 1994 MSC Japan Users' Conf. Proc., Paper No. 28, December 1994.

Shieh, Rong C. *A Superefficient, MSC/NASTRAN-Interfaced Computer Code System for Dynamic Response Analysis of Nonproportionally Damped Elastic Systems*, The MSC 1993 World Users' Conf. Proc., Paper No. 14, May, 1993.

Soni, Ravi, et al. *Development of a Methodology to Predict the Road Noise Performance Characteristics*, The MSC 1993 World Users' Conf. Proc., Paper No. 9, May, 1993.

Tsutsui, Keicchiro; Nogami, Ray. *Development of a Nonlinear Frequency Response Program for Simulating Vehicle Ride Comfort*, MSC 1995 World Users' Conf. Proc., Paper No. 37, May, 1995.

Visintainer, Randal H.; Aslani, Farhang. *Shake Test Simulation Using MSC/NASTRAN*, MSC 1994 World Users' Conf. Proc., Paper No. 32, June, 1994.

Wamsler, Manfred; Krusemann, Rolf. *Calculating and Interpreting Contact Forces Between Brake Disc and Linings in Frequency Response Analysis*, Proc. of the 18th MSC Eur. Users' Conf., Paper No. 7, June, 1991.

Yen, K. Z. Y.; Hsueh, W. C.; Hsui, T. C. *Chatter Suppression of a CNC Lathe in Inside-Diameter Cutting*, The Sixth Annual MSC Taiwan Users' Conf. Proc., Paper No. 4, November, 1994.

DYNAMICS – MODES, FREQUENCIES, AND VIBRATIONS

(Korean). *Structural Analysis of Solar Array Substate*, MSC 1994 Korea Users' Conf. Proc., December, 1994, in Korean.

(Korean). *Optimal Design of Chip Mounter Considering Dynamic Characteristics*, MSC 1994 Korea Users' Conf. Proc., December, 1994, in Korean.

(Korean). *A Study on Vibration Characteristics of a Steering Wheel According to Geometric Variations*, MSC 1994 Korea Users' Conf. Proc., December, 1994, in Korean.

(Korean). *A Study on Idle Vibration Analysis Technique Using Total Vehicle Model*, MSC 1994 Korea Users' Conf. Proc., December, 1994, in Korean.

(Korean). *Vibration Analysis for Outercase in Drum Washer and Floor*, MSC 1994 Korea Users' Conf. Proc., December, 1994, in Korean.

Ahmad, M. Fouad; Guile, Carl W. *Analysis of Coupled Natural Frequencies of Thin-Walled Beams with Open Cross Sections Using MSC/NASTRAN*, The MSC 1990 World Users Conf. Proc., Vol. I, Paper No. 15, March, 1990.

Allen, James J.; Martinez, David R. *Techniques for Implementing Structural Model Identification Using Test Data*, Sandia National Laboratories, June, 1990, (SAND90-1185).

Arakawa, H.; Murakami, T.; Ito, H. *Vibration Analysis of the Turbine Generator Stator Frame*, The MSC 1988 World Users Conf. Proc., Vol. II, Paper No. 43, March, 1988.

Armand, Sasan; Lin, Paul. *Influence of Mass Moment of Inertia on Normal Modes of Preloaded Solar Array Mast*, The MSC 1992 World Users' Conf. Proc., Vol. I, Paper No. 12, May, 1992.

Arora, Tejbir; Birmingham, Lily. *Application of MSC/NASTRAN Superelement Dynamic Reduction Techniques for the Vertical Launching System*, The MSC 1988 World Users Conf. Proc., Vol. I, Paper No. 39, March, 1988.

Barnes, R. A.; Schmid, R.; Adrick, H. C. *Rotor Dynamic Analysis with MSC/NASTRAN via the Important Modes Method*, The 1989 MSC World Users Conf. Proc., Vol. I, Paper 13, March, 1989.

Barnett, Alan R.; Abdallah, Ayma A.; Ibrahim, Omar M.; Manella, Richard T. *Solving Modal Equations of Motion with Initial Conditions Using MSC/NASTRAN DMAP Part 1: Implementing Exact Mode Superposition*, The MSC 1993 World Users' Conf. Proc., Paper No. 12, May, 1993.

Barnett, Alan R.; Abdallah, Ayma A.; Ibrahim, Omar M.; Sullivan, Timothy L. *Solving Modal Equations of Motion with Initial Conditions Using MSC/NASTRAN DMAP Part 2: Coupled vs. Uncoupled Integration*, The MSC 1993 World Users' Conf. Proc., Paper No. 13, May, 1993.

Bella, David F.; Steinhard, E. *Critical Frequency Determination of a Flexible Rotating Structure Attached to a Flexible Support, Proc. of the 18th MSC Eur. Users' Conf.*, Paper No. 28, June, 1991.

Bella, David; Hartmueller, Hans; Muehlenfeld, Karsten; Tokar, Gabriel. *Identification of Critical Speeds of Rotors Attached to Flexible Supports*, The MSC 1993 World Users' Conf. Proc., Paper No. 34, May, 1993.

Blakely, Ken; Rose, Ted. *Cross-Orthogonality Calculations for Pre-Test Planning and Model Verification*, The MSC 1993 World Users' Conf. Proc., Paper No. 72, May, 1993.

Blakely, Ken; Rose, Ted. *Cross-Orthogonality Calculations for Pre-Test Planning and Model Verification*, Proc. of the 20th MSC European Users' Conf., September, 1993.

Brughmans, M.; Lembregts, PhD. F.; Furini, PhD. F.; Storrer, O. *Modal Test on the Pininfarina Concept Car Body "ETHOS 1"*, Actes de la 2ème Confèrence Française Utilisateurs des Logiciels MSC, Toulouse, France, September, 1995.

Brughmans, M.; Lembregts, F, Ph.D.; Furini, F., Ph.D. *Modal Test on the Pininfarina Concept Car Body "ETHOS 1"*, MSC 1995 World Users' Conf. Proc., Paper No. 5, May, 1995.

Buchanan, Guy. *Superelement Data Recovery via the Modal Acceleration Method*, The MSC 1988 World Users Conf. Proc., Vol. I, Paper No. 40, March, 1988.

Budynas, R.; Kolhatkar, S. *Modal Analysis of a Robot Arm Using Finite Element Analysis and Modal Testing*, Proc. of the 8th Int. Modal Analysis Conf., Vol. I, pp. 67-70, January, 1990.

Budynas, R. G.; Krebs, D. *Modal Correlation of Test and Finite Element Results Using Cross Orthogonality with a Reduced Mass Matrix Obtained by Modal Reduction and NASTRAN's Generalized Dynamic Reduction Solution*, Proc. of the 9th Int. Modal Analysis Conf., Vol. I, pp. 549-554, April, 1991.

Butler, Thomas G.; Muskivitch, John C. *Application of Flanigan's Mode Acceleration in MSC/NASTRAN Version 66*, The 1989 MSC World Users Conf. Proc., Vol. I, Paper No. 25, March, 1989.

Caldwell, Steve; Wang, B. P. *Application of Approximate Techniques in the Estimation of Eigenvalue Quality*, The MSC 1993 World Users' Conf. Proc., Paper No. 11, May, 1993.

Campanile, P.; Pisino, E.; Testi, R.; Manzilli, G.; Minen, D. *Flexible Structures in Adams Using Modal Data from NASTRAN*, Proc. of the 21st MSC European Users' Conf., Italian Session, September, 1994.

Carlson, Mark. *Applications of Finite Element Analysis for an Improved Musical Instrument Design*, MSC 1996 World Users' Conf. Proc., Vol. I, Paper No. 8, June, 1996.

Carneiro, S. H. S.; Duarte, J. A. A.; Mendonca, C. B. *Theoretical and Experimental Modal Analysis of the VLS (Satellite Launcher Vehicle) Bent*, Proc. of the 2nd MSC/NASTRAN Users' Conf. in Brazil, Paper No. 3, March, 1993, in Portuguese.

Case, William R. Jr. NASTRAN DMAP Alter for Determining a Local Stiffness Modification to Obtain a Specified Eigenvalue, NASTRAN: Users' Exper., pp. 269-284, September, 1973, (NASA TM X-2893).

Case, William R. *A NASTRAN DMAP Procedure for Calculation of Base Excitation Modal Participation Factors*, Eleventh NASTRAN Users' Colloq., pp. 113-140, May, 1983.

Cattani, E.; Micelli, D.; Sereni, L.; Cocordano, S. *Cylinder Block Eigenfrequencies and Eigenvectors Prediction with a Linear Brick and Wedge Finite Element Model*, Proc. of the 19th MSC European Users' Conf., Paper No. 16, September, 1992.

Chang, Cuann-yeu; Chang, Yuan-bing. *Using MSC/NASTRAN to Obtain Modal Parameters*, The MSC 1988 World Users Conf. Proc., Vol. I, Paper No. 21, March, 1988.

Chargin, M. L.; Dunne, L. W.; Herting, D. N. *Nonlinear Dynamics of Brake Squeal*, MSC 1996 World Users' Conf. Proc., Vol. V, Paper No. 47, June, 1996.

Chen, J. T.; Chyuan, S. W.; You, D. W.; Wong, H. T. *A New Method for Determining the Modal Participation Factor in Support Motion Problems Using MSC/NASTRAN*, The Seventh Annual MSC/NASTRAN Users' Conf. Proc., Taiwan, 1995.

Clary, Robert R. *Practical Analysis of Plate Vibrations Using NASTRAN*, NASTRAN: Users' Exper., pp. 325-342, September, 1971, (NASA TM X-2378).

Cohen, Allan R.; Laurenson, Robert M. *Application of a Substructure Technique for STS/Payload Coupled Modal Analysis*, Proc. of the MSC/NASTRAN Users' Conf., March, 1979.

Concilio, A.; Del Gatto, S.; Lecce, L.; Miccoli, G. *Simple and Cheap Noise and Vibration Active Control System Using Collocated Piezoelectric Devices on a Panel*, Proceedings of the 11th International Modal Analysis Conference, Florida, 1993.

Courtney, Roy Leon. *NASTRAN Modeling Studies in the Normal-Mode Method and Normal-Mode Synthesis*, NASTRAN: Users' Exper., pp. 181-200, September, 1971, (NASA TM X-2378).

Cronkhite, James D.; Smith, Michael R. *Experiences in NASTRAN Airframe Vibration Prediction at Bell Helicopter Textron*, American Helicopter Soc. Dynamics Specialists Mtg., Section 6, Vibrations Session I, Paper No. 1, November, 1989.

Cross, C.; Rao, A. *Comparison of Modal Performance of Alternate Compressor Bracket Design*, Seventh Australasian Users Conf. Proc., Sydney, October, 1993.

Deger, Yasar. *Modal Analysis of a Concrete Gravity Dam - Linking FE Analysis and Test Results*, Proc. of the 20th MSC European Users' Conf., September, 1993.

Deutschel, Brian William. *A Systematic Approach Using Finite Elements for Improving Vehicle Ride*, CAD/CAM Robotics and Factories of the Future Integration of Design, Analysis, and Manufacturing (Proc.), Springer-Verlag Berlin, Heidelberg, Vol. I, pp. 150-154, 1989.

Deutschel, Brian W.; Katnik, Richard B.; Bijlani, Mohan; Cherukuri, Ravi. *Improving Vehicle Response to Engine and Road Excitation Using Interactive Graphics and Modal Reanalysis Methods*, SAE Trans., Paper No. 900817, September, 1991.

Egashira, Yuji. *Large Scale Vibration Analysis of Car Body Using Superelement Method*, The Second MSC/NASTRAN User's Conf. in Japan, October, 1984, in Japanese.

El-Bayoumy, Lotfi. *Identification and Correction of Damaging Resonances in Gear Drives*, Gear Technology, Vol. 1, No. 2, pp. 14-19, August/September, 1984.

Ferg, D.; Foote, L.; Korkosz, G.; Straub, F.; Toossi, M.; Weisenburger, R. *Plan, Execute, and Discuss Vibration Measurements, and Correlations to Evaluate a NASTRAN Finite Element Model of the AH-64 Helicopter Airframe*, National Aeronautics and Space Administration, January, 1990, (NASA CR-181973).

Flanigan, Chris. *Methods for Calculating and Using Modal Initial Conditions in MSC/NASTRAN*, Proc. of the Conf. on Finite Element Methods and Technology, March, 1980.

Flanigan, C. *An Alternate Method for Mode Acceleration Data Recovery in MSC/NASTRAN*, Proc. of the Conf. on Finite Element Methods and Technology, Paper No. 7, March, 1981.

Flanigan, Christopher C. *Efficient and Accurate Procedures for Calculating Data Recovery Matrices for Superelement Models*, The 1989 MSC World Users Conf. Proc., Vol. II, Paper No. 44, March, 1989.

Flanigan, Christopher C. *Implementation of the IRS Dynamic Reduction Method in MSC/NASTRAN*, The MSC 1990 World Users Conf. Proc., Vol. I, Paper No. 13, March, 1990.

Friberg, Olof; Karlsson, Rune; Akesson, Bengt. *Linking of Modal and Finite Elements in Structural Vibration Analysis*, Proc. of the 2nd Int. Modal Analysis Conf. and Exhibit, Vol. 1, pp. 330-339, February, 1984.

Gallaher, Bruce. *Determination of Structural Dynamic Response Sensitivity to Modal Truncation*, MSC/NASTRAN Users' Conf. Proc., Paper No. 10, March, 1986.

Ghosh, Tarun; Nall, Marsha; Muniz, Ben; Cheng, Joseph. *Space Station Solar Array Pointing System Control/Structure Interaction Study Using CO-ST-IN for Modal Reduction*, The MSC 1993 World Users' Conf. Proc., Paper No. 68, May, 1993.

Gieseke, R. K. *Modal Analysis of the Mated Space Shuttle Configuration*, NASTRAN: Users' Exper., pp. 221-236, September, 1971, (NASA TM X-2378).

Girard, A.; Boullet, A.; Dardel, R. *Dynamic Analysis of a Satellite Using the Normal Modes of the Appendages*, Proc. of the MSC/NASTRAN Eur. Users' Conf., April, 1985.

Grandle, Robert E.; Rucker, Carl E. *Modal Analysis of a Nine-Bay Skin-Stringer Panel*, NASTRAN: Users' Exper., pp. 343-362, September, 1971, (NASA TM X-2378).

Gupta, Viney K.; Zillmer, Scott D.; Allison, Robert E. *Solving Large-Scale Dynamic Systems Using Band Lanczos Method in Rockwell NASTRAN on Cray X-MP*, Fourteenth NASTRAN Users' Colloq., pp. 236-246, May, 1986, (NASA CP-2419).

Hardman, E. S. *Static and Normal Modes Analysis of an Aircraft Structure Using the NASTRAN External Superelement Method*, Proc. of the MSC/NASTRAN Eur. Users' Conf., May, 1986.

Harn, Wen-Ren; Hwang, Chi-Ching. *Evaluation of Direct Model Modification Methods via MSC/NASTRAN DMAP Procedures*, The MSC 1990 World Users Conf. Proc., Vol. II, Paper No. 43, March, 1990.

Hayashida, Mirihiro. *Application of Design Sensitivity Analysis to Reduction of Vibration of Ship's Deck Structure*, The Sixth MSC/NASTRAN User's Conf. in Japan, October, 1988, in Japanese.

Herting, D. N.; Joseph, J. A.; Kuusinen, L. R.; MacNeal, R. H. *Acoustic Analysis of Solid Rocket Motor Cavities by a Finite Element Method*, National Aeronautics and Space Administration, pp. 285-324, September, 1971, (NASA TM X-2378).

Herting, David N. *Accuracy of Results with NASTRAN Modal Synthesis*, Seventh NASTRAN Users' Colloq., pp. 389-404, October, 1978, (NASA CP-2062).

Hill, R. G. *The Use of MSC/NASTRAN to Determine the Impact Response of a Reactor Core Due to Seismic Loading*, The MSC 1987 World Users Conf. Proc., Vol. I, Paper No. 26, March, 1987.

Hirano, Tohru. *Visual Evaluation Method for the Vibration Analysis Utilizing a Super Intelligent Color Graphic Display*, The First MSC/NASTRAN User's Conf. in Japan, October, 1983, in Japanese.

Hsueh, W. c.; Hsui, T. C.; Yen, K. Z. Y. *Modal and Frequency Response Analyses of Vertical Machining Center Structures - VC65*, The Fifth Annual MSC Taiwan Users' Conf. Proc., November, 1993.

Huang, Jieh-Shan. *Detect the Variety of Structural System by the Vibration Test*, The 2nd Annual MSC Taiwan Users Conf., Paper No. 11, October, 1990, in Chinese.

Igarashi, Mitsuo. *Eigenvalue Analysis of Shaft Supported by Anti-Isotropic Bearing*, The Fifth MSC/NASTRAN User's Conf. in Japan, October, 1987, in Japanese.

Ito, Hiroyuki. *Application of Modal Analysis Technique for Cars*, The Fourth MSC/NASTRAN User's Conf. in Japan, October, 1986, in Japanese.

Jabbour, K. N. *Normal Mode Analysis of the Radio Astronomy Explorer (RAE) Booms and Spacecraft*, NASTRAN: Users' Exper., pp. 237-250, September, 1971, (NASA TM X-2378).

Jasuja, S. C.; Borowski, V. J.; Anderson, D. H. *Finite Element Modeling Techniques for the Simulation of Automotive Powertrain Dynamics*, Proc. of the 5th Int. Modal Analysis Conf., Vol. II, pp. 1520-1530, April, 1987.

Jiang, K. C.; Gahart, R. *Analysis and Modal Survey Test of Intelsat VIIA Deployed Solar Array*, MSC 1994 World Users' Conf. Proc., Paper No. 27, June 1994.

Jiang, L.; Liew, K.M.; Lim, M.K.; Low, S.C. *Vibratory Behaviour of Delaminated Honeycomb Structures: a 3-D Finite Element Modeling*, Computers and Structures v 55 n 5 Jun 3 1995.

Ju, Yeuan Jyh; Ting, Tienko. *Modelling and Analysis of an Accelerometer Using MSC/ARIES and MSC/NASTRAN*, MSC 1995 World Users' Conf. Proc., Paper No. 22, May, 1995.

Kabe, Alvar M. *Mode Shape Identification and Orthogonalization*, AIAA/ASME/ASCE/AHS/ASC 29th Structures, Structural Dynamics and Materials Conf., Paper No. 88-2354, 1988.

Kajiwara, Itsuro; Nagamatsu, Akio. *Optimum Design of Structure and Control Systems by Modal Analysis*, 1994 MSC Japan Users' Conf. Proc.

Kam, T. Y.; Yang, C. M.; Wu, J. H. *Determination of Natural Frequencies of Laminated Composite Space Structures Via The Experiemental and Finite Element Approaches*, The Sixth Annual MSC Taiwan Users' Conf. Proc., Paper No. 5, November, 1994.

Kang, J. M.; Kim, J. Y.; Lee, K. J.; Yum, D. J.; Seol, Y. S.; Rashed, S. Kawahara, A. *Simulation of 3-D Sloshing and Structural Response in Ship's Tanks Taking Account of Fluid-Structure Interaction*, 1994 MSC Japan Users' Conf. Proc., Paper No. 29, December 1994.

Kasai, Manabu. *Better Accuracy of Response Derived from Modal Analysis*, The Second MSC/NASTRAN User's Conf. in Japan, October, 1984, in Japanese.

Katnik, Richard B.; Yu, Che-Hsi; Wolf, Walt. *Interactive Modal Animation and Structural Modification*, Proc. of the 6th Int. Modal Analysis Conf., Vol. I, pp. 947-952, February, 1988.

Kelley, William R.; Isley, L. D. *Using MSC/NASTRAN for the Correlation of Experimental Modal Models for Automotive Powertrain Structures*, The MSC 1993 World Users' Conf. Proc., Paper No. 8, May, 1993.

Kientzy, Donald; Richardson, Mark; Blakely, Ken. *Using Finite Element Data to Set Up Modal Tests*, Sound and Vibration, June, 1989.

Knott, George; Ishin, Young; Chargin, M. *A Modal Analysis of the Violin*, The MSC 1988 World Users Conf. Proc., Vol. II, Paper No. 42, March, 1988.

Kodiyalam, Srinivas; Graichen, Catherine M.; Connell, Isobel J.; Finnigan, Peter M. *Design Optimization of Satellite Structures for Frequency, Strength, and Buckling Requirements*, Aerospace Sciences Meeting and Exhibit, AIAA, January, 1993.

Krishnamurthy, Ravi S. *Stress and Vibration Analysis of Radial Gas Turbine Components*, Sixteenth NASTRAN Users' Colloq., pp. 128-137, April, 1988, (NASA CP-2505).

Lapi, M.; Grangier, H. *Modal Effective Parameters: an Application to Shipboard Support Structures to Reduce Vibrations Transmission*, Proc. of the 17th MSC Eur. Users' Conf., Paper No. 5, September, 1990.

Larkin, Paul A.; Miller, Michael W. *MSC/NASTRAN Superelement Analysis*, MSC/NASTRAN Users' Conf. Proc., Paper No. 10, March, 1982.

Lawrie, Geoff. *The Determination of the Normal Modes of a Gliding Vehicle*, The Sixth Australasian MSC Users Conf. Proc., November, 1992.

Lee, Jyh-Chian. *Using Residual Vector in MSC/NASTRAN Modal Frequency Response to Improve Accuracy*, The 4th MSC Taiwan Users' Conf., Paper No. 8, November, 1992, in Chinese.

Lee, Sang H. *Effective Modal Mass for Characterization of Vibration Modes*, Proc. of the Second MSC/NASTRAN Users' Conf. in Korea, Paper No. 22, October, 1991.

Liepins, Atis A.; Conaway, John H. *Application of NASTRAN to Propeller-Induced Ship Vibration*, NASTRAN: Users' Exper., pp. 361-376, September, 1975, (NASA TM X-3278).

Lim, Tae W.; Kashangaki, Thomas A. L. *Structural Damage Detection of Space Truss Structures Using Best Achievable Eigenvectors*, AIAA Journal v32 n 5 May 1994.

Lipman, Robert R. *Computer Animation of Modal and Transient Vibrations*, Fifteenth NASTRAN Users' Colloq., pp. 111-117, August, 1987, (NASA CP-2481).

Liu, Dauh-Churn, Shieh, Niahn-Chung. *Vibration Suppression of High Precision Grinding Machine Using Finite Element Method (MSC/NASTRAN)*, The Sixth Annual MSC Taiwan Users' Conf. Proc., Paper No. 7, November, 1994.

Lu, Y. P.; Killian, J. W.; Everstine, G. C. *Vibrations of Three Layered Damped Sandwich Plate Composites*, J. of Sound and Vibration, Vol. 64, No. 1, pp. 63-71, 1979.

Lui, C. Y.; Mason, D. R. *Space Shuttle Redesigned Solid Rocket Motor Nozzle Natural Frequency Variations with Burn Time*, AIAA/SAE/ASME 27th Joint Propulsion Conf., Paper No. AIAA 91-2301, June, 1991.

Lundgren, Gert. *MSC's Solvers Predict the Best Attachment of the Sunbeam Tiger Fiberglass Front-End*, MSC 1996 World Users' Conf. Proc., Vol. II, Paper No. 16, June, 1996.

Marcus, Melvyn S. *A Finite-Element Method Applied to the Vibration of Submerged Plates*, J. of Ship Research, Vol. 22, No. 2, pp. 94-99, June, 1978.

Marcus, Melvyn S.; Everstine, Gordon C.; Hurwitz, Myles M. *Experiences with the QUAD4 Element for Shell Vibrations*, Sixteenth NASTRAN Users' Colloq., pp. 39-43, April, 1988, (NASA CP-2505).

Mase, M.; Saito, H. *Application of FEM for Vibrational Analysis of Ground Turbine Blades*, MSC/NASTRAN Users' Conf., March, 1978.

McMeekin, Michael; Kirchman, Paul. *An Advanced Post Processing Methodology for Viewing MSC/NASTRAN Generated Analyses Results*, MSC 1994 World Users' Conf. Proc., Paper No. 21, June, 1994.

Mei, Chuh; Rogers, James L. Jr. *NASTRAN Nonlinear Vibration Analyses of Beam and Frame Structures*, NASTRAN: Users' Exper., pp. 259-284, September, 1975, (NASA TM X-3278).

Mei, Chuh; Rogers, James L. Jr. *Application of the TRPLT1 Element to Large Amplitude Free Vibrations of Plates*, Sixth NASTRAN Users' Colloq., pp. 275-298, October, 1977, (NASA CP-2018).

Meyer, Karl A. *Normal Mode Analysis of the IUS/TDRS Payload in a Payload Canister/Transporter Environment*, Eighth NASTRAN Users' Colloq., pp. 113-130, October, 1979, (NASA CP-2131).

Michiue, Shinsuke. *On the Accuracy in Vibration Analysis for Cylindrical Shell-Comparison Between QUAD4/QUAD8*, The First MSC/NASTRAN User's Conf. in Japan, October, 1983, in Japanese.

Miller, R. D. *Theoretical Analysis of HVAC Duct Hanger Systems*, Fifteenth NASTRAN Users' Colloq., pp. 222-249, August, 1987, (NASA CP-2481).

Nack, Wayne V.; Joshi, Arun M. *Friction Induced Vibration*, MSC 1995 World Users' Conf. Proc., Paper No. 36, May, 1995.

Nagayasu, Katsuyosi. *Method for Prediction of Noise-Oriented Vibration on Pipe Lines to Refrigerators*, The Third MSC/NASTRAN User's Conf. in Japan, October, 1985, in Japanese.

Nagendra, Gopal K.; Herting, David N. *Design Sensitivity for Modal Analysis*, The 1989 MSC World Users Conf. Proc., Vol. I, Paper No. 22, March, 1989.

Nagy, Lajos I.; Cheng, James; Hu, Yu-Kan. *A New Method Development to Predict Brake Squeal Occurence*, MSC 1994 World Users' Conf. Proc., Paper No. 14, June, 1994.

Neads, M. A.; Eustace, K. I. *The Solution of Complex Structural Systems by NASTRAN within the Building Block Approach*, NASTRAN User's Conf., May, 1979.

Newman, Malcolm; Pipano, Aaron. *Fast Modal Extraction in NASTRAN via the FEER Computer Program*, NASTRAN: Users' Exper., pp. 485-506, September, 1973, (NASA TM X-2893).

Nishiwaki, Nobukiyo. *Coupled Vibration of Rotating Disc and Blades*, The Second MSC/NASTRAN User's Conf. in Japan, October, 1984, in Japanese.

Overbye, Vern D. *MSC/NASTRAN Dynamic Analysis: Modal or Direct?*, MSC/NASTRAN Users' Conf. Proc., Paper No. 6, March, 1986.

Pamidi, M. R.; Pamidi, P. R. *Modal Seismic Analysis of a Nuclear Power Plant Control Panel and Comparison with SAP IV*, NASTRAN: Users' Exper., pp. 515-530, October, 1976, (NASA TM X-3428).

Pamidi, P. R. *On the Append and Continue Features in NASTRAN*, Seventh NASTRAN Users' Colloq., pp. 405-418, October, 1978, (NASA CP-2062).

Paolozzi, A. *Interfacing MSC/NASTRAN with a Structural Modification Code*, Proc. of the 18th MSC Eur. Users' Conf., Paper No. 30, June, 1991.

Park, H. B.; Suh, J. K.; Cho, H. G.; Jung, G. S. *A Study on Idle Vibration Analysis Technique Using Total Vehicle Model*, MSC 1995 World Users' Conf. Proc., Paper No. 6, May, 1995.

Parker, G. R.; Brown, J. J. *Kinetic Energy DMAP for Mode Identification*, MSC/NASTRAN Users' Conf. Proc., Paper No. 8, March, 1982.

Parker, G. R.; Brown, J. J. *Evaluating Modal Contributors in a NASTRAN Frequency Response Analysis*, MSC/NASTRAN Users' Conf. Proc., Paper No. 14, March, 1983.

Pulgrano, Louis J.; Masters, Steven G. *Self-Excited Oscillation of a 165 Foot Water Tower*, MSC 1995 World Users' Conf. Proc., Paper No. 32, May, 1995.

Reyer, H. *Modal Synthesis with External Superelements in MSC/NASTRAN*, Proc. of the MSC/NASTRAN Eur. Users' Conf., May, 1984.

Rose, Ted. *A Method to Apply Initial Conditions in Modal Transient Solutions*, The MSC 1991 World Users' Conf. Proc., Vol. I, Paper No. 13, March, 1991.

Rose, Ted. *Some Suggestions for Evaluating Modal Solutions*, The MSC 1992 World Users' Conf. Proc., Vol. I, Paper No. 10, May, 1992.

Rose, Ted. *Using Optimization in MSC/NASTRAN to Minimize Response to a Rotating Imbalance*, MSC 1994 Korea Users' Conf. Proc., December, 1994.

Rose, Ted. *Using Dynamic Optimization to Minimize Driver Response to a Tire Out-of-Balance*, MSC 1994 Korea Users' Conf. Proc., December, 1994.

Rose, Ted. *Using Optimization in MSC/NASTRAN to Minimize Response to a Rotating Imbalance*, 1994 MSC Japan Users' Conf. Proc., Paper No. 28, December 1994.

Rose, Ted L. *Using Optimization in MSC/NASTRAN to Minimize Response to a Rotating Imbalance*, The Sixth Annual MSC Taiwan Users' Conf. Proc., Paper No. D, November, 1994.

Saito, Hiroshi; Watanabe, Masaaki. *Modal Analysis of Coupled Fluid-Structure Response*, MSC/NASTRAN Users' Conf. Proc. March, 1982.

Salvestro, Livio; Sirocco, Howden; Currie, Andrew. *Cyclic Symmetry Analysis of an Air Blower Fan*, Seventh Australasian Users Conf. Proc., Sydney, October, 1993.

Scanlon, Jack; Swan, Jim. *A Stand-Alone DMAP Program for Modal Cross-Correlation*, MSC 1995 World Users' Conf. Proc., Paper No. 40, May, 1995.

Schiavello, D. V.; Sinkiewicz, J. E. *DMAP for Determining Modal Participation*, MSC/NASTRAN Users' Conf. Proc., Paper No. 15, March, 1983.

Schwering, W. Shulze. *A DMAP for Identification of Modeshapes*, Proc. of the MSC/NASTRAN Eur. Users' Conf., June, 1983.

Shalev, D.; Unger, A. *Nonlinear Analysis Using a Modal-Based Reduction Technique*, Composite Structures v31 n 4 1995.

Shalev, Doron; Unger, A. *Nonlinear Analysis Using a Modal Based Reduction Technique*, The MSC 1993 World Users' Conf. Proc., Paper No. 51, May, 1993.

Shippen, J. M. *Normal Modes Analysis of Spin Stabilised Spacecraft Possessing Cable Booms*, Proc. of the 18th MSC Eur. Users' Conf., Paper No. 29, June, 1991.

Shirai, Yujiro; Arakawa, Haruhiko; Toda, Nobuo; Taneda, Yuji; Sakura, Kiyoshi. *Active Vibration Control for Aircraft Wing*, JSME International Journal, v 36 n 3 Spe 1993.

Shy, Tyson; Hsiu, T. C.; Yen, K. Z. Y. *Optimization of Structure Design of a Machining Center*, The Sixth Annual MSC Taiwan Users' Conf. Proc., Paper No. 6, November, 1994.

Somayajula, Gopichand; Stout, Joseph; Tucker, John. *Eigenvalue Reanalysis Using Subspace Iteration Techniques*, The 1989 MSC World Users Conf. Proc., Vol. I, Paper No. 26, March, 1989.

Stack, Charles P.; Cunningham, Timothy J. *Design and Analysis of Coriolis Mass Flowmeters Using MSC/NASTRAN*, The MSC 1993 World Users' Conf. Proc., Paper No. 54, May, 1993.

Starnes, James H. Jr. *Vibration Studies of a Flat Plate and a Built-Up Wing*, NASTRAN: Users' Exper., pp. 637-646, September, 1971, (NASA TM X-2378).

Su, Hong. *Structural Analysis of Ka-BAND Gimbaled Antennas for a Communications Satellite System*, MSC 1996 World Users' Conf. Proc., Vol. IV, Paper No. 33, June, 1996.

Subrahmanyam, K. B.; Kaza, K. R. V.; Brown, G. V.; Lawrence, C. *Nonlinear Vibration and Stability of Rotating, Pretwisted, Preconed Blades Including Coriolis Effects*, J. of Aircraft, Vol. 24, No. 5, pp. 342-352, May, 1987.

Sundaram, S. V.; Hohman, Richard L.; Richards, Timothy R. *Vibration Modes of a Tire Using MSC/NASTRAN*, MSC/NASTRAN Users' Conf. Proc., Paper No. 26, March, 1985.

Tamba, Richard; Mowbray, Graham; Rao, Ananda. *An Effective Method to Increase the Natural Frequencies of a Transmission Assembly*, The Sixth Australasian MSC Users Conf. Proc., November, 1992.

Tawekal, Ricky; Budiyanto, M. Agus. *Finite Element Model Correlation for Structures*, The MSC 1993 World Users' Conf. Proc., Paper No. 73, May, 1993.

Tawekal, Ricky L.; Miharjana, N. P. *Validation of 3650 DWT Semi Containe Ship Finite Element Model by Full Scale Measurements*, MSC 1994 World Users' Conf. Proc., Paper No. 19, June, 1994.

Thornton, Earl A. *A NASTRAN Correlation Study for Vibrations of a Cross-Stiffened Ship's Deck*, NASTRAN: Users' Exper., pp. 145-160, September, 1972, (NASA TM X-2637).

Ting, Tienko; Chen, Timothy L. C.; Twomey, William. *A Practical Solution to Mode Crossing Problem in Continuous Iterative Procedure*, The 1989 MSC World Users Conf. Proc., Vol. I, Paper No. 14, March, 1989.

Ting, Tienko; Chen, T.; Twomey, W. *Correlating Mode Shapes Based on Modal Assurance Criterion*, The MSC 1992 World Users' Conf. Proc., Vol. I, Paper No. 21, May, 1992.

Tokuda, Naoaki; Mitikami, Shinsuke; Sakata, Yoshiuki. *Accuracy of Vibration Analysis for Thin Cylindrical Shell by MSC/NASTRAN*, MSC/NASTRAN Users' Conf. Proc., Paper No. 28, March, 1984.

Vaillette, David. *Evaluation of the Modal Response of a Pressure Vessel Filled with a Fluid*, The MSC 1991 World Users' Conf. Proc., Vol. I, Paper No. 24, March, 1991.

Vance, Judy, M.; Bernard, James E. *Approximating Eigenvectors and Eigenvalues Across a Wide Range of Design*, Finite Elements in Analysis and Design v 14 n 4 Nov 1993.

Vandepitte, D.; Wijker, J. J.; Appel, S.; Spiele, H. *Normal Modes Analysis of Large Models, and Applications to Ariane 5 Engine Frame*, Proc. of the 18th MSC Eur. Users' Conf., Paper No. 6, June, 1991.

Wamsler, M.; Komzsik, L.; Rose, T. *Combination of Quasi-Static and Dynamic System Mode Shapes*, Proc. of the 19th MSC European Users' Conf., Paper No. 13, September, 1992.

Wang, B. P.; Cheu, T. C.; Chen, T. Y. *Optimal Design of Compressor Blades with Multiple Natural Frequency Constraints*, ASME Design Engineering Division Publication, Vol. 5, pp. 113-117, September, 1987.

Wang, B. P.; Lu, C. M.; Yang, R. J. *Topology Optimization Using MSC/NASTRAN*, MSC 1994 World Users' Conf. Proc., Paper No. 12, June, 1994.

Wang, Bo Ping. *Minimum Weight Design of Structures with Natural Frequency Constraints Using MSC/NASTRAN*, The MSC 1988 World Users Conf. Proc., Vol. II, Paper No. 60, March, 1988.

Welte, Y. *Vibration Analysis of an 8MW Diesel Engine*, Proc. of the MSC/NASTRAN Eur. Users' Conf., May, 1986.

West, Timothy S. *Approximate Dynamic Model Sensitivity Analysis For Large, Complex Space Structures*, MSC 1996 World Users' Conf. Proc., Vol. I, Paper No. 6, June, 1996.

Wijker, J. J. *Substructuring Technique Using the Modal Constraint Method*, Proc. of the MSC/NASTRAN Eur. Users' Conf., June, 1983.

Wijker, J. J. *MSC/NASTRAN Normal Mode Analysis on CRAY Computers*, Proc. of the MSC/NASTRAN Eur. Users' Conf., June, 1983.

Yang, Howard J. *Sorted Output in MSC/NASTRAN*, Proc. of the Conf. on Finite Element Methods and Technology, Paper No. 4, March, 1981.

Yen, K. Z. Y.; Hsueh, W. C.; Hsui, T. C. *Chatter Suppression of a CNC Lathe in Inside-Diameter Cutting*, The Sixth Annual MSC Taiwan Users' Conf. Proc., Paper No. 4, November, 1994.

Zhu, H.; Knight, D. *Finite Element Forced Response Analysis on the Mondeo Front End Accessory Drive System*, Proc. of the 20th MSC European Users' Conf., September, 1993.

DYNAMICS – RANDOM RESPONSE

Barnett, Alan R.; Widrick, Timothy W.; Ludwiczak, Damian R. *Combining Acceleration and Displacement Dependent Modal Frequency Responses Using An MSC/NASTRAN DMAP Alter*, MSC 1996 World Users' Conf. Proc., Vol. II, Paper No. 17, June, 1996.

Chiang, C. K.; Robinson, J. H.; Rizzi, S. A. Equivalent Linearization Solution Sequence for MSC/NASTRAN, Winter Annual Meeting of the American Society of Mechanical Engineers, pp. 133-138, November, 1992.

Ciuti, Gianluca. *Avionic Equipment Dynamic Analysis*, MSC 1995 European Users' Conf. Proc., Italian Session, September, 1995.

Coyette, J. P.; Lecomte, C.; von Estorff, O. *Evaluation of the Response of a Coupled Elastic Structure Subjected to Random Mechanical or Acoustical Excitations Using MSC/NASTRAN and SYSNOISE*, MSC European Users' Conf., Paper No. 21, September, 1996.

Crispino, Maurizio. *A 3-D Model for the Evaluation through Random Analysis of Vertical Dynamic Overloads in High Speed Railway Lines*, MSC 1995 European Users' Conf. Proc., Italian Session, September, 1995.

Galletly, Robert; Wagner, R. J.; Wang, G. J.; Zins, John. *Random Vibration and Acoustic Analysis Using ARI RANDOM*, a NASTRAN Post Processor, MSC/NASTRAN Users' Conf. Proc., Paper No. 26, March, 1984.

Hatheway, A. *Random Vibrations in Complex Electronic Structures*, MSC/NASTRAN Users' Conf. Proc., Paper No. 13, March, 1983.

Hatheway, Alson E. *Evaluation of Ceramic Substrates for Packaging of Leadless Chip Carriers*, MSC/NASTRAN Users' Conf. Proc., Paper No. 16, March, 1982.

Michels, Gregory J. *Vibroacoustics Random Response Analysis Methodology*, MSC 1995 World Users' Conf. Proc., Paper No. 9, May, 1995.

Palmieri, F. W. *Example Problems Illustrating the Effect of Multiple Cross Correlated Excitations on the Response of Linear Systems to Gaussian Random Excitations*, The MSC 1988 World Users Conf. Proc., Vol. I, Paper No. 18, March, 1988.

Palmieri, F. W. *A Method for Predicting the Output Cross Power Spectral Density Between Selected Variables in Response to Arbitrary Random Excitations*, The MSC 1988 World Users Conf. Proc., Vol. I, Paper No. 19, March, 1988.

Parthasarathy, Alwar; Elzeki, Mohamed; Abramovici, Vivianne. PSDTOOL-A DMAP *Enhancement to Harmonic/ Random Response Analysis in MSC/NASTRAN*, The MSC 1993 World Users' Conf. Proc., Paper No. 36, May, 1993.

Robinson, J. H.; Chiang, C. K.; Rizzi, S. A. *Nonlinear Random Response Prediction Using MSC/NASTRAN*, National Aeronautics and Space Administration, Hampton, VA, Langley Research Center, October, 1993.

Robinson, Jay H.; Chiang, C. K. *An Equivalent Linearization Solution Sequence for MSC/NASTRAN*, The MSC 1993 World Users' Conf. Proc., Paper No. 35, May, 1993.

Schwab, H. L.; Caffrey, J.; Lin, J. *Fatigue Analysis Using Random Vibration*, MSC 1995 World Users' Conf. Proc., Paper No. 17, May, 1995.

Shieh, Niahn Chung. *Investigation of Swept Sine on Random Load*, The 4th MSC Taiwan Users' Conf., Paper No. 18, November, 1992, in Chinese.

Zins, J. *Random Vibration and Acoustic Analysis Using ARI RANDOM a NASTRAN Postprocessor*, Proc. of the MSC/NASTRAN Eur. Users' Conf., Paper No. 4, May, 1984.

DYNAMICS – REDUCTION METHODS

Abdallah, Ayman A.; Barnett, Alan R.; Widrick, Timothy W.; Manella, Richard T.; Miller, Robert P. *Stiffness-Generated Rigid-Body Mode Shapes for Lanczos Eigensolution with Support DOF Via a MSC/NASTRAN DMAP Alter*, MSC 1994 World Users' Conf. Proc., Paper No. 10, June, 1994.

Flanigan, Christopher C. *Implementation of the IRS Dynamic Reduction Method in MSC/NASTRAN*, The MSC 1990 World Users Conf. Proc., Vol. I, Paper No. 13, March, 1990.

Fox, Gary L. *Evaluation and Reduction of Errors Induced by the Guyan Transformation*, Tenth NASTRAN Users' Colloq., pp. 233-248, May, 1982, (NASA CP-2249).

Komzsik, L.; Dilley, G. *Practical Experiences with the Lanczos Method*, Proc. of the MSC/NASTRAN Users Conf., Paper No. 13, March, 1985.

Kuang, Jao-Hwa; Lee, Chung-Ying. *On a Guyan-Reduction Recycled Eigen Solution Technique*, The 2nd Annual MSC Taiwan Users Conf., Paper No. 13, October, 1990.

Levy, Roy. *Guyan Reduction Solutions Recycled for Improved Accuracy*, NASTRAN: Users' Exper., pp. 201-220, September, 1971, (NASA TM X-2378).

Maekawa, Seiyou. *Effect of Guyan Reduction and Generalized Dynamic Reduction*, The Second MSC/NASTRAN User's Conf. in Japan, October, 1984, in Japanese.

Mera, A. *MSC/NASTRAN Normal Mode Analysis with GDR: An Evaluation of Limitations*, MSC/NASTRAN Users' Conf. Proc., Paper No. 27, March, 1985.

Mera, Andrew. *Static Reduction and Symmetry Transformation of Large Finite Element Models*, Proc. of the MSC/NASTRAN Users' Conf., Paper No. 12, March, 1979.

Vandepitte, D.; Wijker, J. J.; Appel, S.; Spiele, H. *Normal Modes Analysis of Large Models, and Applications to Ariane 5 Engine Frame*, Proc. of the 18th MSC Eur. Users' Conf., Paper No. 6, June, 1991.

Vollan, Arne; Kaporin, Igor; Babikov, Pavel. *Practical Experience with Different Iterative Solvers for Linear Static and Modal Analysis of Large Finite Element Models*, Proc. of the 21st MSC European Users' Conf., Italian Session, September, 1994.

Walker, James W. *Evaluation of MSC/NASTRAN Generalized Dynamic Reduction and Response Spectrum Analysis by Comparison with STARDYNE*, MSC/NASTRAN Users' Conf., March, 1978.

DYNAMICS – RESPONSE SPECTRUM

Barber, Pam; Arden, Kevin. *Dynamic Design Analysis Method (DDAM) Using MSC/NASTRAN*, MSC 1994 World Users' Conf. Proc., Paper No. 31, June, 1994.

Cutting, Fred. *Individual Modal Accelerations as the Result of a Shock Response Spectra Input to a Complex Structure*, The 1989 MSC World Users Conf. Proc., Vol. I, Paper No. 11, March, 1989.

Gassert, W.; Wolf, M. *Floor Response Spectra of a Reactor Building Under Seismic Loading Calculated with a 3-D Building Model*, Proc. of the MSC/NASTRAN Eur. Users' Conf., June, 1983.

Hirata, M.; Ishikawa, K.; Korosawa, M.; Fukushima, S.; Hoshina, H. *Seismic Analysis of Plutonium Glovebox by MSC/NASTRAN*, January, 1993.

Kubota, Minoru. *Response Spectrum Analysis of Underground Tank Dome Roof Using Image Superelement Method*, The Fifth MSC/NASTRAN User's Conf. in Japan, October, 1987, in Japanese.

Moharir, M. M. *Shock Wave Propagation and NASTRAN Linear Algorithms*, MSC/NASTRAN Users' Conf. Proc., Paper No. 7, March, 1986.

Overbye, Vern D. *MSC/NASTRAN Dynamic Analysis: Modal or Direct?, MSC/NASTRAN Users' Conf. Proc.*, Paper No. 6, March, 1986.

Parris, R. A. *Aspects of Seismic Analysis Using MSC/NASTRAN*, Proc. of the MSC/NASTRAN Eur. Users' Conf., Paper No. 7, June, 1983.

Parthasarathy, A. *CONSPEC-A DMAP System for Conventional Response-Spectrum Analysis in MSC/NASTRAN*, MSC/NASTRAN Users' Conf. Proc., Paper No. 8, March, 1986.

Petteno, L.; Rossetto, P. *Tecnomare Experiences in DMAP Processing*, Proc. of the MSC/NASTRAN Eur. Users' Conf., May, 1984.

Rose, Ted L. *Using Superelements for Response Spectrum and Other Handy Alters*, The 1989 MSC World Users Conf. Proc., Vol. II, Paper No. 45, March, 1989.

Walker, James W. *Evaluation of MSC/NASTRAN Generalized Dynamic Reduction and Response Spectrum Analysis by Comparison with STARDYNE*, MSC/NASTRAN Users' Conf., March, 1978.

DYNAMICS – SEISMIC

Bonaldi, P.; Peano, A.; Ruggeri, G.; Venturuzzo, M. *Seismic and Impact Analyses of Nuclear Island Buildings of Italian Unified Nuclear Design*, Proc. of the 15th MSC/NASTRAN Eur. Users' Conf., October, 1988.

Burroughs, John W. *An Enhancement of NASTRAN for the Seismic Analysis of Structures*, Ninth NASTRAN Users' Colloq., pp. 79-96, October, 1980, (NASA CP-2151).

Chen, J. T.; Chyuan, S. W.; Yeh, C. S.; Hong, H. K. *Comparisons of Analytical Solution and MSC/NASTRAN Results on Multiple-Support Motion of Long Bridge During Earthquake*, The 4th MSC Taiwan Users' Conf., Paper No. 20, November, 1992.

Chen, J. T.; Hong, H. K.; Yen, C. S.; Chyuan, S. W. *Integral Representations and Regularizations for a Divergent Series Solution of a Beam Subjected to Support Motions*, Earthquake Engineering and Structural Dynamics, Vol. 25, pp. 909-925, 1996.

Chen, Yohchia. *Nonlinear Seismic Analysis of Bridges: Practical Approach and Comparative Study*, The MSC 1993 World Users' Conf. Proc., Paper No. 57, May, 1993.

Chen, Yohchia. *Refined Analysis for Soil-Pipe Systems*, MSC 1994 World Users' Conf. Proc., Paper No. 38, June, 1994.

Dahlgren, F.; Citrolo, J.; Knutson, D.; Kalish, M. *Dynamic Analysis of the BPX Machine Structure*, Proc. of the 14th IEEE/NPSS Symp. on Fusion Engineering, Vol. 1, pp. 47-49, 1992.

Fallet, P.; Derivery, J. C. *MSC/NASTRAN Earthquake Analysis*, NASTRAN User's Conf., May, 1979.

Hill, Ronald G. *Nonlinear Seismic Analysis of a Reactor Structure with Impact Between Core Components*, NASTRAN: Users' Exper., pp. 397-418, September, 1975, (NASA TM X-3278).

Hirata, M.; Ishikawa, K.; Korosawa, M.; Fukushima, S.; Hoshina, H. *Seismic Analysis of Plutonium Glovebox by MSC/NASTRAN*, January, 1993.

Liepins, Atis A.; Nazemi, Hamid. *Virtual Mass of Fluid in Egg-Shaped Digesters*, The MSC 1993 World Users' Conf. Proc., Paper No. 77, May, 1993.

Nomura, Yoshio. *Seismic Response Analysis by MSC/NASTRAN for Coupled Structure - Ground and Pile*, The Fifth MSC/NASTRAN User's Conf. in Japan, October, 1987, in Japanese.

Pamidi, M. R.; Pamidi, P. R. *Modal Seismic Analysis of a Nuclear Power Plant Control Panel and Comparison with SAP IV*, NASTRAN: Users' Exper., pp. 515-530, October, 1976, (NASA TM X-3428).

Tsaur, D. H.; Chyuan, S. W.; Chen, J. T. *On the Accuracy of MSC/NASTRAN on Response of Two-Span Beams to Spatially Varying Seismic Excitation*, The 4th MSC Taiwan Users' Conf., Paper No. 9, November, 1992.

Yiak, K. C.; Pezeshk, S. *Seismic Study of a Cable-Stayed Bridge*, Proceedings of the Structures Congress '94, Atlanta, 1994.

Zhou, Hongye; Chen, Youping. *The Influence of Phase-Difference Effects on Earthquake Response of Cable-Stayed Bridges*, MSC 1994 World Users' Conf. Proc., Paper No. 37, June, 1994.

DYNAMICS – TRANSIENT ANALYSIS

(Korean). *Vibration Analysis for Outercase in Drum Washer and Floor*, MSC 1994 Korea Users' Conf. Proc., December, 1994, in Korean.

Aslani, Chang; Yatheendar, Manicka; Visintainer, Randal, H.; Rohweder, David S.; Lopez de Alda, Juan. *Simulation of Proving Ground Events for Heavy Truck Cabs Using Adams, MSC/NASTRAN, and P/FATIGUE*, MSC 1994 World Users' Conf. Proc., Paper No. 5, June 1994.

Barber, Pam; Arden, Kevin. *Dynamic Design Analysis Method (DDAM) Using MSC/NASTRAN*, MSC 1994 World Users' Conf. Proc., Paper No. 31, June, 1994.

Barnett, Alan R.; Ibrahim, Omar M.; Sullivan, Timothy L.; Goodnight, Thomas W. *Transient Analysis Mode Participation for Modal Survey Target Mode Selection Using MSC/NASTRAN DMAP*, MSC 1994 World Users' Conf. Proc., Paper No. 8, June, 1994.

Bellinger, Dean. *Dynamic Analysis by the Fourier Transform Method with MSC/NASTRAN*, MSC 1995 World Users' Conf. Proc., Paper No. 10, May, 1995.

Chargin, M. L.; Dunne, L. W.; Herting, D. N. *Nonlinear Dynamics of Brake Squeal*, MSC 1996 World Users' Conf. Proc., Vol. V, Paper No. 47, June, 1996.

Cifuentes, A. O.; Herting, D. N. *Transient Response of a Beam to a Moving Mass Using a Finite Element Approach, Innovative Numerical Methods in Engineering*, Proc. of the Fourth Int. Symp. on Numerical Methods in Engineering, Springer-Verlag, pp. 533-539, March, 1986.

Cifuentes, A. O.; Lalapet, S. *Transient Response of a Plate to an Orbiting Mass*, Proc. of the Second Panamerican Cong. of Applied Mechanics, January, 1991.

Cifuentes, A. O.; Lalapet, S. *A General Method to Determine the Dynamic Response of a Plate to an Orbiting Mass*, Computers and Structures, Vol. 42, No. 1, pp. 31-36, 1992.

Dai, Chung C.; Yang, Jackson C. S. *Direct Transient Analysis of a Fuse Assembly by Axisymmetric Solid Elements*, Thirteenth NASTRAN Users' Colloq., pp. 431-452, May, 1985, (NASA CP-2373).

Deloo, Ph.; Klein, M. *In-Orbit Disturbance Sensitivity Analysis of the Hubble Space Telescope New Solar Arrays*, Proc. of the 19th MSC European Users' Conf., Paper No. 11, September, 1992.

Everstine, Gordon C. *A NASTRAN Implementation of the Doubly Asymptotic Approximation for Underwater Shock Response*, NASTRAN: Users' Exper., pp. 207-228, October, 1976, (NASA TM X-3428).

Flanigan, Christopher C. *Accurate Enforced Motion Analysis Using MSC/NASTRAN Superelements*, MSC 1994 World Users' Conf. Proc., Paper No. 25, June, 1994.

Frye, John W. *Transient Analysis of Bodies with Moving Boundaries Using NASTRAN*, NASTRAN: Users' Exper., pp. 377-388, September, 1975, (NASA TM X-3278).

Hirata, M.; Ishikawa, K.; Korosawa, M.; Fukushima, S.; Hoshina, H. *Seismic Analysis of Plutonium Glovebox by MSC/NASTRAN*, January, 1993.

Katnik, Richard B.; Deutschel, Brian; Cherukuri, Ravi. *Transient Response of a Vehicle Over Road Bumps Using the Fourier Transform in a Modal Subspace*, The MSC 1992 World Users' Conf. Proc., Vol. I, Paper No. 6, May, 1992.

Kim, Hyoung M.; Bartkowicz, Theodoore J.; Van Horn, David A. *Data Recovery and Model Reduction Methods for Large Structures*, The MSC 1993 World Users' Conf. Proc., Paper No. 23, May, 1993.

Larkin, Paul A.; Miller, Michael W. *STS Coupled Loads Analysis Using MSC/NASTRAN*, MSC/NASTRAN Users' Conf. Proc., Paper No. 18, March, 1985.

Lee, J. H.; Tang, J. H. K. *Dynamic Response of Containments Due to Shock Wave*, Proc. of the Int. Conf. on Containment Design, pp. 25-32, June, 1984.

Lee, Sang H.; Bock, Tim L.; Hsieh, Steve S. *Adaptive Time Stepping Algorithm for Nonlinear Transient Analysis*, The MSC 1988 World Users Conf. Proc., Vol. II, Paper No. 54, March, 1988.

Leifer, Joel; Gross, Michael. *Non-Linear Shipboard Shock Analysis of the Tomahawk Missile Shock Isolation System*, 58th Shock and Vibration Symp., Vol. 1, pp. 97-117, October, 1978, (NASA CP-2488).

Lipman, Robert R. *Computer Animation of Modal and Transient Vibrations*, Fifteenth NASTRAN Users' Colloq., pp. 111-117, August, 1987, (NASA CP-2481).

Mattana, G.; Miranda, D. *MSC/NASTRAN Applications in P-180 Analysis*, Proc. of the MSC/NASTRAN First Italian Users' Conf., October, 1987.

McMeekin, Michael; Kirchman, Paul. *An Advanced Post Processing Methodology for Viewing MSC/NASTRAN Generated Analyses Results*, MSC 1994 World Users' Conf. Proc., Paper No. 21, June, 1994.

McNamee, Martin J.; Zavareh, Parviz. *Nonlinear Transient Analysis of a Shock Isolated Mechanical Fuse*, The MSC 1990 World Users Conf. Proc., Vol. I, Paper No. 21, March, 1990.

Neilson, H. C.; Everstine, G. C.; Wang, Y. F. *Transient Response of Submerged Fluid-Coupled Double-Walled Shell Structure to a Pressure Pulse*, J. of the Acoustic Soc. of America, Vol. 70, No. 6, pp. 1776-1782, December, 1981.

Pamidi, P. R. *On the Append and Continue Features in NASTRAN*, Seventh NASTRAN Users' Colloq., pp. 405-418, October, 1978, (NASA CP-2062).

Rose, Ted. *A Method to Apply Initial Conditions in Modal Transient Solutions*, The MSC 1991 World Users' Conf. Proc., Vol. I, Paper No. 13, March, 1991.

Rose, Ted.; McNamee, Martin. *A DMAP Alter to Allow Amplitude- Dependent Modal Damping in a Transient Solution*, MSC 1996 World Users' Conf. Proc., Vol. V, Paper No. 50, June, 1996.

Swan, Jim. *A DMAP Alter for Interface Loads Across Superelements in Dynamic Analyses*, The MSC 1992 World Users' Conf. Proc., Vol. I, Paper No. 23, May, 1992.

Tang, C. C. *Space Station Freedom Solar Array Wing: Nonlinear Transient Analysis of Plume Impingement Load*, MSC 1994 World Users' Conf. Proc., Paper No. 35, June, 1994.

Urban, Michael R.; Dobyns, Alan. *MSC/NASTRAN Transient Analysis of Cannon Recoil Loads on Composite Helicopters*, The MSC 1991 World Users' Conf. Proc., Vol. I, Paper No. 28, March, 1991.

West, Timothy S. *Approximate Dynamic Model Sensitivity Analysis For Large, Complex Space Structures*, MSC 1996 World Users' Conf. Proc., Vol. I, Paper No. 6, June, 1996.

Wingate, Robert T.; Jones, Thomas C.; Stephens, Maria V. *NASTRAN Postprocessor Program for Transient Response to Input Accelerations*, NASTRAN: Users' Exper., pp. 707-734, September, 1973, (NASA TM X-2893).

Yang, Jackson C. S.; Goeller, Jack E.; Messick William T. *Transient Analysis Using Conical Shell Elements*, NASTRAN: Users' Exper., pp. 125-142, September, 1973, (NASA TM X-2893).