NX Nastran Dynamic Analysis

Transient, Frequency Response, Random Response, and Response Spectrum Analysis Types

Finite Element Analysis
Predictive Engineering

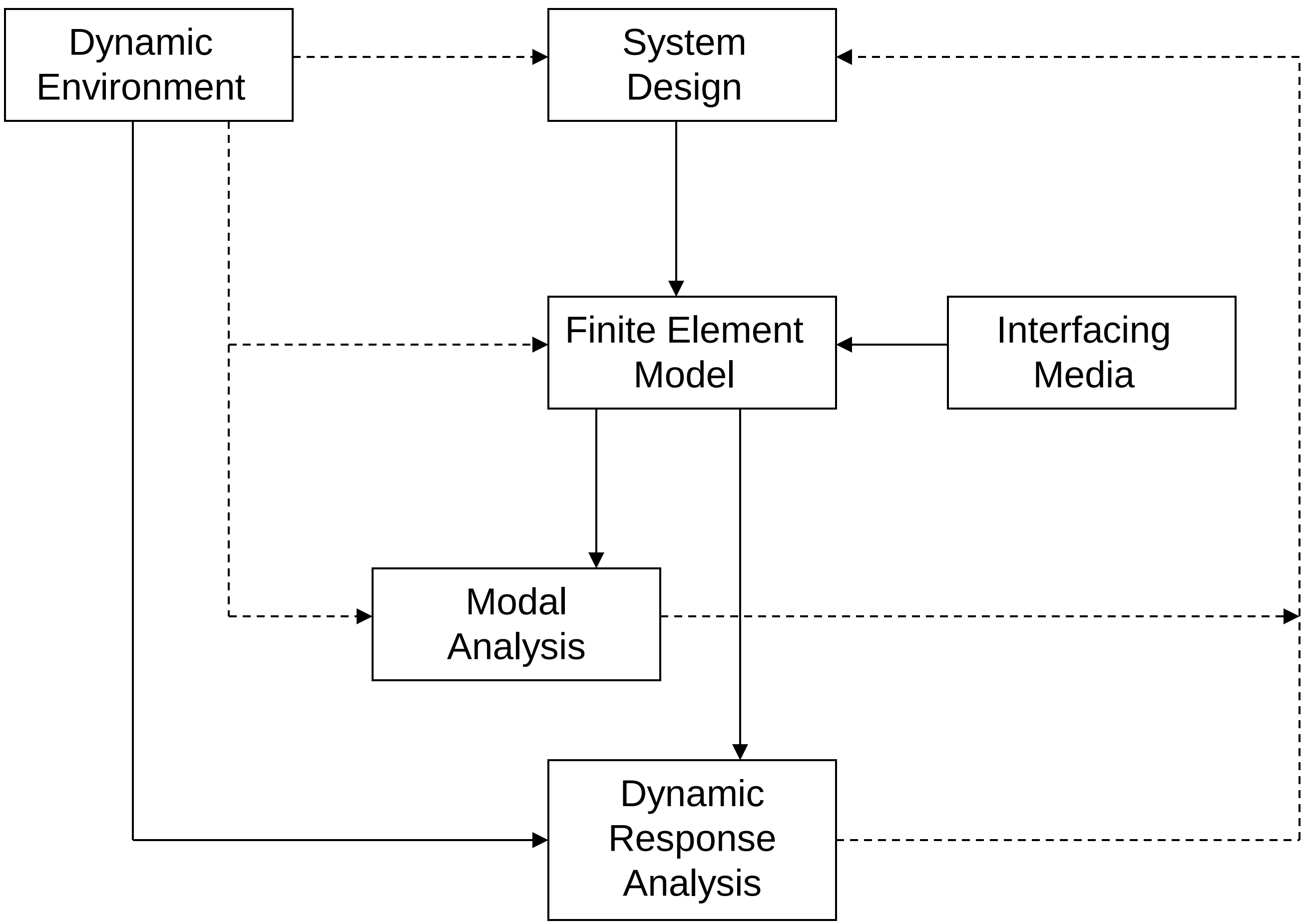
Review of Fundamentals

NX Nastran Dynamic Analysis

Dynamic Analysis Process Overview

- NX Nastran is a general purpose Finite Element Analysis solver capable of simulating a broad range of engineering problems in many different industries.
- The solver can be run as a stand-alone solver using an existing NASTRAN input file or in conjunction with EDS's Finite Element Analysis Pre and Post-Processor, FEMAP.
- NX Nastran 1.0 is analogous with MSC NASTRAN 2001, release 9 and runs on Windows NT, 2000, XP, and 2003 server. Versions of the stand-alone solver are available on various UNIX (HP, Sun, SGI, and IBM) and LINUX platforms.

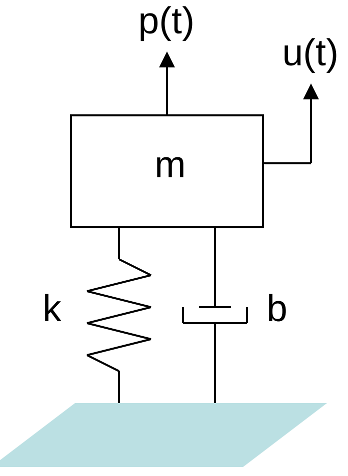
Dynamic Analysis Process Overview



Single DOF System

- Dynamic equation of motion:

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



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

u, $\dot{u}$, $\ddot{u}$ and p are time varying in general.
m, b, and k are constants

Units

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

- Fundamental units

  - Length, L (inch, meter)
  - Mass, M (slug, kilogram)
  - Time, T (second)

- Fundamental and derived units

  - m M
  - b MT^{-1}
  - k MT^{-2}
  - p MLT^{-2}
  - ụ L
  - $\dot{u}$ LT^{-1}
  - $\ddot{u}$ LT^{-2}

Units

Variable	Dimensions	English	Metric
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

L = Length
M = Mass
T = Time
- = Dimensionless

Units

Variable	Dimensions	English	Metric
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
Moment	ML^2T^{-2}	in-lb	N-m
Mass Density	ML^{-3}	in-sec^2/in^4	kg/m^3
Young's Modulus	$ML^{-1}T^{-2}$	lb/in^2	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 Coeff.	MT-1	lb-sec/in	N-sec/m
Stress	$ML^{-1}T^{-2}$	lb/in^2	Pa: N/m^2
Strain	-	-	-

Single DOF System – Undamped Free Vibrations

- Dynamic equation

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

- Solution

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

$$\omega_n = \sqrt{\frac{k}{m}} = \text{natural frequency (rad/sec)}$$

$$f_n = \frac{\boldsymbol{\omega_n}}{\mathbf{2\pi}} = \text{natural frequency (cycles/sec)}$$

- Initial conditions

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

$$A = \frac{\dot{u}}{\omega_n}(t = 0)$$

$u(0)$ and $\dot{u}(0)$ are known

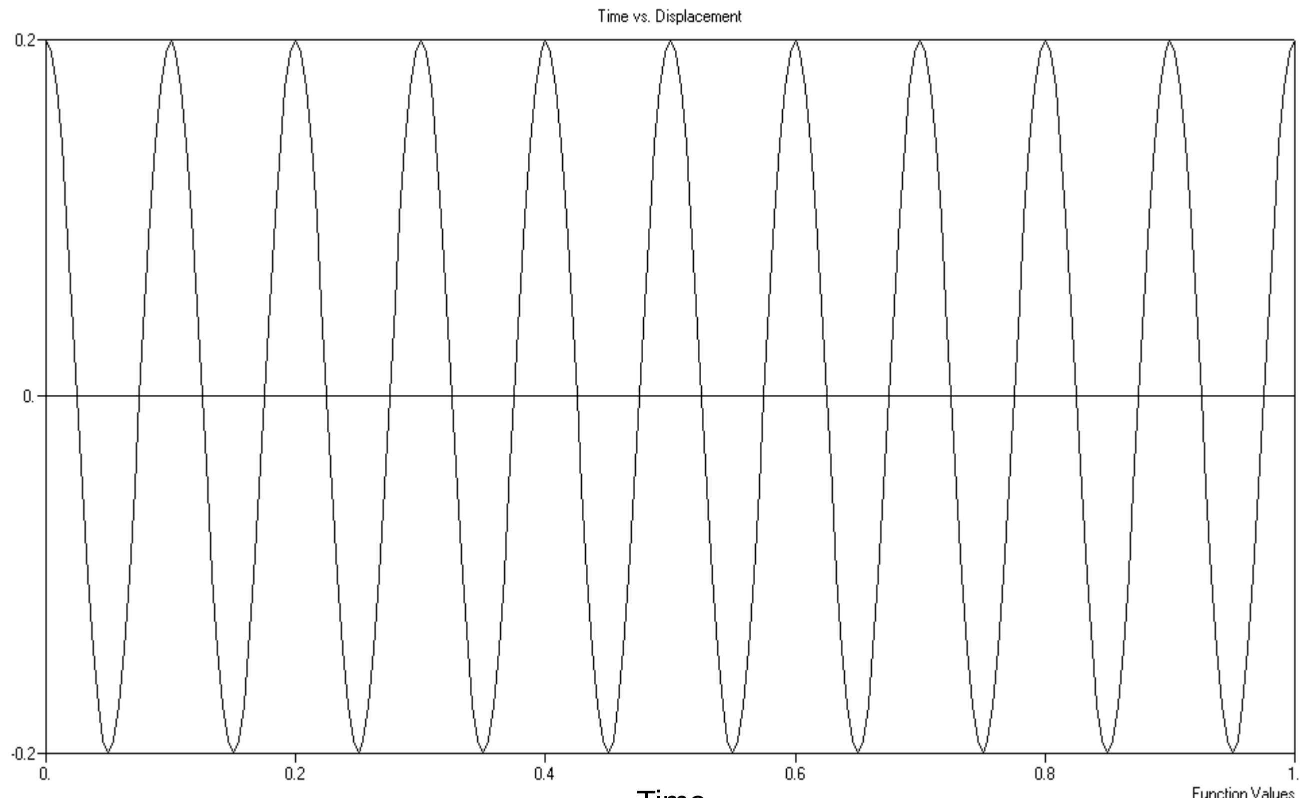
- Finally

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

Finite Element Analysis
Predictive Engineering

Single DOF System – Undamped Free Vibrations

Displacement



Time

SDOF Oscillator – Nonzero initial conditions

Single DOF System – Damped Free Vibrations

- Dynamic Equation

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

- Critical Damping

$$b_c = 2\sqrt{km} = 2m\omega_n$$

- Initial conditions

$$\xi = \frac{b}{b_c}$$

- The amount of damping determines the form of the solution
  - Underdamped

$$b < b_c$$

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

Damped Natural Frequency:

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

Single DOF System – Damped Free Vibrations

- Critically damped

$$b = b_c$$

- No oscillation occurs

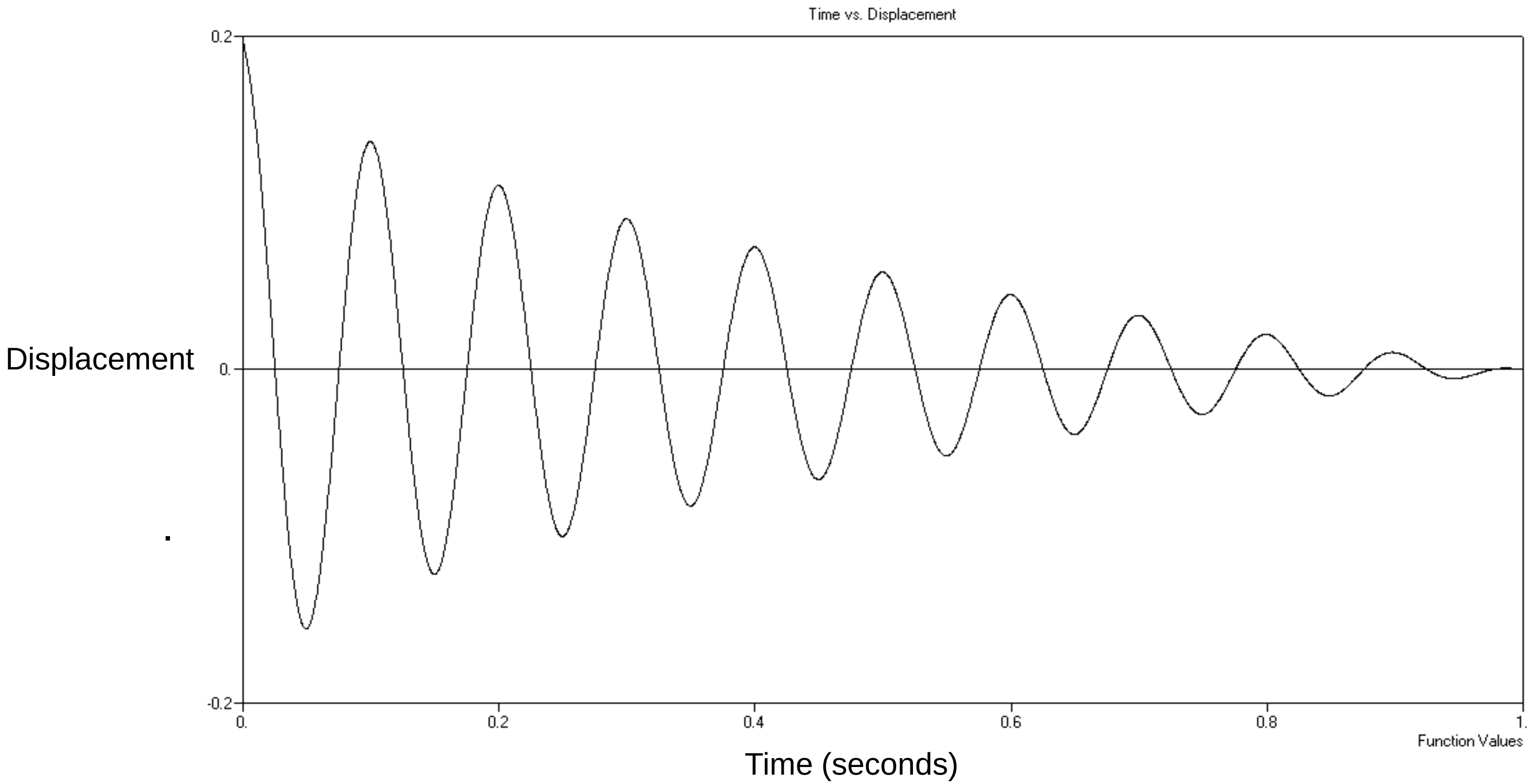
$$u(t) = (A + Bt)e^{-bt/2m}$$

- Overdamped

$$b > b_c$$

  - No oscillation occurs. The system gradually returns to Equilibrium (at rest, undisplaced) position.
- The usual analysis case is underdamped
- Structures have viscous damping in the 0-10% range

Single DOF System – Damped Free Vibrations



Single DOF System – Undamped Sinusoidal Vibrations

- Dynamic Equation

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

  - Where ω = forcing frequency
- Solution

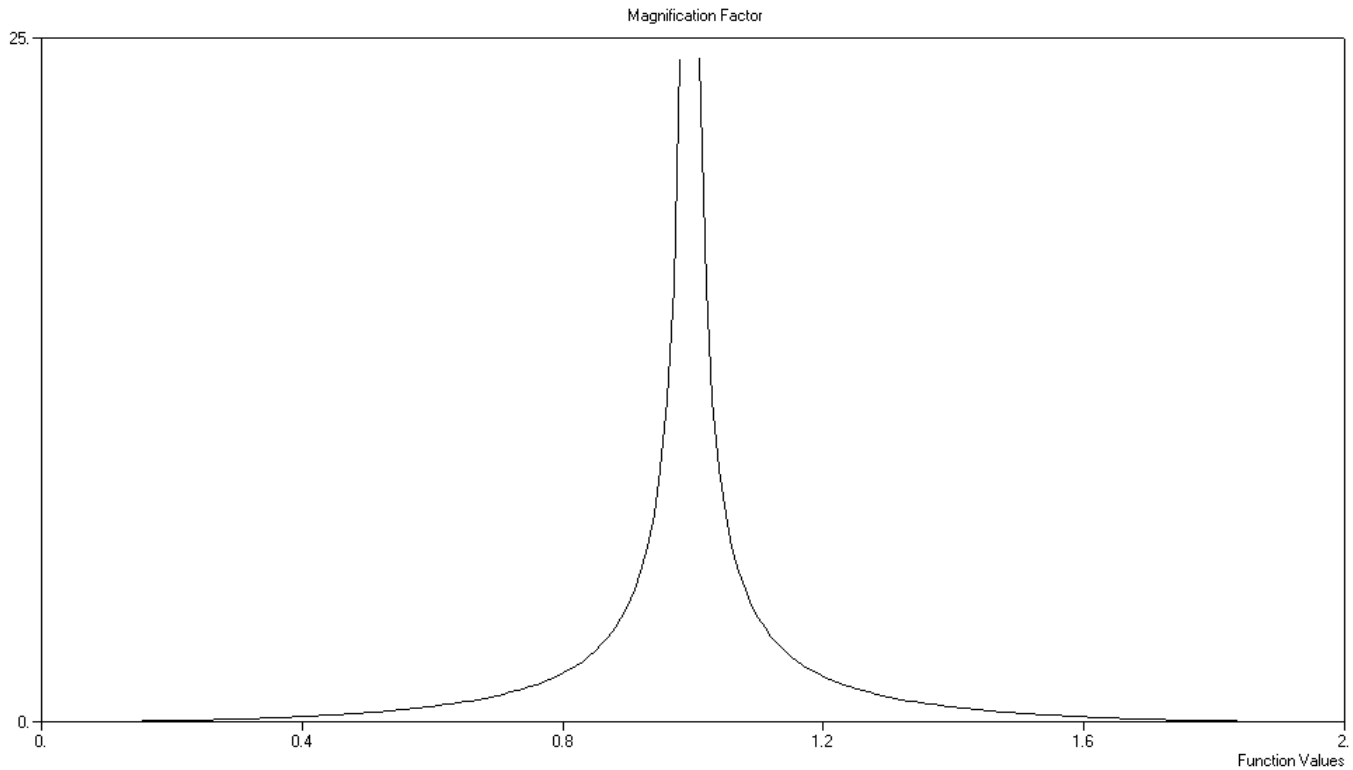
$$u(t) = \underbrace{A \sin\omega_d t + B \cos \omega_d t}_{\text{Initial Conditions}} + \underbrace{\frac{P/k}{1-\omega^2/\omega_n^2} \sin\omega t}_{\text{Steady-State}}$$

where

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

- Steady-state Solution
  - P/k is the static response
  - $\dfrac{1}{1-\omega^2/\omega_n^2}$ is the dynamic magnification factor

Magnification factor



Frequency ((0.5)*Hz)

Finite Element Analysis Predictive Engineering

Single DOF System – Damped Sinusoidal Vibrations

- Dynamic Equation

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

- Transient solution decays rapidly and is of no interest
- Steady-state Solution

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

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

- θ is phase <u>lead</u>

Single DOF System – Damped Sinusoidal Vibrations

- For $\dfrac{\omega}{\omega_n} >> 1$
  - Magnification factor $\longrightarrow$ 1(Static Solution)
  - Phase angle $\longrightarrow$ 360°(Response is in phase with the force)
- For $\dfrac{\omega}{\omega_n} << 1$
  - Magnification factor $\longrightarrow$ 0 (No Response)
  - Phase angle $\longrightarrow$ 360°(Response has opposite sign of force)
- For $\dfrac{\omega}{\omega_n} \approx 1$
  - Magnification factor $\longrightarrow$ $1/2\xi$
  - Phase angle $\longrightarrow$ 270°

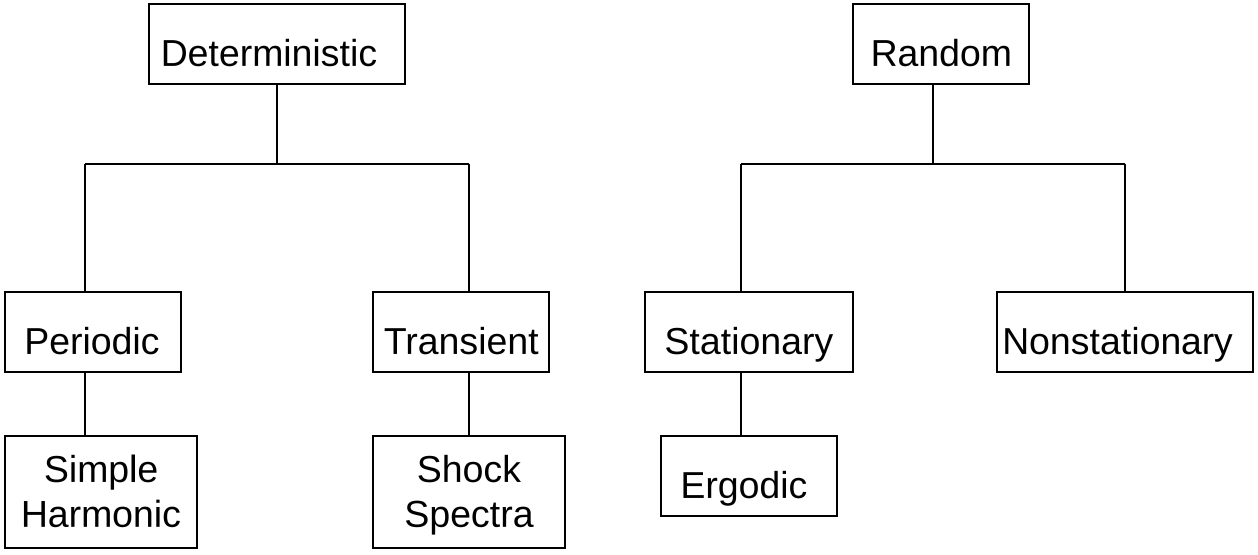
Multi-Degree of Freedom System

- The equation becomes:

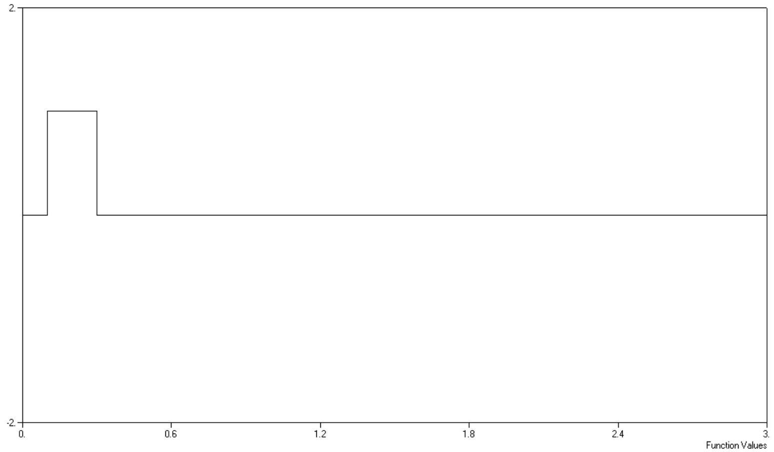
$$[M]\{\ddot{u}\} + [B]\{\dot{u}\} + [K]\{u\} = \{P\}$$

- Where: **{u}** = Displacement
  [M] = Mass Matrix
  [B] = Damping Matrix
  [K] = Stiffness Matrix
  {P} = Forcing function

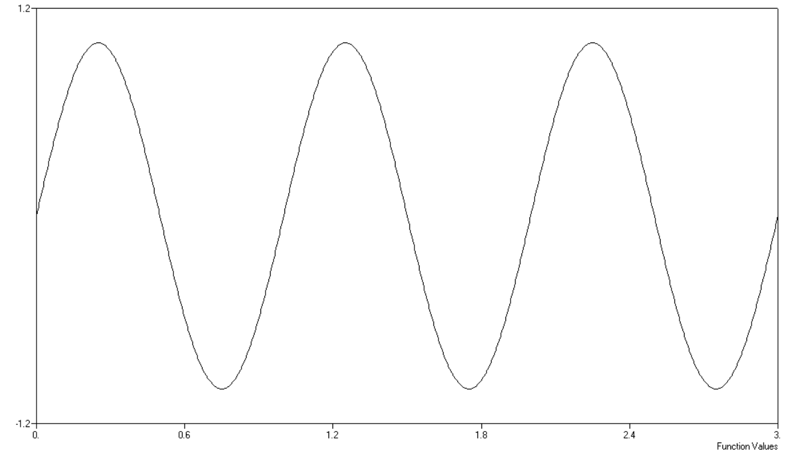
Classification of Dynamic Environments



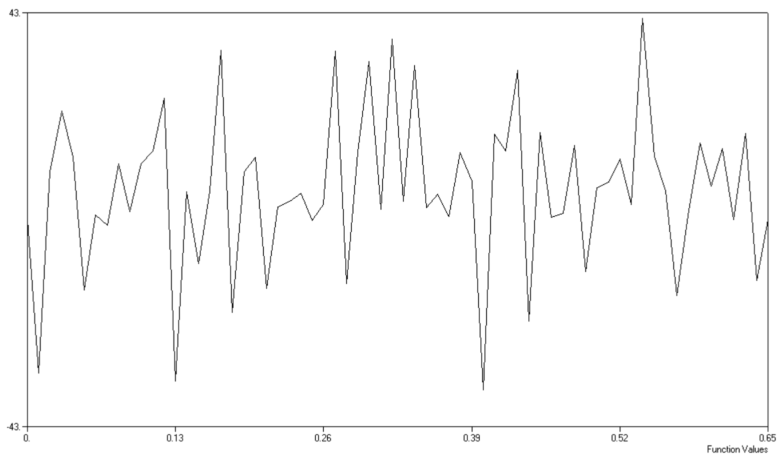
NX Nastran Dynamic Excitations



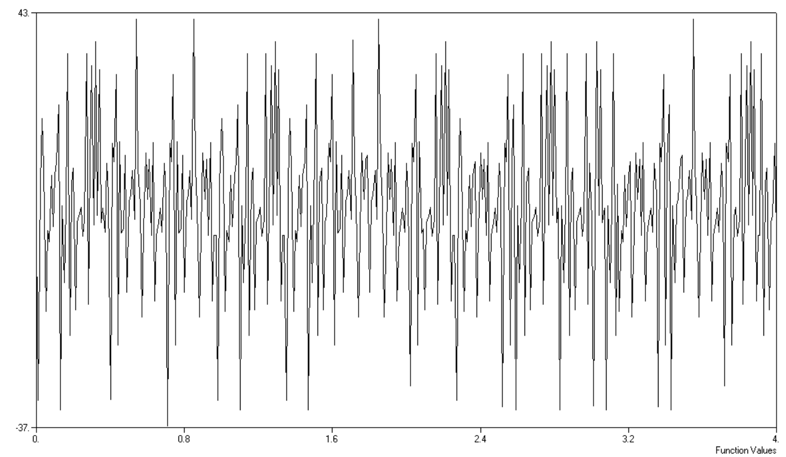
Pulse



Sinusoidal



Transient



Random

Finite Element Analysis
Predictive Engineering

Finite Element Dynamic Modeling Considerations

- Frequency Range
- Grid Points/constraints/elements
- Linear versus nonlinear behavior
- Interaction with adjacent media
- Test/measured data integration
- Damping

Dynamics Modeling Input

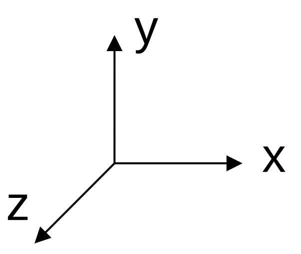
NX Nastran Dynamic Analysis

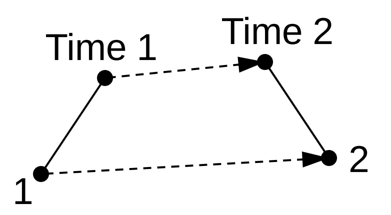
NX Nastran Input File Setup

- FMS and NASTRAN Statements – File allocation and system call
- Executive Control Section – Solution type, time allowed, system diagnostics
    - CEND – Required Delimiter Entry
- Case Control Section – Output requests, selects certain Bulk Data items
    - BEGIN BULK – Required Delimiter Entry
- Bulk Data Section – Structural Model definition, solution conditions
    - ENDDATA – Required Delimiter Entry

Finite Element Analysis

- Real World is not comprised of only SDOF systems!
- Finite Elements are used to model the mass, damping, and stiffness of complex systems and structures.
- Degrees of freedom (DOF) are independent coordinates that describe the motion of the structure at any instant in time.
- GRIDs are used to model the continuous structire as a discrete entity.
- Each GRID may have six DOFs: 3 in translation directions (X, Y, and Z) and 3 in rotation about the X, Y, and Z axes.





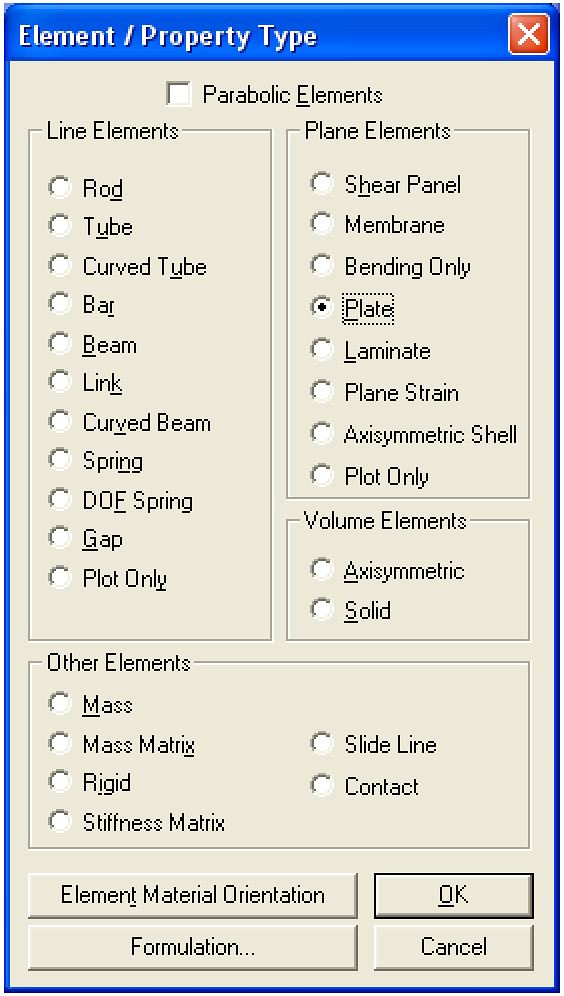
- Book-keeping is done via the matrices that define the relationships between the DOFs.

Commonly Used Elastic Elements

		Number of DOFs
One-Dimensional Geometry		
ROD	Pin-ended rod	4
BAR	Prismatic beam	12
BEAM	Straight beam with warping	12
BEND	Curved beam, pipe, or elbow	12
Two-Dimensional Geometry		
TRIA3	Triangular plate	15
QUAD4	Quadrilateral plate	20
SHEAR	4-sided shear panel	8
TRIA6	Triangular plate with midside nodes	30
QUAD8	Quadrilateral plate with midside nodes	40
Three-Dimensional Geometry		
HEXA	Solid with six quadrilateral faces	24-60
TETRA	Solid with four triangular faces	12-30
PENTA	Solid with two triangular faces and three quadrilateral faces	18-45
Zero-Dimensional Geometry		
ELAS	Simple spring connecting two degrees of freedom	2

Commonly Used Elastic Elements

Use the Model -> Property command and in the Define Property dialog box, click *Elem/Property Type* Button. The desired Element/Property Type can then be chosen from the dialog box.



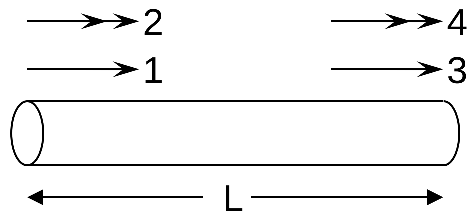
Coupled vs. Lumped Mass

- Coupled mass is generally more accurate than lumped mass.
- Lumped mass is preferred for computational speed in dynamic analysis
- User-selectable coupled mass matrix for elements
  - PARAM, COUPMASS, 1 to select coupled mass
  - The Default is lumped mass
- Elements which have either lumped or coupled mass:
  - BAR, BEAM, CONROD, HEXA, PENTA, QUAD4, QUAD8, ROD, TETRA, TRIA3, TRIA6, TRIAX6, TUBE
- Elements which have lumped mass only:
  - CONEAX, SHEAR
- Elements which have coupled mass only:
  - BEND, HEX20

Coupled vs. Lumped Mass

- Lumped mass contains only diagonal, translational components (no rotational components).
- Coupled mass contains off-diagonal translational components, as well as, rotations for BAR (no torsion), BEAM, and BEND elements.

ROD Finite Element



Length, L
Area, A
Torsional Constant, J
Young's Modulus, E
Shear Modulus, G

Stiffness Matrix:

$$k = \begin{bmatrix} \frac{AE}{L} & 0 & \frac{-AE}{L} & 0 \\ 0 & \frac{GJ}{L} & 0 & \frac{-GJ}{L} \\ \frac{-AE}{L} & 0 & \frac{AE}{L} & 0 \\ 0 & \frac{-GJ}{L} & 0 & \frac{GJ}{L} \end{bmatrix}$$

Classical consistent Mass:

$$m = \rho AL \begin{bmatrix} 1/3 & 0 & 1/6 & 0 \\ 0 & \frac{I\rho}{3A} & 0 & \frac{I\rho}{6A} \\ 1/6 & 0 & 1/3 & 0 \\ 0 & \frac{I\rho}{6A} & 0 & \frac{I\rho}{3A} \end{bmatrix}$$

ROD Finite Element

NX Nastran Lumped Mass:

$$m = \rho AL \begin{bmatrix} 1/2 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 1/2 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

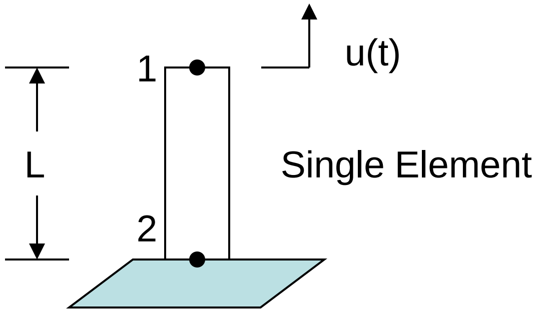
NX Nastran Coupled Mass:

$$m = \rho AL \begin{bmatrix} 5/12 & 0 & 1/12 & 0 \\ 0 & 0 & 0 & 0 \\ 1/12 & 0 & 5/12 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

The translational terms represent the average of lumped mass and classical consistent mass. This average is best for ROD and BAR elements.

Justification of Nastran Coupled Mass Convention

Consider a fixed-free rod:



Exact quarter-wave natural frequency:

$$\omega_{1/4} = \frac{\pi\sqrt{E/\rho}}{2L} = 1.5708\,\frac{\sqrt{E/\rho}}{L}$$

Justification of Nastran Coupled Mass Convention

Different approximations

- Lumped Mass

$$\omega_L = \sqrt{2}\,\frac{\sqrt{E/\rho}}{L} = 1.414\,\frac{\sqrt{E/\rho}}{L} \quad (-10\%)$$

- Classical consistent mass

$$\omega_L = \sqrt{3}\,\frac{\sqrt{E/\rho}}{L} = 1.732\,\frac{\sqrt{E/\rho}}{L} \quad (+10\%)$$

NX Nastran

- Coupled Mass

$$\omega_L = \sqrt{12/5}\,\frac{\sqrt{E/\rho}}{L} = 1.549\,\frac{\sqrt{E/\rho}}{L} \quad (-1.4\%)$$

Mass Units

- NX Nastran assumes consistent units. BE CAREFUL!!!!
- Weight units may be input instead of mass units if this is more convenient. The PARAM,WTMASS must then be used to convert them to mass.
- Weight-to-mass conversion:

  Mass = (1/G)* Weight (G=Gravity Acceleration)
  Mass Density = (1/G) * Weight Density

- PARAM,WTMASS, factor performs conversion with factor = 1/G

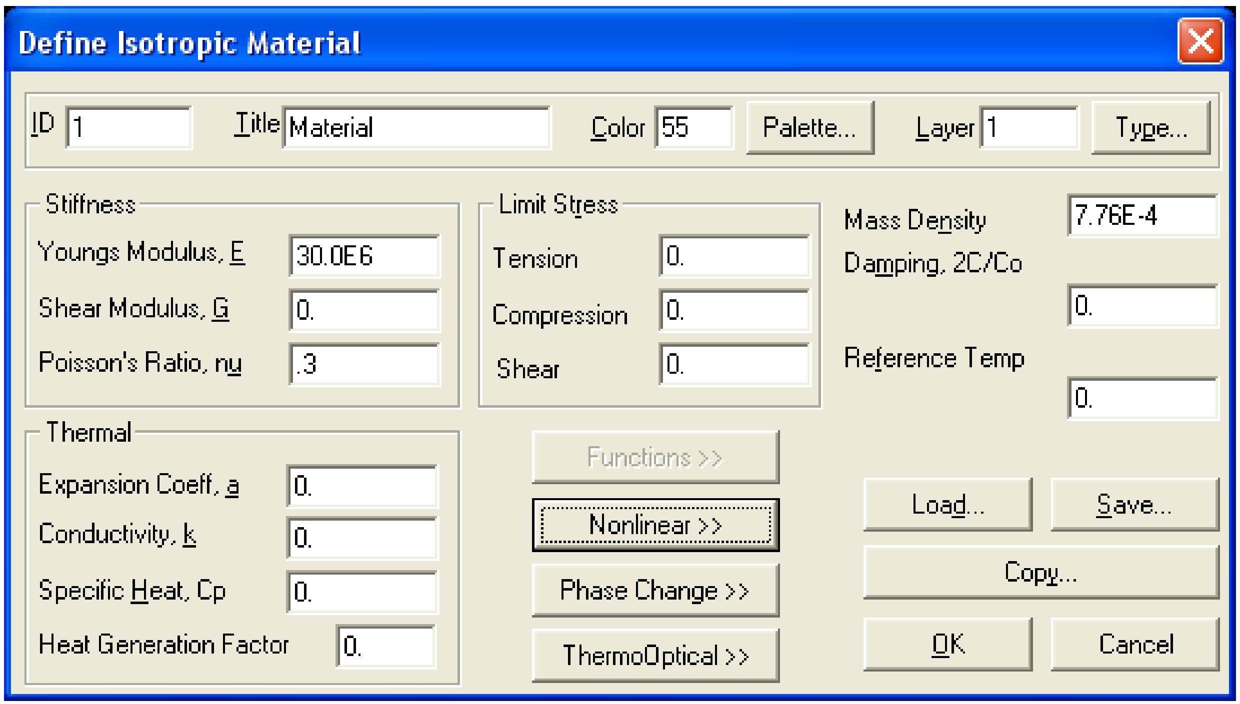
  The default value for the factor is 1.0
- Example
  - Input RHO (ρ) = 0.3 for weight density
  - Use PARAM,WTMASS,0.00259 (1/386.4) to multiple 0.3 for G = 386.4 in/sec^2
- PARAM,WTMASS is used once per run and multiples all weight/mass input (including MASSi, CONMi, and nonstructural mass input). Use all mass of all weight inputs (Do not mix!!)

Mass Units

- Material Density
  - MATi entries

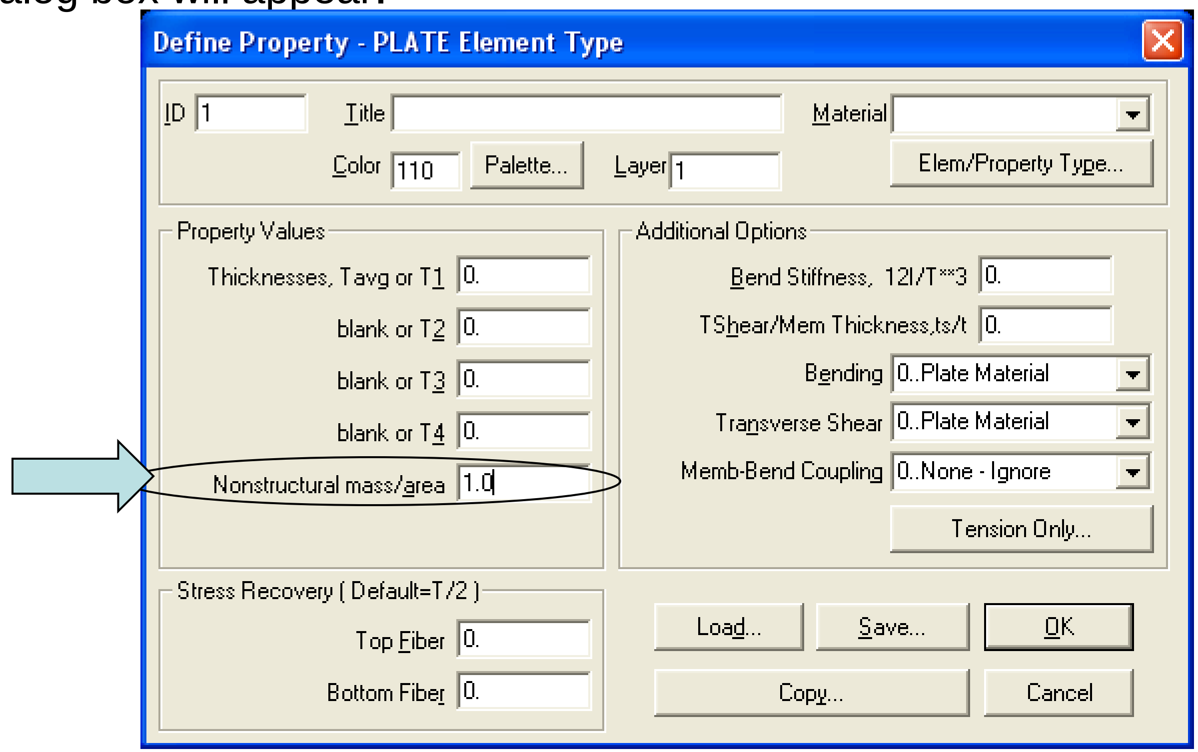
1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
MAT1	1	3.E+7		0.3	7.76E-4	0.	0.		

  - Select Model->Material and this dialog box will appear



Mass Units

- Nonstructural mass
  - Mass input on element property entry which is not associated with geometric properties of the element. Input as mass/length for line elements and mass/area for elements with 2-D geometry
  - Select Model->Property…click Elam/Property Type button. Select plate element and this dialog box will appear.



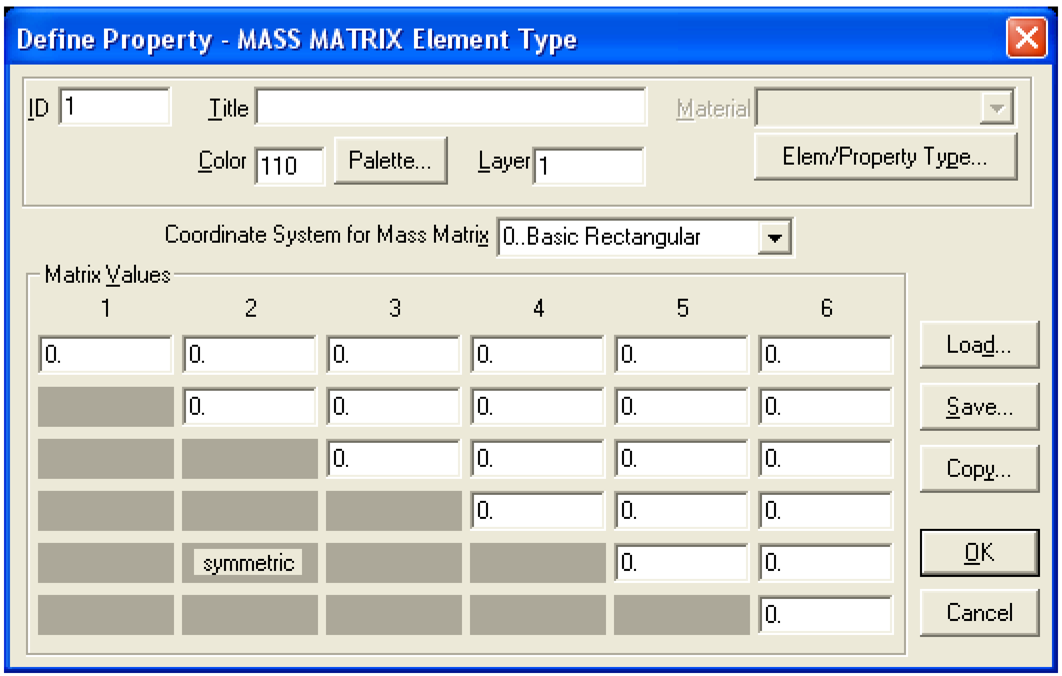
Mass Units

- Scalar Mass
  - CMASSi, PMASS

  – Grid point mass
  - CONM1 (6 x 6 mass matrix) – User defines half of the terms, symmetry is assumed.

Select Model->Property…click Elem/Property Type button.

Select Mass Matrix and this dialog box will appear

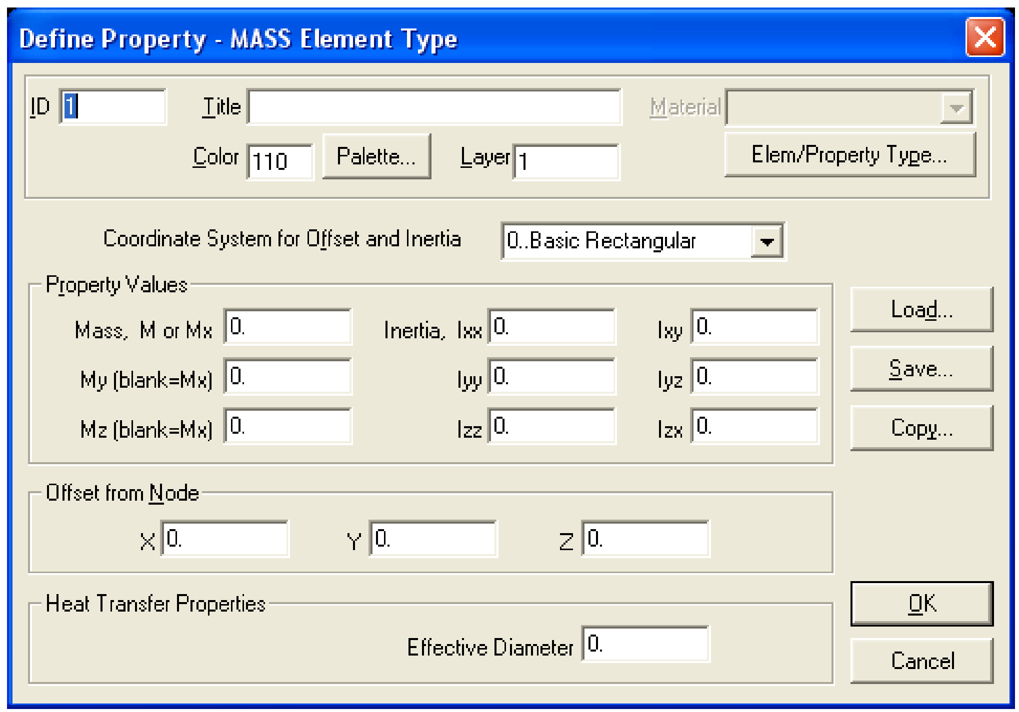


Mass Units

- CONM2 (concentrated mass)

$$\begin{bmatrix} M & & & & & \\ & M & & & \text{SYM} & \\ & & M & & & \\ & & & I11 & & \\ & & & -I21 & I22 & \\ & & & -I31 & -I32 & I33 \end{bmatrix}$$

Select Model->Property…click Elem/Property Type button. Select Mass and this dialog box will appear:



Basic NX Nastran Set Operations

- See the NX Nastran Quick Reference Guide

Grid Set (G) = N + M

M – Multipoint Constraints

Independent DOF (N) = F + S

S – Single Point Constraints

Unconstrained DOF (F) = A + O

O – Static Condensation, Guyan or CMS

Analysis Set A = L + R

R – Free-Body Partitioning

Solve A-Set Modes

Reverse Process for Data Recovery to G-Set

Tips on Model Verification

- PARAM,GRDPNT,V1 (V1 > 0)
  - Grid point weight generator
- PARAM,USETPRT, V1 (V1 = 0,1,2)
  - NX Nastran set tables
- As always, engineering judgment

Normal Modes Analysis

NX Nastran Dynamic Analysis

Natural Frequencies and Normal Modes

- Determine the dynamic characteristics of the structure

- For instance, suppose a piece of rotating machinery, such as a motor, is mounted on a structure. The running motor will produce a frequency that may be close to one of the natural frequencies of the structure. The motor's frequency may "excite" the structure and create excessive vibration.

- Sometimes static loads can be subject to dynamic amplification

- Natural frequencies and normal modes are often used as a base or a guide to subsequent dynamic analysis (transient response, response spectrum), such as what should be the appropriate Δt for integrating the equation of motion in transient analysis.

- Transient analysis can also take advantage of normal modes and natural frequencies using a technique known as modal expansion

- Can be used to guide placement of accelerometers in physical testing

Theoretical Results

- Start with this equation:

$$[M]\{\ddot{x}\} + [K]\{x\} = 0 \qquad (1)$$

- Assume a harmonic solution:

$$\{x\} = \{\Phi\}e^{i\omega t} \qquad (2)$$

(Physically, this means that all the coordinates are in synchronous motion and the system configuration does not change shape during motion, only amplitude)

- From the harmonic solution equation:

$$\{\ddot{x}\} = -\,\omega^2\{\Phi\}e^{i\omega t} \qquad (3)$$

Theoretical Results

- Through substitution of (2) and (3) into equation (1)

$$-\omega^2 [M] \{\Phi\} e^{i\omega t} + [K] \{\Phi\} e^{i\omega t} = 0$$

- That equation simplifies to:

$$([K] - \omega^2 [M]) \{\Phi\} = 0 \qquad (4)$$

- This is an Eigenvalue problem

Theoretical Results

- From equation (4) there are two possible solutions:

  If det ([K] - ω^2 [M]) ≠ 0, the only possibility, from equation (4), is

  {Φ} = 0

  which produces a so-called "trivial" solution and will not reveal anything about the behavior of the system from a physical perspective

  If the det ([K] - ω^2 [M]) = 0, there is a "non-trivial" solution for {Φ}

- The Eigenvalue problem reduces to solving:

  det ([K] - ω^2 [M]) = 0 **or** det ([K] - λ [M]) = 0

  where $\lambda = \omega^2$

Theoretical Results

- If the structure has N dynamic degrees of freedom (with mass), there are N number of ω's that are solutions to the Eigenvalue problem.
  - These ω's (ω_1, ω_2, …, ω_N) are the natural frequencies of the structure, sometimes referred to as:
    - Characteristic Frequencies
    - Fundamental Frequencies
    - Resonance Frequencies

- The eigenvector $\{\Phi_j\}$ associated with the natural frequency ω_j is called the normal mode or mode shape
  - The normal mode corresponds to deflated shape patterns of the structure

- When a structure is vibrating, its shape at any time is a linear combination of its normal modes.

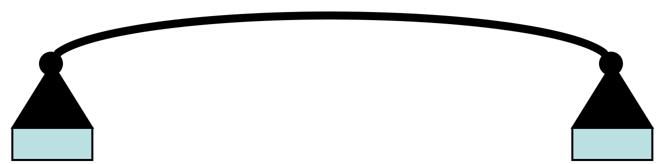
Theoretical Results

- Example

Simply Supported Beam

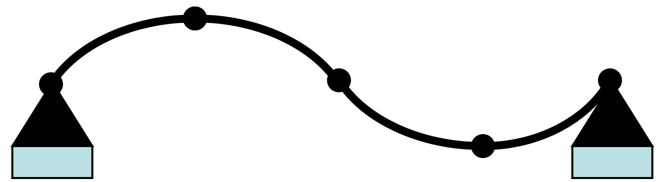
- Example plots of the first three modes:

- Mode 1



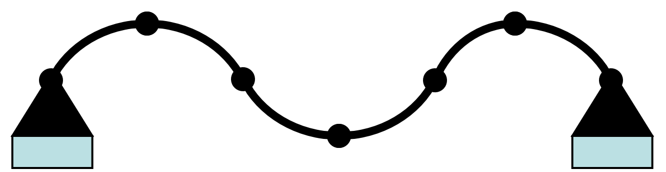
ω_1

- Mode 2



ω_2

- Mode 3



ω_3

Finite Element Analysis Predictive Engineering

Facts Regarding Normal Modes

- When [K] and [M] are symmetric and real (this is true for all the standard structural finite elements), the following orthogonality property holds:

$$\{\Phi_i\}^T[M]\{\Phi_j\} = 0 \quad \text{If } i \neq j$$

and

$$\{\Phi_i\}^T[K]\{\Phi_j\} = 0 \quad \text{If } i \neq j$$

also

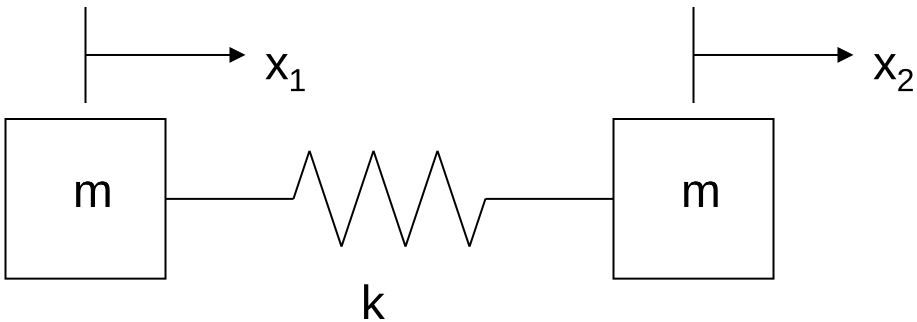
$$\omega_j^2 = \frac{\{\Phi_j\}^T[K]\{\Phi_j\}}{\{\Phi_j\}^T[M]\{\Phi_j\}}$$

- The natural frequencies (ω_1, ω_2, …) are expressed in radians/second. They can also be expressed in hertz (cycles/second) using:

$$f_j\,(\text{hertz}) = \frac{\omega_j\,(\text{radians/second})}{2\pi}$$

Facts Regarding Normal Modes

- For Example the system below is unconstrained and has a rigid-body mode

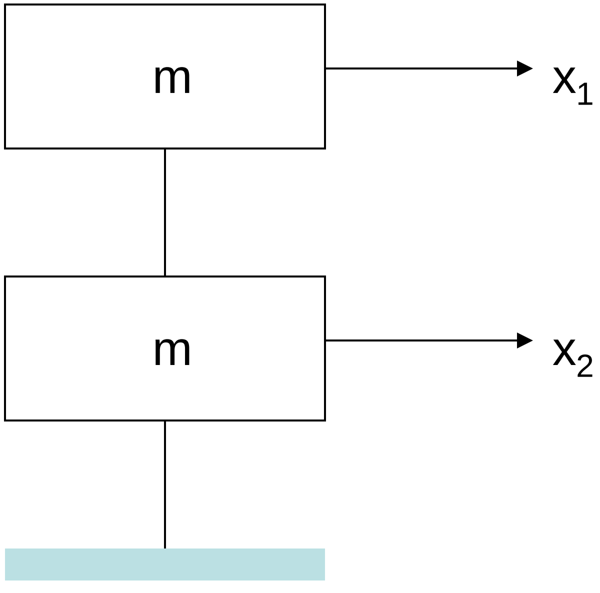


$$\omega_1 = 0 \qquad \{\Phi_1\} = \begin{Bmatrix} 1 \\ 1 \end{Bmatrix}$$

- When a structure is not fully constrained, meaning it will exhibit a "rigid-body" mode (stress free mode) or a mechanism, at least one natural frequency will be zero.

Facts Regarding Normal Modes

- Scaling of normal modes is arbitrary. For instance:



$$\{\Phi_1\} = \begin{Bmatrix} 1 \\ 0.5 \end{Bmatrix}, \{\Phi_1\} = \begin{Bmatrix} 300 \\ 150 \end{Bmatrix}, \text{ and } \{\Phi_1\} = \begin{Bmatrix} 0.66 \\ 0.33 \end{Bmatrix}$$

All represent the same mode of vibration

Facts Regarding Normal Modes

- FFor practical considerations, modes should be normalized by a chosen convention. In NX Nastran there are three normalization choices (except when using the Lanczos method)

  - TThe unit value of generalized mass (default)

$$\{\Phi_i\}^T [M]\{\Phi_i\} = 1.0$$

  - TThe unit value of the largest A-set component in each mode
  - TThe unit value of a specific component (not recommended)

- IIn Lanczos method, normalization is to a unit value of generalized mass or to a unit value of the largest component.

Additional Modal Properties

- Since strains, internal loads, and stresses develop when a structure deforms, additional useful modal information can be recovered by utilizing:
  - sStrain-displacement relationships

    $\{\varepsilon\} = [K_{\varepsilon u}]\ \{u\}$

  - SStress-strain relationships

    $\{\sigma\} = [K_{\sigma\varepsilon}]\ \{\varepsilon\}$

  - SStatic force-displacement relationships

    $\{P_{st}\} = [K]\ \{u\}$

  - EElement strain energy relationships

    $V_e = 1/2\{u_e\}T[K_{ee}]\{u_e\}$

Methods of Computation

- NX Nastran provides the user with three types of methods for eigenvalue extraction

- Tracking Methods
  - Natural Frequencies (Eigenvalues) are determined one at a time using an iterative approach
  - Two variations of the inverse power method are provided using INV and SINV
  - This approach is more convenient when a small number are to be determined
    - In general, SINV is more reliable than INV

- Transformation Methods
  - The original eigenvalue problem

$$([K] - \lambda [M])\{\Phi\} = 0$$

is transformed to the form:

$$[A]\{\Phi\} = \lambda\{\Phi\} \qquad \text{where } [A] = [M]^{-1}[K]$$

Methods of Computation

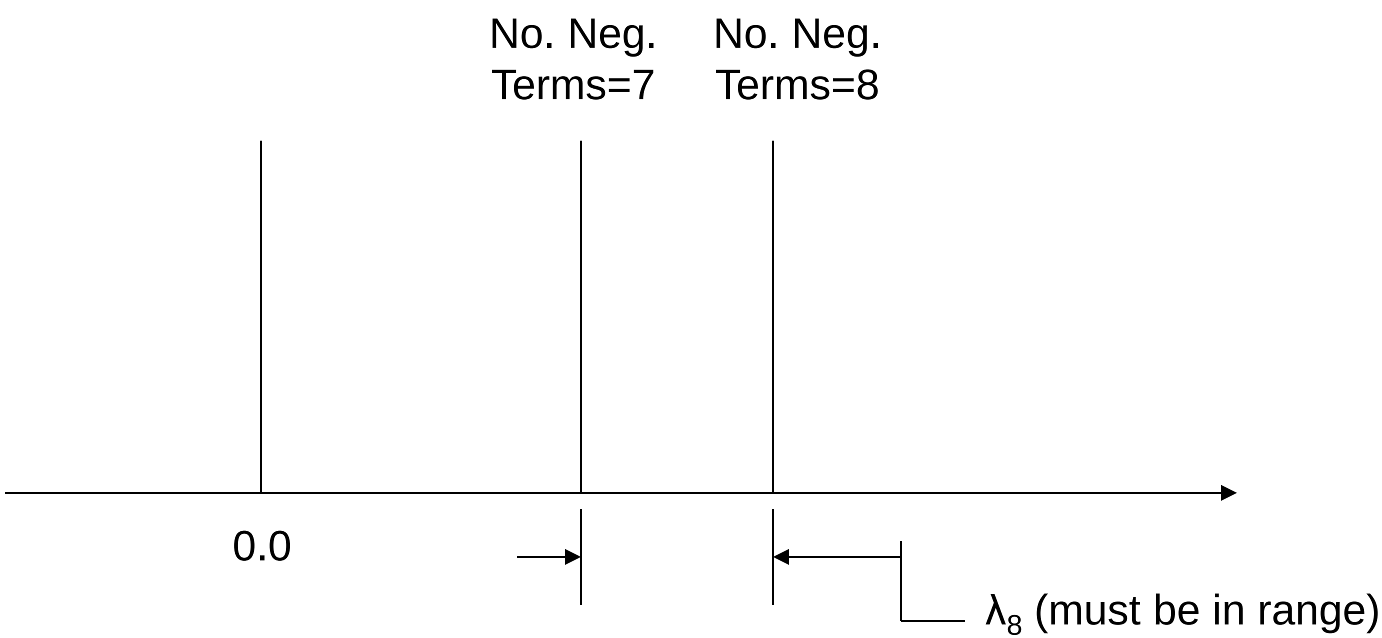
- Then, the Matrix [A] is transformed into a tri-diagonal matrix using either the Givens technique or the Householder technique
- Finally, all eigenvalues are extracted at once using the QR algorithm
- Two variations of both the Givens and the Householder methods are provided for use:
    - GIV
    - MGIV
    - HOU
    - MHOU
- These methods are more efficient when a large number of eigenvalues are needed to be extracted

- Lanczos Method

- The newest method, Lanczos is a combined tracking-transformation method
    - This method is most efficient for computing a few eigenvalues of large, sparse problems

Sturm Sequence Theory

- CChoose λ.
- FFactor [K – λ_iM] into [L][D][L^T].
- TThe number of negative terms on the factor diagonal is the number of eigenvalues below λ.

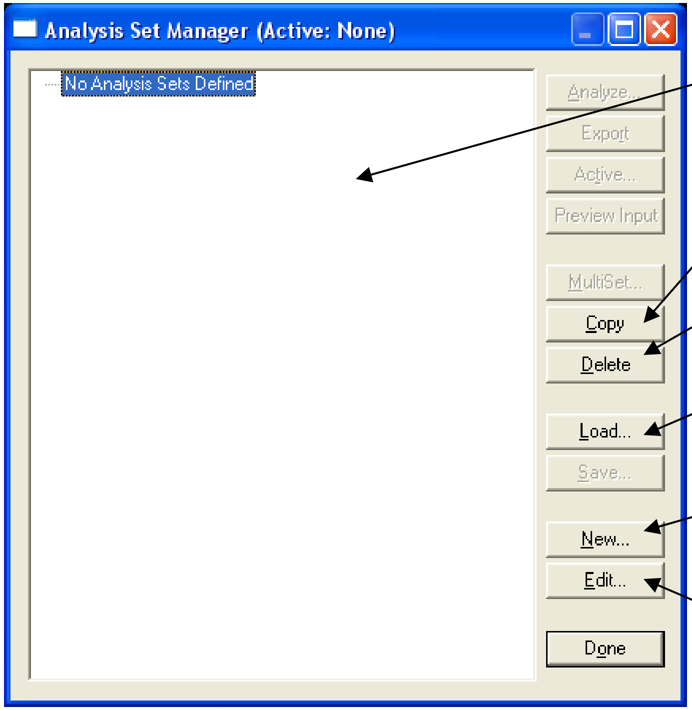


Lanczos Method

- **Block, shifted, inverted Lanczos**
- **Random starting vectors**
- **Automatic shift logic**
- **Partial and selective othogonalization**
- **Sturm sequence diagnosis**
- **Givens plus QL eigensolution**
- **Can be used for both buckling and normal modes analysis**
- **Mass and largest component normalize only**

Creating Analysis Set with Analysis Set Manager

All Analysis cases should be set-up using the Analysis Set Manager. The Analysis Set Manager is accessed using the Model->Analysis command. Note: For users of previous versions of FEMAP, the analysis set manager was added for version 8.0, it is the recommended way to create an input file.



Analysis Set Manager – Main Window

Copies Existing Case

Delete Existing Case

Loads Saved Set

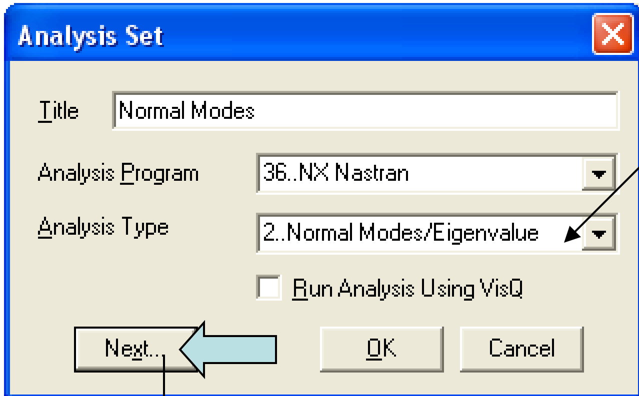
Creates New Set

Edits Existing Set Information

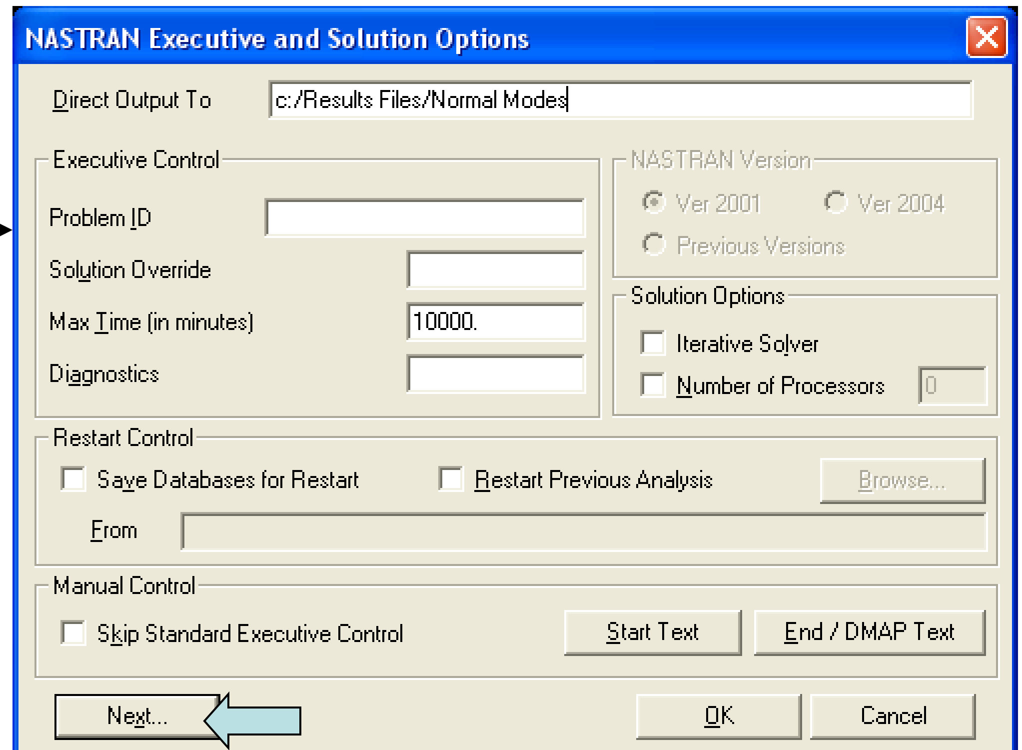
Create Analysis Set

In Analysis Set Manager dialog box, click the New button.

The Analysis Set dialog box will appear:



Select "2..Normal Modes/Eigenvalue" from the Analysis Type drop-down menu. Then click "Next" button



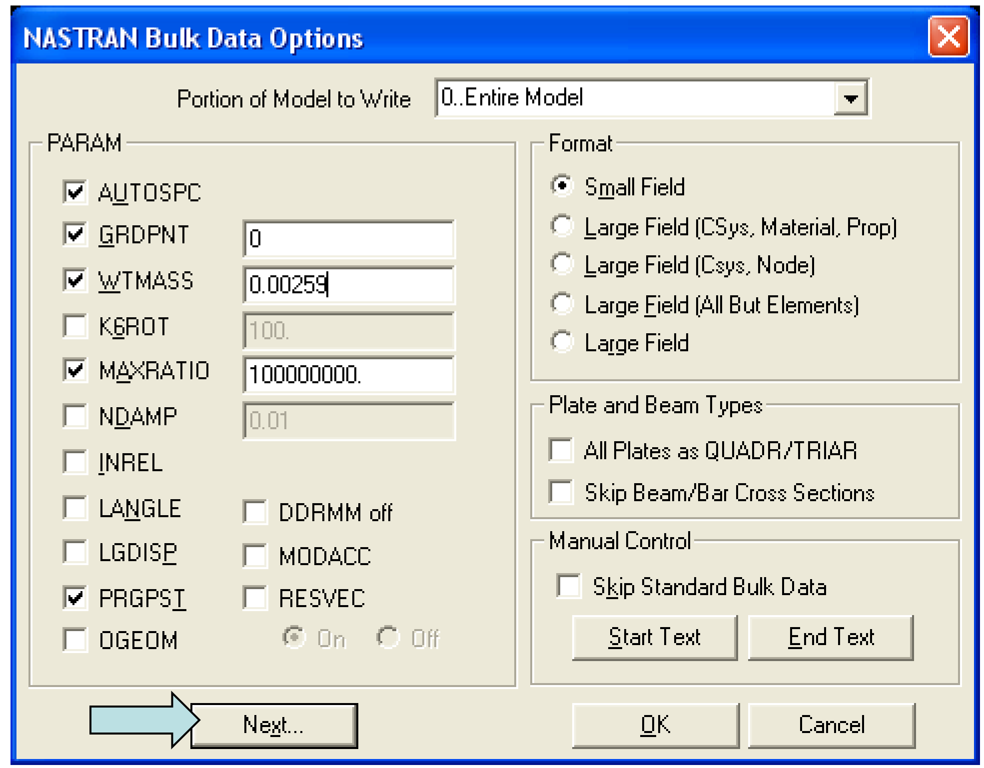
In the NASTRAN Executive and Solution Options dialog box, Executive Control options can be specified, such as problem ID, diagnostics, restarts, or output directories. Also, Solution options such as using the iterative solver or multiple processors can be selected. Click "Next" button.

Create Analysis Set

The NASTRAN Bulk Data Options dialog box will appear:

In the NASTRAN Bulk Data Options dialog box, important PARAMs for dynamic analysis can be specified such as GRDPNT and WTMASS. Also, input file formats, plate and beam element types (for instance, CQUADR for plates with "drilling" degree-of-freedom), and manual additions to the Bulk Data.

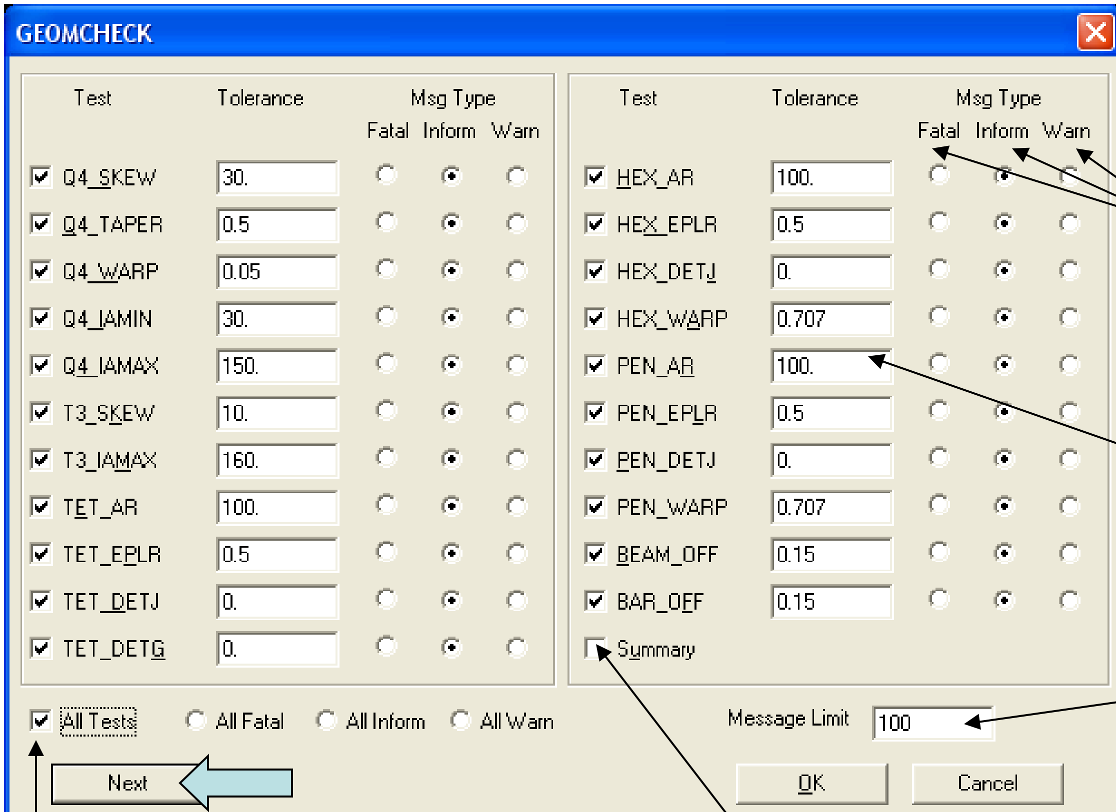
PARAMs not in the PARAM portion of the dialog box may be added to the analysis set using the "Manual Control" section of the dialog box. For example, the PARAM,USETPRT can be added to the NX Nastran analysis "deck" by clicking the "Start Text" button, and entering PARAM,USETPRT in text format.



Click the "Next" button

Creating Analysis Set

NASTRAN GEOMCHECK Diagnostic specific Dialog Box



Type of Message to alert user in .f06 file (Fatal, Warning, or Information)

Tolerance values (different for each Test, defaults listed)

Message Limit (If more than this many occur, won't be written to .f06 file)

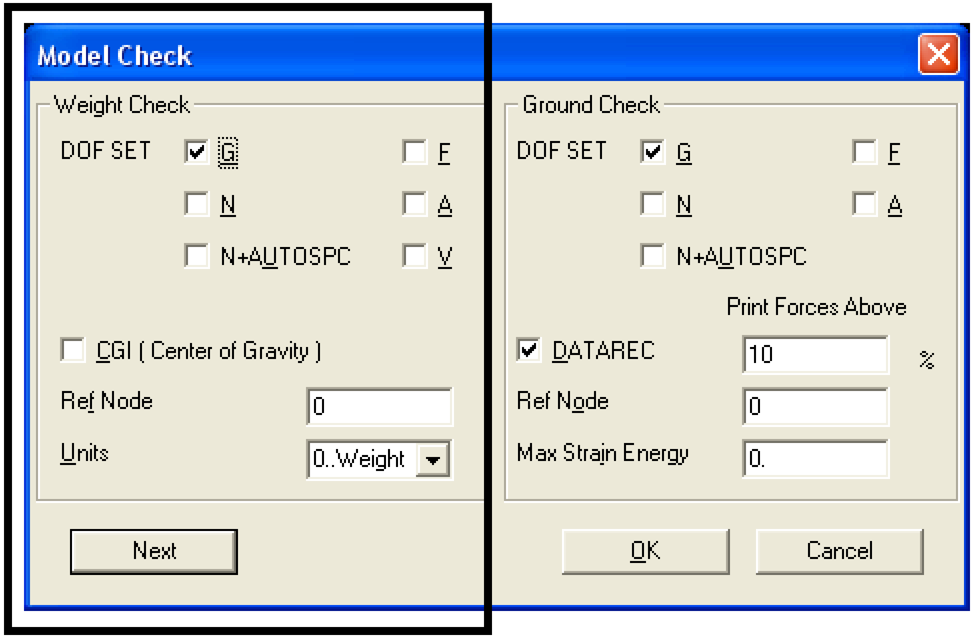
Click the "Next" button

Summary Report

All Tests on/off toggle

Special Case Control using FEMAP

NASTRAN Model Check Dialog Box (Weight Check)



WEIGHTCHECK is a Case Control command used at each stage of the mass matrix reduction, compute rigid body mass, and compare it with the rigid body mass of the G-set

The DOF sets can be chosen from the letters G, F, A, V, N, or N +AUTOSPC. For further information on DOF sets see NX Nastran Quick Reference Guide Section about DOF Set Definition.

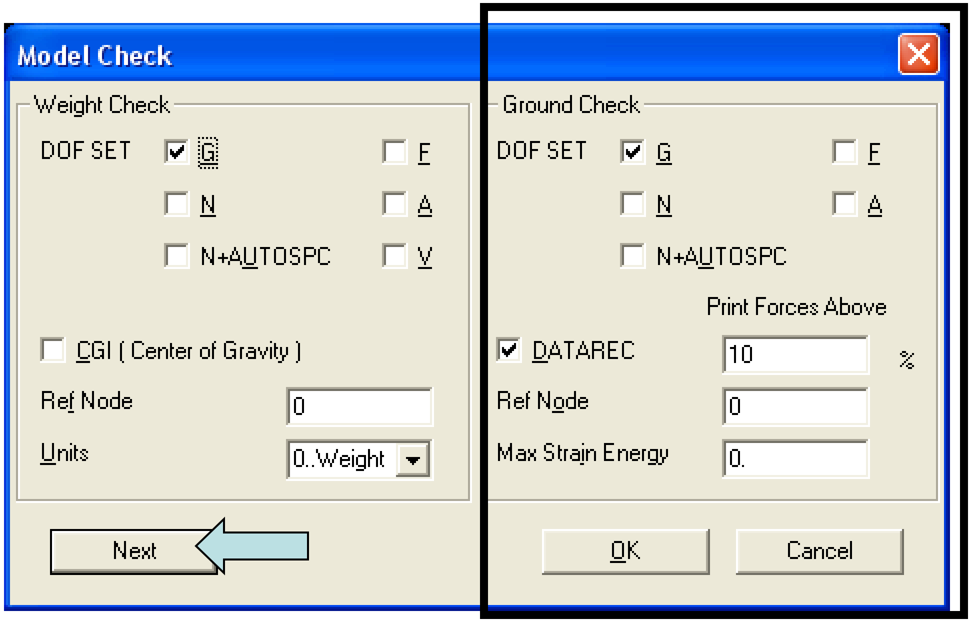
CGI requests output of center of gravity and mass moment of inertia (Default:CGI=NO)

Ref Node refers to GRID point used for calculation of the rigid body motion. (Default is the origin of the basic coordination system.)

Units selects the output in units of weight or mass (Default=WEIGHT)

Special Case Control using FEMAP

NASTRAN Model Check Dialog Box (Ground Check)



GROUNDCHECK is a Case Control command used to perform grounding check analysis on stiffness matrix to expose unintentional constraints by moving the model rigidly.

The DOF sets can be chosen from the letters G, F, N, A, or N +AUTOSPC. For further information on DOF sets see NX Nastran Quick Reference Guide Section about DOF Set Definition.

DATAREC refers to data recovery of grounding forces. (Will Print top “n”% of forces)

Ref Node refers to GRID point used for calculation of the rigid body motion.

Max Strain Energy states what the maximum strain energy that passes the “check” (Default value is largest term in the stiffness matrix divided by 1.E10)

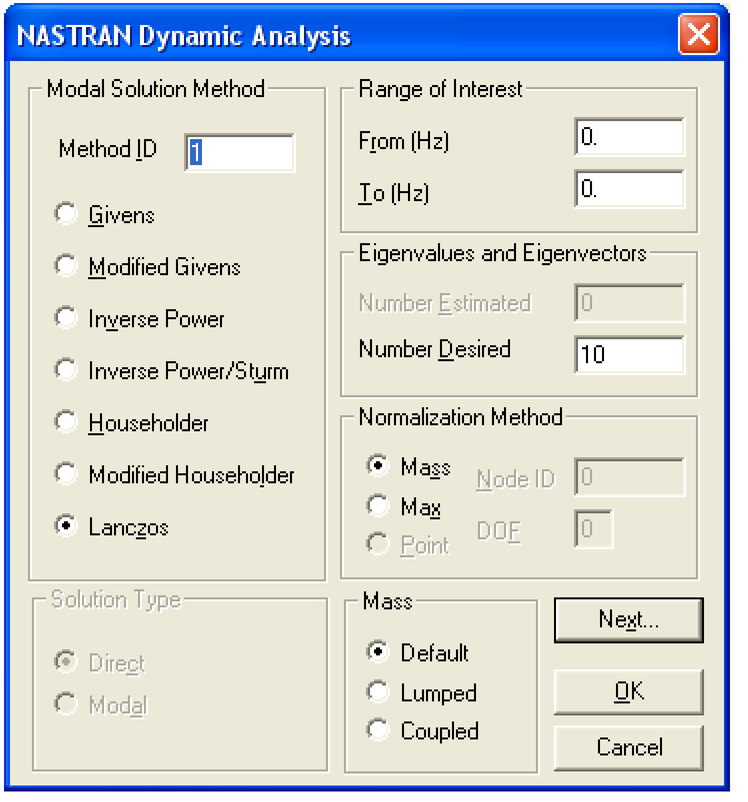
Create Analysis Set

The NASTRAN Dynamic Analysis dialog box will appear:

In this dialog box, the Modal Solution Method can be chosen.

The Frequency range of interest, the number of modes to be retrieved, the normalization method, and the type of mass (Lumped or Coupled) can be set.

Also, the Solution type can be set to either Direct or Modal for more advanced dynamic analysis.

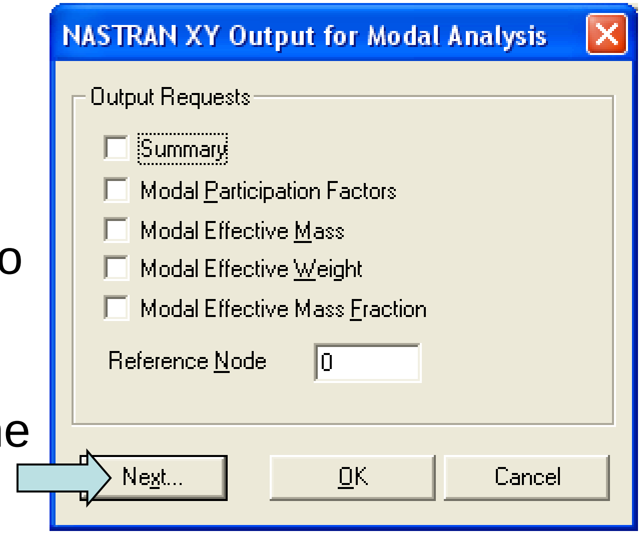


Click the “Next” button

Create Analysis Set

The NASTRAN XY Output for Modal Analysis dialog box will appear:

The NASTRAN XY Output for Modal Analysis dialog box appears if you pick the *Modal* solution type on the NASTRAN Dynamic Analysis dialog box for the following solution types: Normal Modes/Eigenvalue, Random, and Buckling. It also applies to Transient Dynamic/Time History and Frequency/Harmonic response when the system modes are calculated. This dialog box controls the type of modal participation information that is written to the PRINT output file (*.f06).



Click the "Next" button 3 times

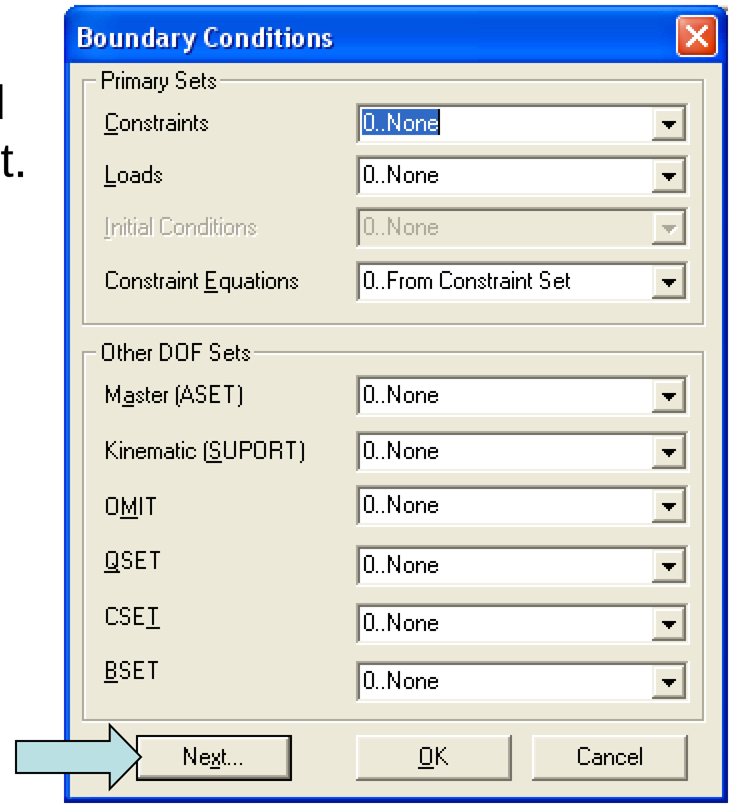
If you enter a *Reference Node,* Nastran will use it for the calculation. If you leave the value as 0, Nastran will use the origin of the global rectangular coordinate system. FEMAP will read the output information into a FEMAP function. In FEMAP, you can display this data as an XY plot.

Create Analysis Set

The Boundary Conditions dialog box will appear:

The Boundary Conditions dialog box is used to choose the Constraint and Load sets to be used in the active analysis set.

Other boundary condition sets such as initial conditions, constraint equations, and other DOF sets can be chosen in this dialog box as well.



Click the “Next” button

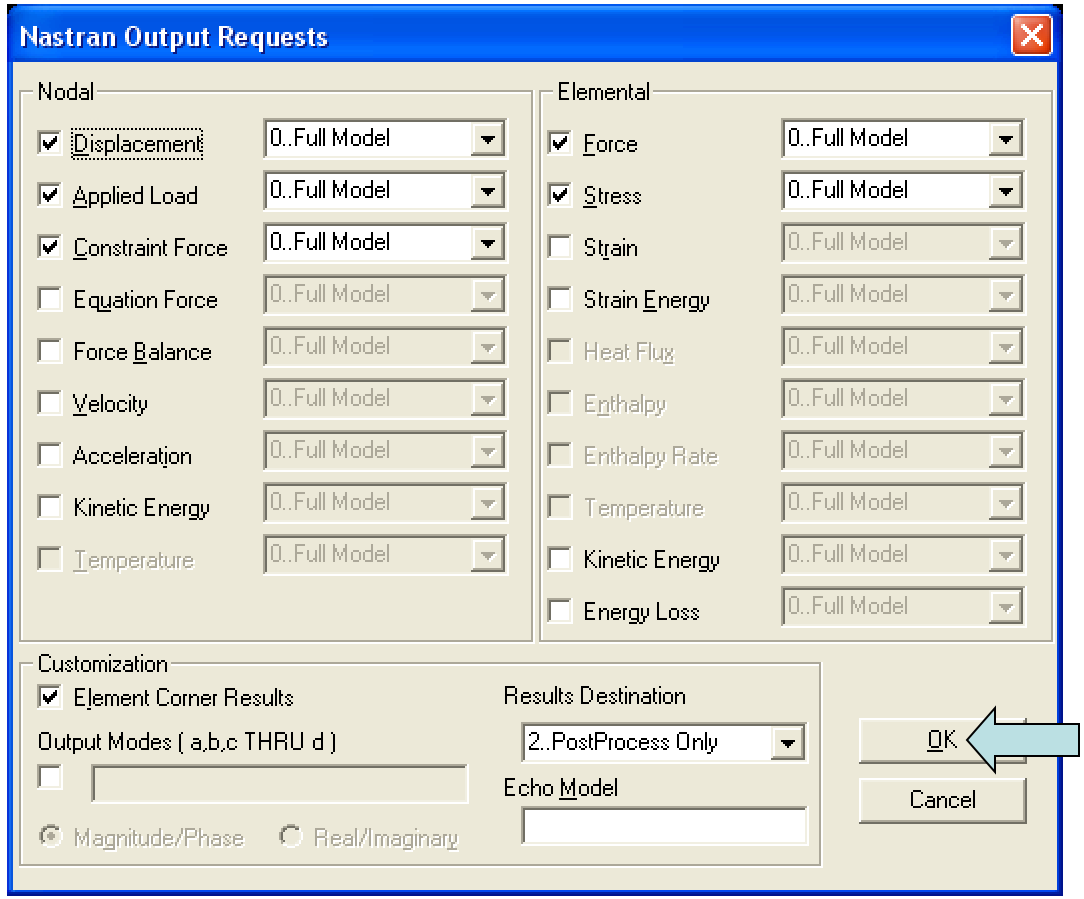
Create Analysis Set

The Nastran Output Requests dialog box will appear:

The Nastran Output Requests dialog box allows the user to select what output NX Nastran should create for post-processing.

Element corner (Guass Points) results and Output modes can be specified.

Types of results files to create (print “f06” files, post-process “op2” files, punch “pch” files, or a combination of these output files can be selected with the Results Destination drop-down menu.



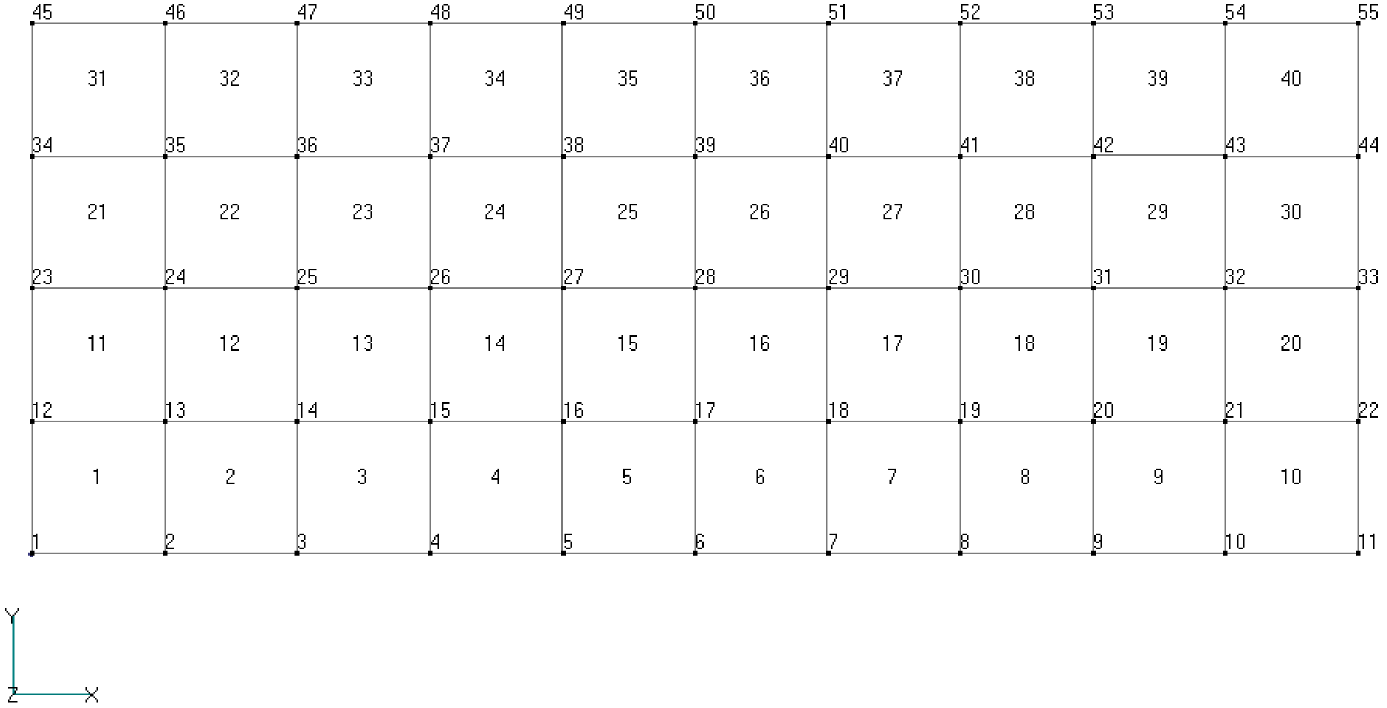
Click the “OK” button

Problem #1

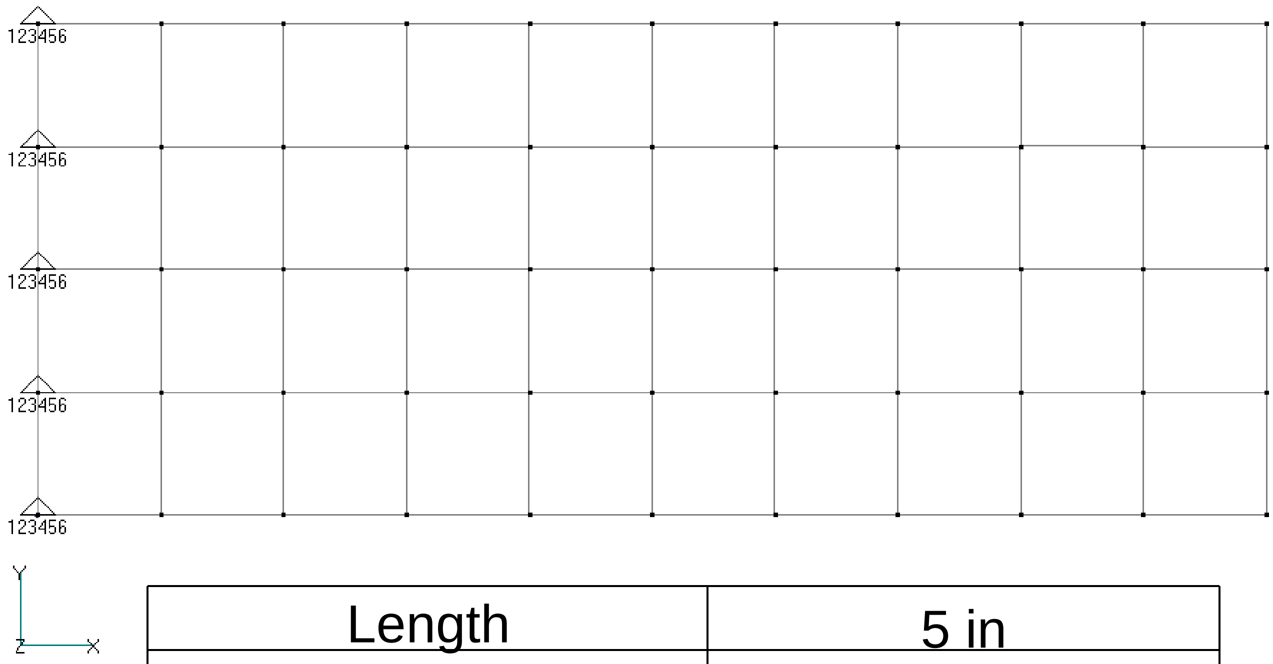
Modal Analysis of a Flat Plate

Problem #1: Modal Analysis of a Flat Plate

For this problem, use Lanczos method to find the first ten natural frequencies and mode shapes of a flat rectangular plate. Build a finite element representation of the rectangular plate. The plate will be 5 inches by 2 inches and the material and element properties are on the next page. The left side of the plate will be fixed.



Problem #1: Modal Analysis of a Flat Plate



Length	5 in
Height	2 in
Thickness	0.100 in
Weight Density	0.282 lbs/in^3
Mass/Weight Factor	2.59E-3 sec^2/in
Elastic Modulus	30.0E6 lbs.in^2
Poisson's Ratio	0.3

Problem #1: Modal Analysis of a Flat Plate

Use these results for comparison:

Mode 1	133.1684 Hz
Mode 2	648.7171 Hz
Mode 3	821.3796 Hz
Mode 4	2043.021 Hz
Mode 5	2277.875 Hz
Mode 6	2357.667 Hz
Mode 7	3704.534 Hz
Mode 8	4343.623 Hz
Mode 9	4762.875 Hz
Mode 10	5569.165 Hz

Reduction in Dynamic Analysis

NX Nastran Dynamic Analysis

Introduction to Dynamic Reduction

- Definition
  - Dynamic Reduction means reducing a given dynamic math model to one with fewer degrees of freedom.

- Why Reduction for Dynamics?
  - The model may be to large to solve without reduction.
  - The model has more detail than required.
  - Dynamic reduction is cheaper then the precise analysis of a large model. In other words, each analysis will take less time when reduction is put to use.
  - Dynamic reduction is more accurate (and will likely solve in les time) than constructing a separate, smaller dynamic model.

Reduction Methods for Dynamics Available with NX Nastran

- Guyan reduction (static condensation)
- Modal reduction

Static Condensation (Internal Calculation)

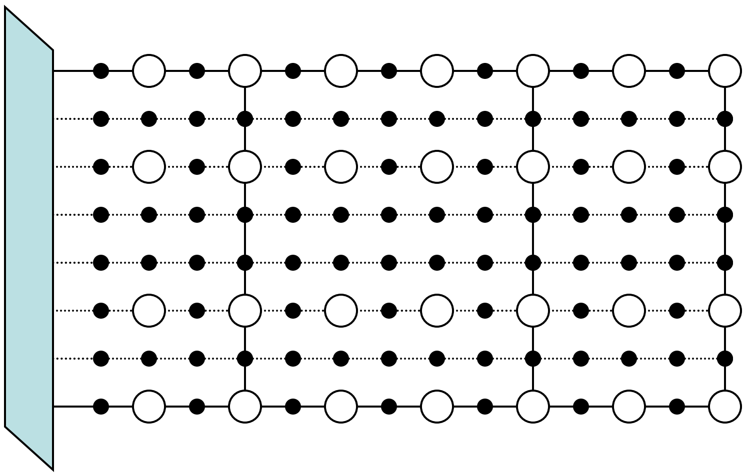
- Let $\{u_f\}$ be the set of the unconstrained (free) structural coordinates
- Partition

$$\{u_f\} = \frac{\{u_o\}}{\{u_a\}}$$

where

u_a = analysis set

u_o = omitted set



● Degrees of freedom removed during

○ User-Selected dynamic

Finite Element Analysis Predictive Engineering

Static Condensation (Internal Calculation)

- Form a static equation for u_f and partition the stiffness matrix into the O-set and the A-set.

$$\left[\begin{array}{c:c} K_{oo} & K_{oa} \\ \hdashline K_{oa}T & \overline{K}_{aa} \end{array}\right] \left\{\begin{array}{c} u_o \\ \hdashline u_a \end{array}\right\} = \left\{\begin{array}{c} P_o \\ \hdashline P_a \end{array}\right\}$$

- Assume P_o is zero and solve for u_o in terms of u_a

$$\{u_o\} = [G_{oa}]\ \{u_a\}$$

$$[G_{oa}] = -[K_{oo}]^{-1}\ [K_{oa}]$$

Static Condensation (Internal Calculation)

- Transformation from the A-set to F-set is:

$$\{u_f\} = \left\{ \begin{array}{c} u_o \\ \hdashline u_a \end{array} \right\} = \underbrace{\left\{ \begin{array}{c} G_{oa} \\ \hdashline I \end{array} \right\}}_{\Psi} \{u_a\}$$

- O-set is dependent upon the A-set. The motion of the O-set is a linear combination of the A-set motions. The columns of G_{oa} are the static shape vectors
- The equations of motion for the F-set are written in terms of the A-set

$$\Psi^T M_f \Psi\{\ddot{u}_a\} + \Psi^T B_f \Psi\{\dot{u}_a\} + \Psi^T K_f \Psi\{u_a\} = \Psi^T P_f$$

or

$$M_{aa}\ddot{u}_a + B_{aa}\dot{u}_a + K_{aa}u_a = P_a$$

Static Condensation (Internal Calculation)

- Dynamics problems are solved in terms of the reduced coordinates (A-set). O-set components are recovered.
- O-set mass, damping and stiffness is spread to the A-set.
- The largest cost (increased run time) is associated with the formulation of M_{aa} and B_{aa}, particularly for nondiagonal (coupled mass) M_{ff}.
- The resulting K_{aa}, B_{aa}, and M_{aa} are small and dense (i.e. matrix bandedness is destroyed).

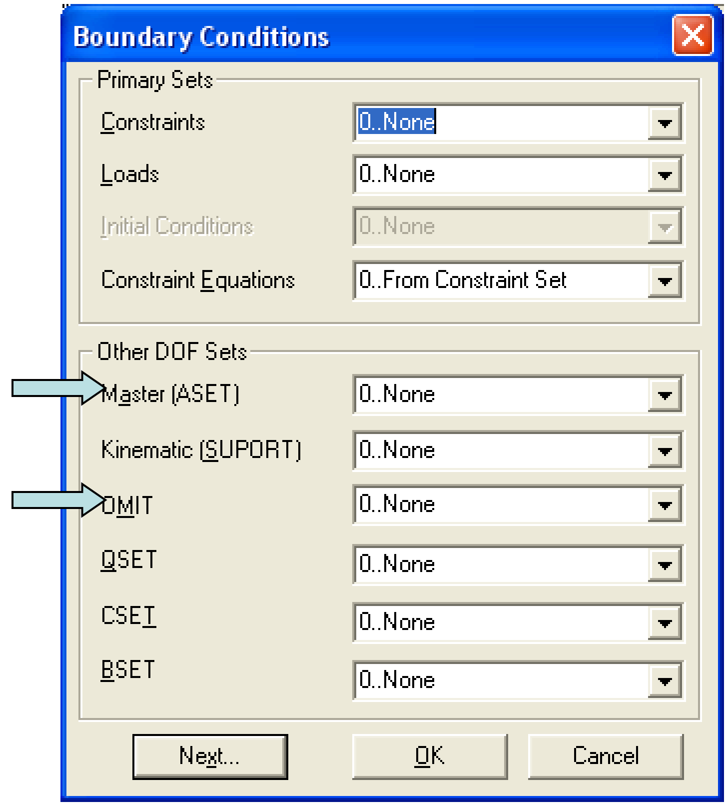
Static Condensation (Internal Calculation)

Summary

- Separate free degrees of freedom (u_f) into the omitted set (u_o) and the analysis set (u_a) by means of OMIT entries or ASET entries.
- Retain only a small fraction of the DOFs (typically 10% or less) in the analysis set because the computer costs for static condensation increase rapidly with the size of the analysis set. Otherwise, retain all of the DOFs.
- Retain DOFs with large concentrated masses in the analysis set.
- Retain DOFs that are loaded (in transient and frequency response analysis)
- Retain DOFs to adequately describe deflected shape or modes of interest.

Solution Control for Guyan Reduction

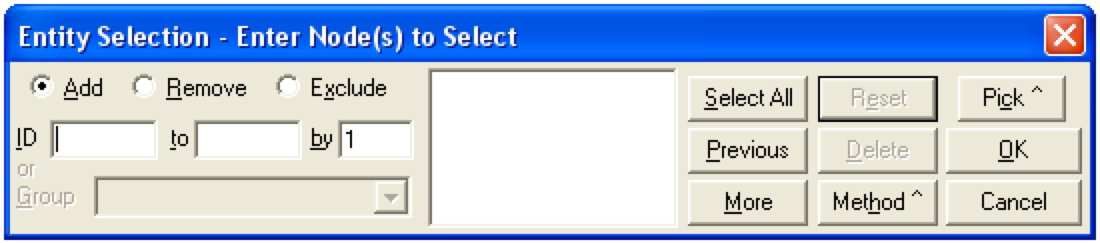
- Executive Control Section
  - Any SOL
- Case Control Section
  - No special commands required
- Bulk Data section
  - ASET (optional* - specifies A-set)
  - OMIT (optional* - specifies O-set)



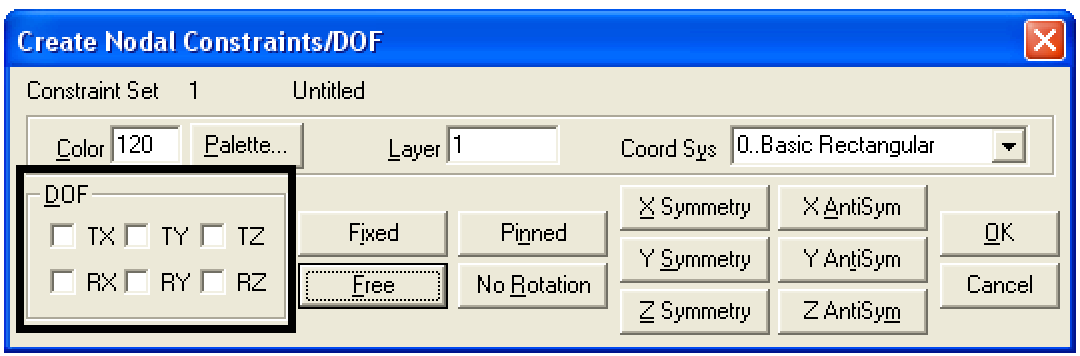
Using the Analysis Set Manager in FEMAP, the A-Set or O-set can be selected as boundary condition sets in the lower portion of the Boundary Conditions dialog box.

Solution Control for Guyan Reduction

- Creating an A-Set:
- First, create a new constraint set using the Model->Constraint->set command.
- Next, use the Model->Constraint->Nodal command…



- Choose the nodes wanted for the A-Set, then Click OK.
- Now, choose which DOFs for the nodes should be in the A-Set.



- Finally, click OK, then Cancel.

Difficulties with Guyan Reduction

- User effort in selecting A-Set points
- Accuracy depends on user’s skill in selecting A-Set points
- Regardless of user’s skill, high accuracy requires a large number of A-Set points (increases run-time) – 2 to 5 times the number of modes desired
- Stiffness reduction is exact; mass and damping reductions are only approximations
- No loss in accuracy of modes occurs when omitting mass-less degrees of freedom
- Errors are most pronounced at higher modes
- Local modes may be missed altogether
- Not generally recommended, except when performing test-analysis correlation

Difficulties with Guyan Reduction

- The static condensation approximation may miss the local dynamic effects

$$\{u_o\} = [G_{oa}]\,\{u_a\} + \cancelto{0}{\{u_o^o\}}$$

Local Dynamic Effect

Physical Variables

Static Transformation

$$\{u_o^o\} = [K_{oo}^{-1}]\,\{P_o\}$$

Loads on O-set Components

FINITE ELEMENT ANALYSIS

Predictive Engineering

Modal Reduction

- All NX Nastran linear dynamic solutions have two versions:
  - Direct – The solution is solved in terms of the A-Set
  - Modal – The solution is solved in terms of modal coordinates (H-set)

- In the modal solution sequences the A-Set coordinates are written in terms of modal coordinates

$$\{u_a\} = [\Phi_a]\,\{\,\xi\,\}$$

Modal Coordinates

Matrix of Mode Shapes

- Modal vectors (mode shapes) are solutions to the undamped eigenvalue problem (A-Set coordinates)

$$[M_{aa}]\,\{\ddot{u}_a\} + [K_{aa}]\,\{u_a\} = 0$$

Modal Reduction

- Equations of motion for the A-Set are written in terms of modal coordinates (H-set notation, modal coordinates are handled internally) Note: E-Set DOFs are not shown here for clarity.

$$[\Phi_a]^T[M_{aa}][\Phi_a][\xi] + [\Phi_a]^T[B_{aa}][\Phi_a][\xi] + [\Phi_a]^T[K_{aa}][\Phi_a][\xi] = [\Phi_a]^T[P_{aa}]$$

If $[\Phi]$ is mass normalized and there are no K2PP, M2PP, B2PP, or TF, then:

$$[\, I \,]\{\xi\} + [\Phi]^T[B_{aa}][\Phi_a]\{\xi\} + [\omega_i^2][\xi] = [\Phi]^T[P_a]$$

Note: A-Set matrices may be reduced matrices from Guyan Reduction or GDR.

Transformation from model coordinates to the F-Set would require two transformations.

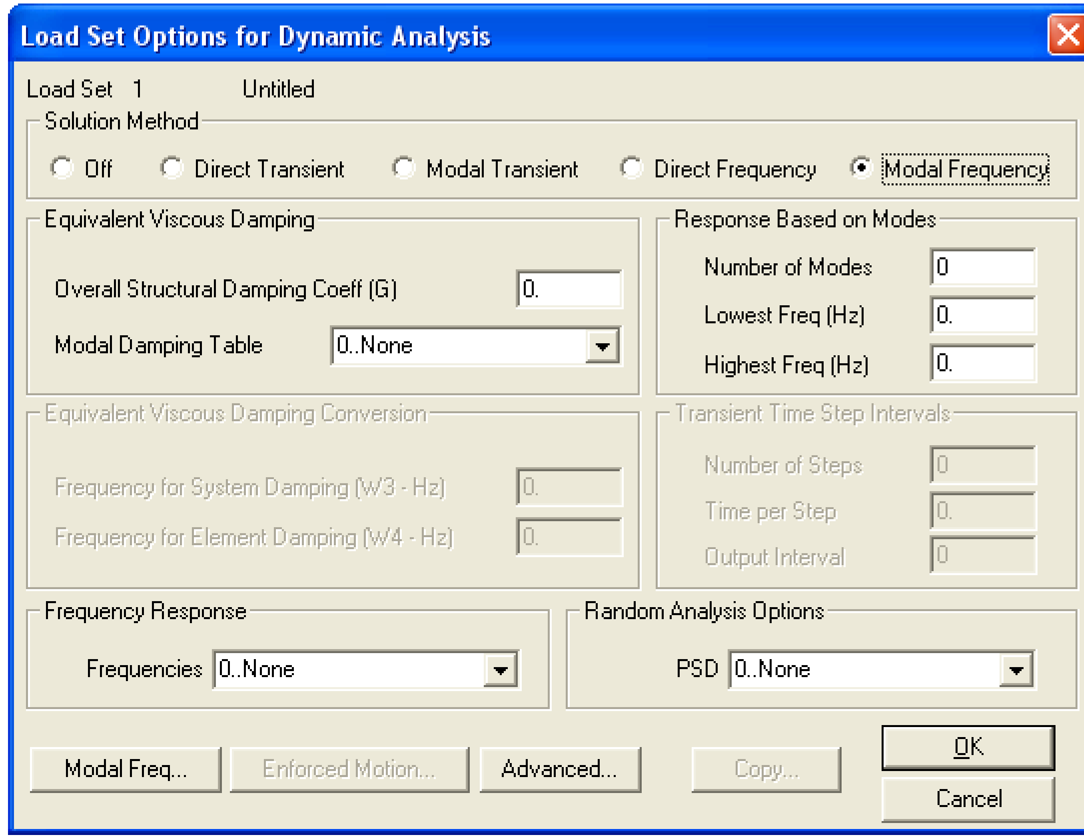
$$\{u_f\} = [\Psi]\,\{u_a\}$$

$$\{u_a\} = [\Phi_a]\,\{\xi\}$$

$$\{u_f\} = [\Psi]\,\{\Phi_a\}\,\{\xi\}$$

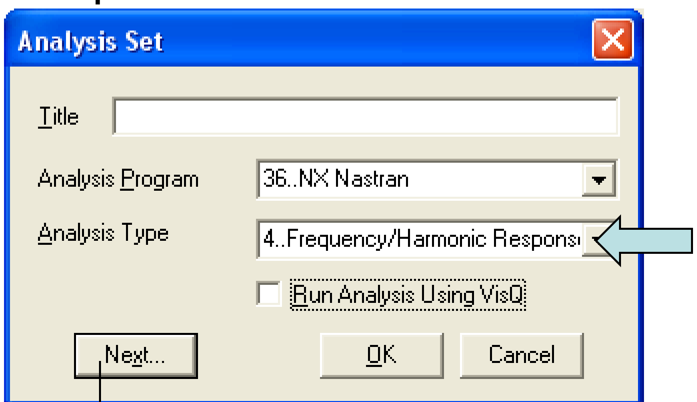
Solution Control for Modal Reduction

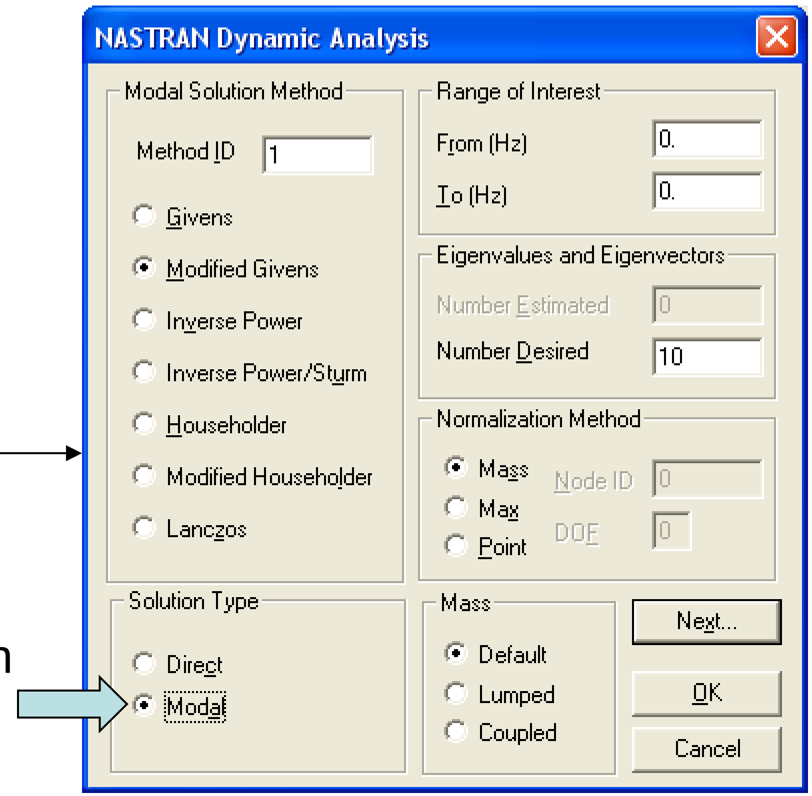
- Executive Control Section
  - •Any modal dynamic analysis SOL
- Case Control Section
  - METHOD (required – selects Bulk Data EIGR or EIGRL entry)
- Bulk Data section
  - EIGR or EIGRL (required – selects parameters for eigenanalysis)



Solution Control for Modal Reduction

- Using the Analysis Set Manager:
  - Choose “4..Frequency/Harmonic Response” from the Analysis Type drop-down menu





Click the Next button ***5*** times until the NASTRAN Dynamic Analysis dialog box appears. Click Modal in the Solution Type portion of the dialog box. Then click OK.

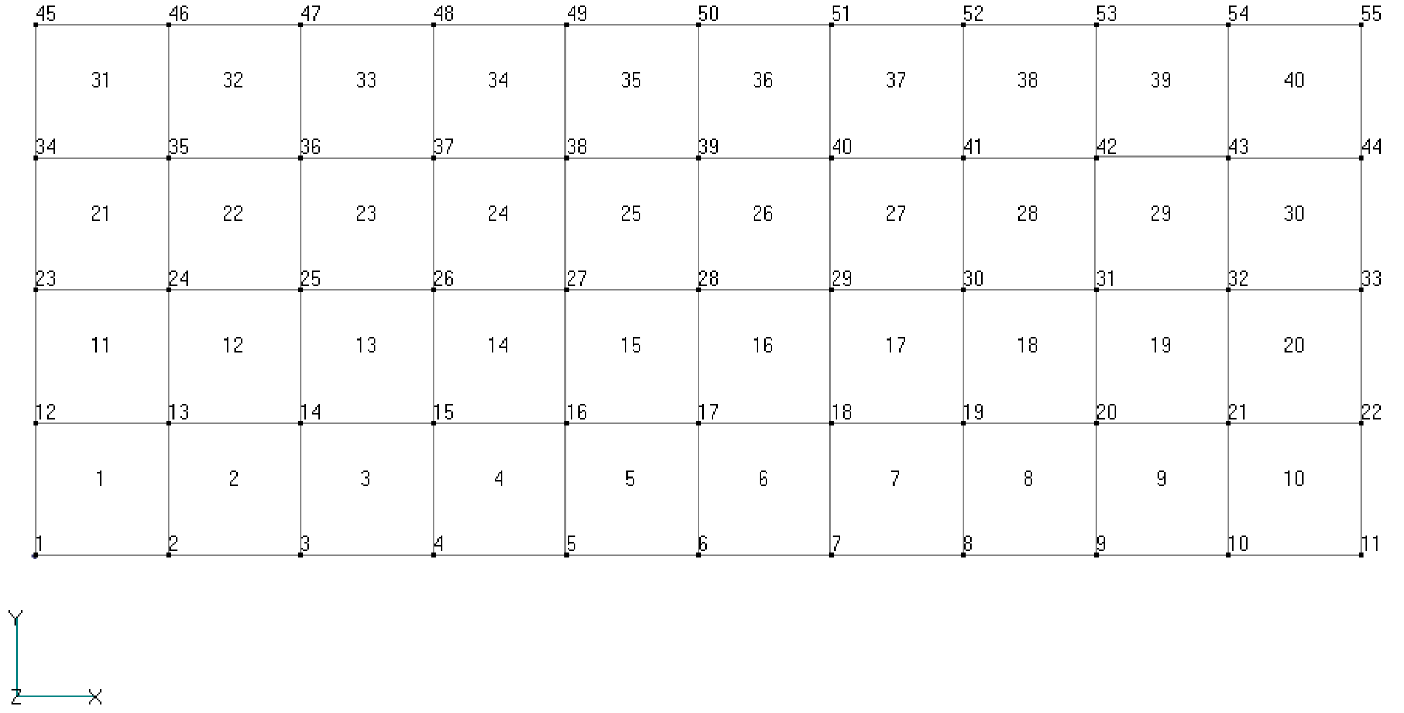
FINITE ELEMENT ANALYSIS Predictive Engineering

Problem #2

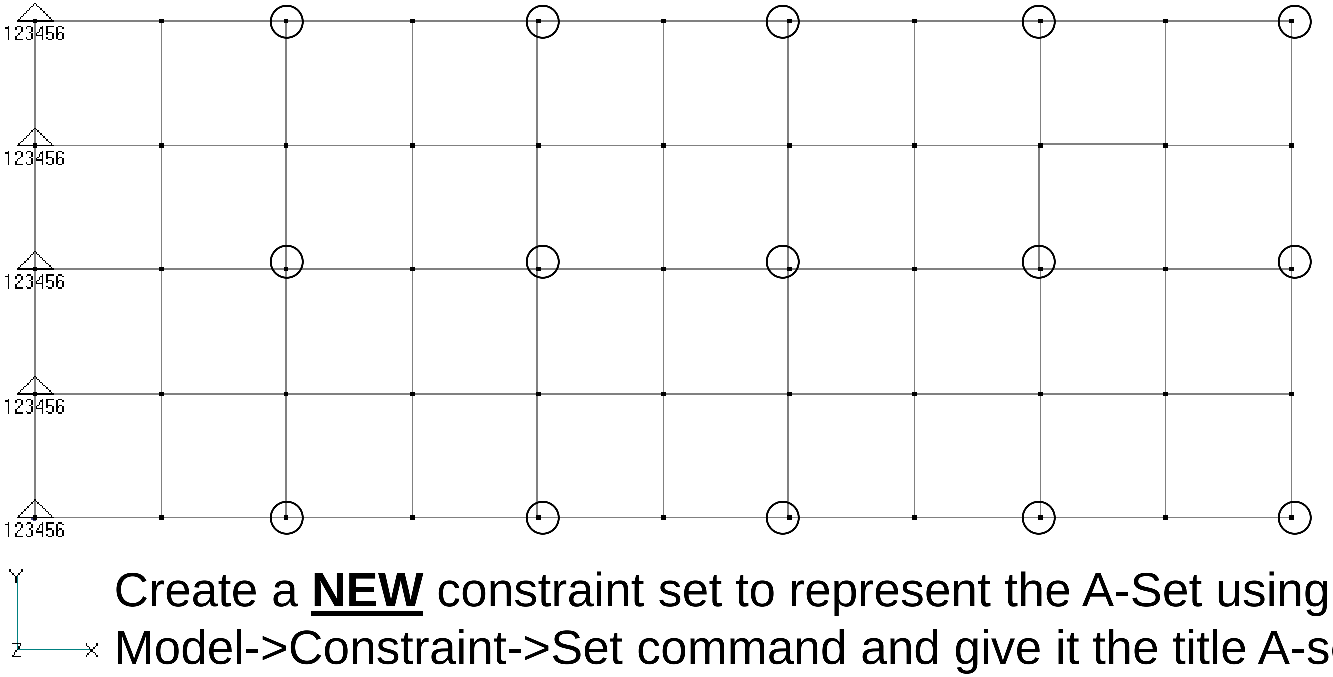
Normal Modes Analysis using
Guyan Reduction

Problem #2: Normal Modes Analysis of a Flat plate using Static Reduction

For this problem, use Guyan Reduction to reduce the model used in Problem #1. Then find the first five natural frequencies and mode shapes using the Automatic Givens method. Use the nodes indicated on the next page for the A-Set.



Problem #2: Modal Analysis of a Flat Plate Using Static Reduction



Create a **NEW** constraint set to represent the A-Set using the Model->Constraint->Set command and give it the title A-set to avoid confusion. Then use the Model->Constraint->Nodal command and choose the nodes that are circled in the picture above (Nodes:3,5,7,9,11,25,27,29,31,33,47,49,51,53,55) to include in the A-set. After the nodes are chosen, click OK and in the Create Nodal Constraints/DOF dialog box click the Fixed button to include all six DOFs for each node that was selected.

Problem #2: Modal Analysis of a Flat Plate

Use these results for comparison:

Mode 1	133.1727 Hz
Mode 2	649.0864 Hz
Mode 3	822.2924 Hz
Mode 4	2054.124 Hz
Mode 5	2295.341 Hz
Mode 6	2360.133 Hz
Mode 7	3764.084 Hz
Mode 8	4432.393 Hz
Mode 9	4836.664 Hz
Mode 10	5560.032 Hz

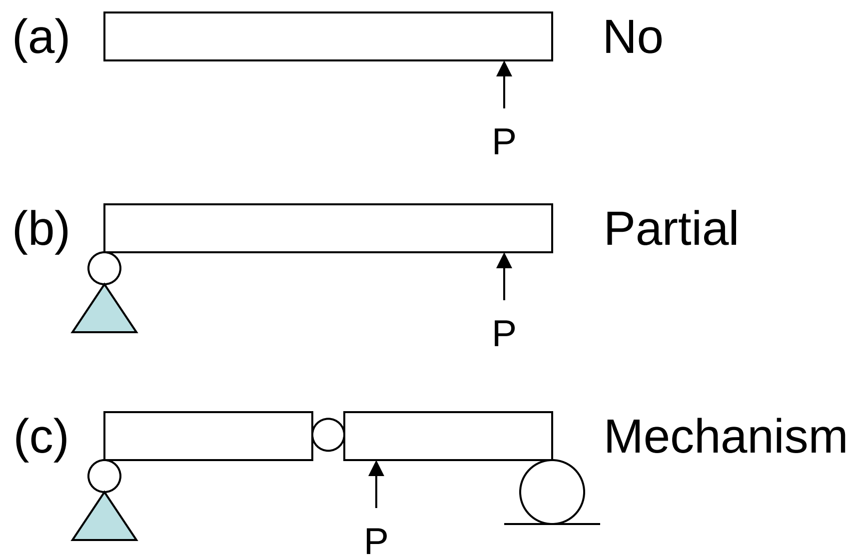
Rigid Body Modes

NX Nastran Dynamic Analysis

Rigid Body Modes

- A structure has the ability to displace without developing internal loads or stresses if it is not sufficiently grounded.

  Examples:



- In cases (a) and (b), the structure can displace as a rigid body.

Rigid Body Modes

- The presence of rigid body and/or mechanism modes is evidenced by zero frequency values in the solution of the eigenvalue problem.

$$[K]\{\Phi\} = [M]\{\Phi\}\lambda$$

- On the assumption that the mass matrix [M] is positive definite, zero eigenvalues result from a positive semi-definite stiffness, i.e.:

$$\{\Phi\}_{RIG}{}^{T}[M]\{\Phi_{RIG}\} > 0$$

$$\{\Phi\}_{RIG}{}^{T}[K]\{\Phi_{RIG}\} = 0$$

- SUPORT does not constrain the structure. It simply defines the R-Set components. In normal modes analysis, rigid body modes are calculated using the R-Set as reference degrees of freedom.

Calculation of Rigid Body Modes

- If R-Set is present, rigid body modes are calculated in NX Nastran by the following method:

Step 1: "a"-set partitioning

$$\{u_a\} = \left[\frac{u_l}{u_r}\right]$$

Step 2: Solve for u_l in terms of u_r.

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

Note: Pr is not actually applied!!!

$$\{u_l\} = [D_m]\,\{u_r\}$$

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

$$[\Psi_{RIG}] = \begin{bmatrix} D_m \\ I_r \end{bmatrix}$$

Calculation of Rigid Body Modes

Step 3: Mass matrix operations

$$[M_r] = \begin{bmatrix} D_m \\ I_r \end{bmatrix}^{\mathbf{T}} [M_{aa}] \begin{bmatrix} D_m \\ I_r \end{bmatrix}$$

where $[M_r]$ is not diagonal in general

- Using Gram-Schmidt orthogonalization (in the READ module), the matrix $[M_r]$ is orthogonalized by the transformation $[\Phi_{ro}]$, that is:

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

Calculation of Rigid Body Modes

Step 4: Rigid body mode construction

$$[\Phi_a]_{RIG} = \begin{bmatrix} D_m \Phi_{ro} \\ \Phi_{ro} \end{bmatrix}$$

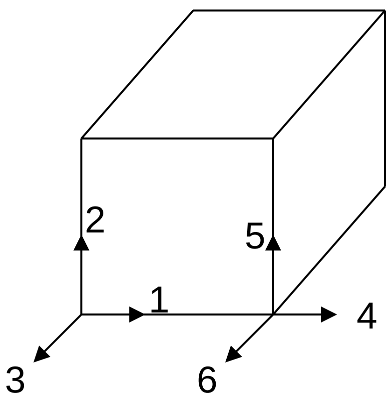
with the property:

$$[\Phi_a]_{RIG}{}^T [K_{aa}][\Phi_a]_{RIG} = K_{rr} \approx 0$$

$$[\Phi_a]_{RIG}{}^T [M_{aa}][\Phi_a]_{RIG} = [M_o]$$

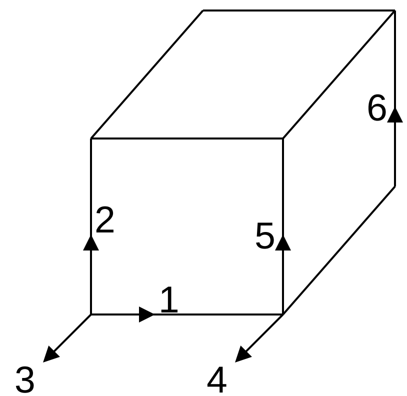
Selection of “SUPORT” Degrees of Freedom

- Care must be taken when selecting SUPORT DOFs.
- SUPORT DOFs must be able to displace independently without developing internal stresses (statically determinate)



Bad Selection for SUPORT

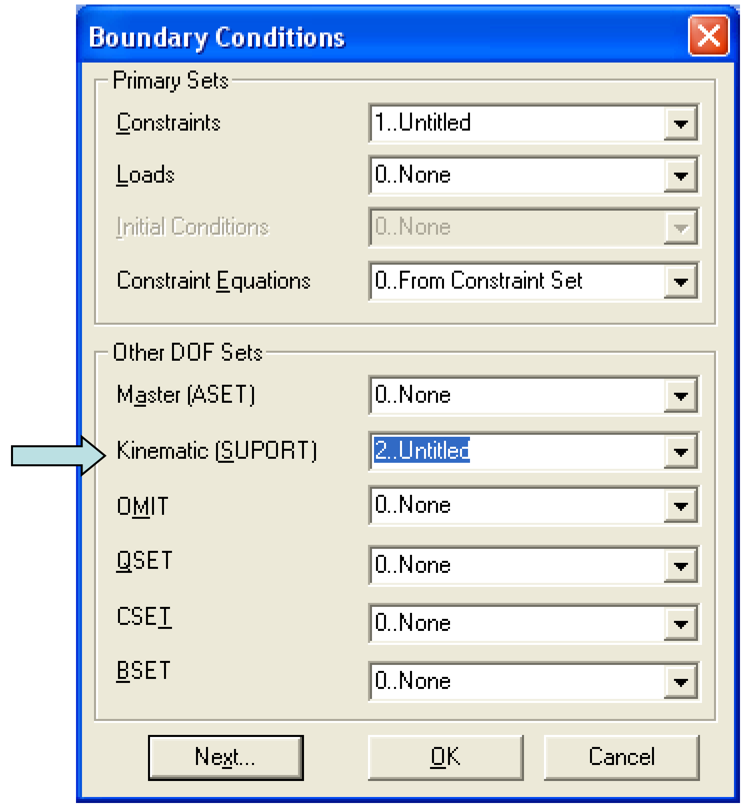
(The independent displacement of 1 and 4 may produce internal stress)



Good Selection for SUPORT

Selection of "SUPORT" Degrees of Freedom

- In NX Nastran for FEMAP, the "SUPORT" degrees of freedom can be entered by creating a constraint set with the desired nodes and degrees of freedom (much like the way the A-set was created for Guyan Reduction in the last section), then selecting that constraint set from the Kinematic (SUPORT) drop down menu in the Boundary Conditions dialog box of the Analysis Set Manager



Finite Element Analysis
Predictive Engineering

Checking of "SUPORT" Degrees of Freedom

- NX Nastran calculates internal strain-energy (work) for each rigid body vector.

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

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

Rigid Body Vectors

Strain Energy Matrix, Diagonals Printed

- If actual rigid body modes exist, the strain-energy is ≈ 0
- Note that [X] is also the transformation of the stiffness matrix $[K_{aa}]$ to R-Set coordinates, which by definition of rigid body (zero frequency) vector properties, should be null.

Checking of "SUPORT" Degrees of Freedom

- NX Nastran also calculates the rigid body error ratio

$$\varepsilon = \frac{\|[X]\|}{\|K_{rr}\|}$$

Where $\| \;\; \|$ means Euclidian norm of the matrix

$$\| \;\; \| = \sqrt{\sum_i \sum_j x_{ij}^2}$$

- Only one value of ε is calculated using [X] and [K_{rr}] based on all SUPORT DOFs

Checking of "SUPORT" Degrees of Freedom

- Except for round-off errors, the rigid body error ratio and the strain energy should be zero if a compatible set of statically determinate supports are chose by the user. The quantities may be non-zero for any of the following reasons:
  - Round-off error accumulation
  - The u_r set is over-determined leading to redundant supports (high energy strain).
  - The u_r set is underspecified leading to a singular reduced stiffness matrix (high rigid body error ratio).
  - The multipoint constraints are incompatible (high strain energy and high rigid body error ratio)
  - There are too many single point constraints (high strain energy and high rigid body error ratio)
  - K_{rr} is null (unit value for rigid body error but low strain energy). This is an acceptable condition and may occur when generalized dynamic reduction is used.

Rigid Body Modes

- In NX Nastran, flexible body modes associated with the A-Set mass and stiffness matrices are calculated. The first N modes calculated by the eigenanalysis (where N is the number of DOFs in the R-Set) are discarded. The N rigid body modes are substituted in their place.

$$\{u_a\} = [\Phi_{aFLEX}] = \begin{Bmatrix} \xi_{RIG} \\ \xi_{FLEX} \end{Bmatrix}$$

Note: NX Nastran does not check that discarded modes are rigid body modes (i.e., ω=0)

Rigid Body Modes

- When this transformation is applied to the dynamic system and modes are the unit mass normalized, we obtain:

$$\begin{bmatrix} I_{RIG} & 0 \\ 0 & I_{FLEX} \end{bmatrix} \begin{Bmatrix} \xi_{RIG} \\ \xi_{FLEX} \end{Bmatrix} + [\Phi^T B \Phi] \begin{Bmatrix} \xi_{RIG} \\ \xi_{FLEX} \end{Bmatrix} + \begin{bmatrix} 0 & 0 \\ 0 & \omega_{FLEX}^2 \end{bmatrix} \begin{Bmatrix} \xi_{RIG} \\ \xi_{FLEX} \end{Bmatrix}$$

$$= \begin{Bmatrix} \Phi_{RIG}{}^T P \\ \Phi_{FLEX}{}^T P \end{Bmatrix} + \begin{bmatrix} \Phi_{RIG}{}^T \\ \Phi_{FLEX}{}^T \end{bmatrix} \{N + Q\}$$

Rigid Body Modes

- As a result of the transformation, the following consequences occur:
  - Constraint forces are not externally active, i.e.,

$$\begin{bmatrix} \Phi_{RIG}^{T} \\ \Phi_{FLEX}^{T} \end{bmatrix} \{Q\} = \{0\}$$

  - If damping elements are not connected to ground, then:

$$[\Phi_{RIG}^{T}][B] = [0]$$

Thus,

$$\begin{bmatrix} \Phi_{RIG}^{T} \\ \Phi_{FLEX}^{T} \end{bmatrix} [B][\Phi_{RIG}\ \Phi_{FLEX}] = \left[\begin{array}{c|c} 0 & 0 \\ \hline 0 & \Phi_{FLEX}^{T}[B]\ \Phi_{FLEX} \end{array}\right]$$

  - If damping is “proportional”, then:

$$\begin{bmatrix} \Phi_{RIG}^{T} \\ \Phi_{FLEX}^{T} \end{bmatrix} [B][\Phi_{RIG}\ \Phi_{FLEX}] = \begin{bmatrix} 0 & 0 \\ 0 & 2\xi_i\omega_i \end{bmatrix}$$

Dynamic Matrix Assembly

NX Nastran Dynamic Analysis

Dynamic Matrix Assembly

- NX Nastran provides direct and modal methods for performing transient and frequency response and complex mode analysis.

- The dynamic matrices are assembled differently depending on the analysis and method.

Damping

- Damping represents energy dissipation observed in structures

- Damping is difficult to accurately model since damping results from many mechanisms:
  - Viscous effects (shock absorber, dashpot)
  - External friction (slippage in structural joints)
  - Internal friction (characteristic of material type)
  - Structural nonlinearities (plasticity)
- Analytical conveniences used to model damping

- Viscous damping force
- $\mathbf{f_v = b\dot{u}}$

b = viscous damping coefficient

- $\mathbf{m\ddot{u} + b\dot{u} + ku = p}$

- Structural damping force
- $\mathbf{f_s = igku}$ where $i = \sqrt{-1}$

g = structural damping coefficient

- $\mathbf{m\ddot{u} + (1 + ig)ku = p}$

FINITE ELEMENT ANALYSIS Predictive Engineering

Structural Damping versus Viscous Damping

- Assume sinusoidal response:

$$u = \bar{u}e^{i\omega t}$$

Then $\dot{u} = i\omega\bar{u}e^{i\omega t}$ $\qquad \ddot{u} = -\omega^2\bar{u}e^{i\omega t}$

- Viscous damping:
  - $m\ddot{u} + b\dot{u} + ku = p(t)$
  - $m(-\omega^2\bar{u}e^{i\omega t}) + b(i\omega\bar{u}e^{i\omega t}) + k\bar{u}e^{i\omega t} = p(t)$
  - $-\omega^2 m\bar{u}e^{i\omega t} + ib\omega\bar{u}e^{i\omega t} + k\bar{u}e^{i\omega t} = p(t)$

- Structural damping:
  - $m\ddot{u} + (1 + ig)ku = p(t)$
  - $m(-\omega^2\bar{u}e^{i\omega t}) + (1 + ig)k\bar{u}e^{i\omega t} = p(t)$
  - $-\omega^2 m\bar{u}e^{i\omega t} + igk\bar{u}e^{i\omega t} + k\bar{u}e^{i\omega t} = p(t)$

Structural Damping versus Viscous Damping

- Both equations are identical if:

$$gk = b\omega \longrightarrow b = \frac{gk}{\omega}$$

Therefore, if structural damping g is to be modeled using viscous damping b, then the equality holds at only one frequency ω_3 (or ω_4)

$$b = \frac{gk}{\omega}$$

if

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

$$b = \frac{gk}{\omega_n} = gm\omega_n$$

but

$$b_c = 2m\omega_n$$

Structural Damping versus Viscous Damping

then

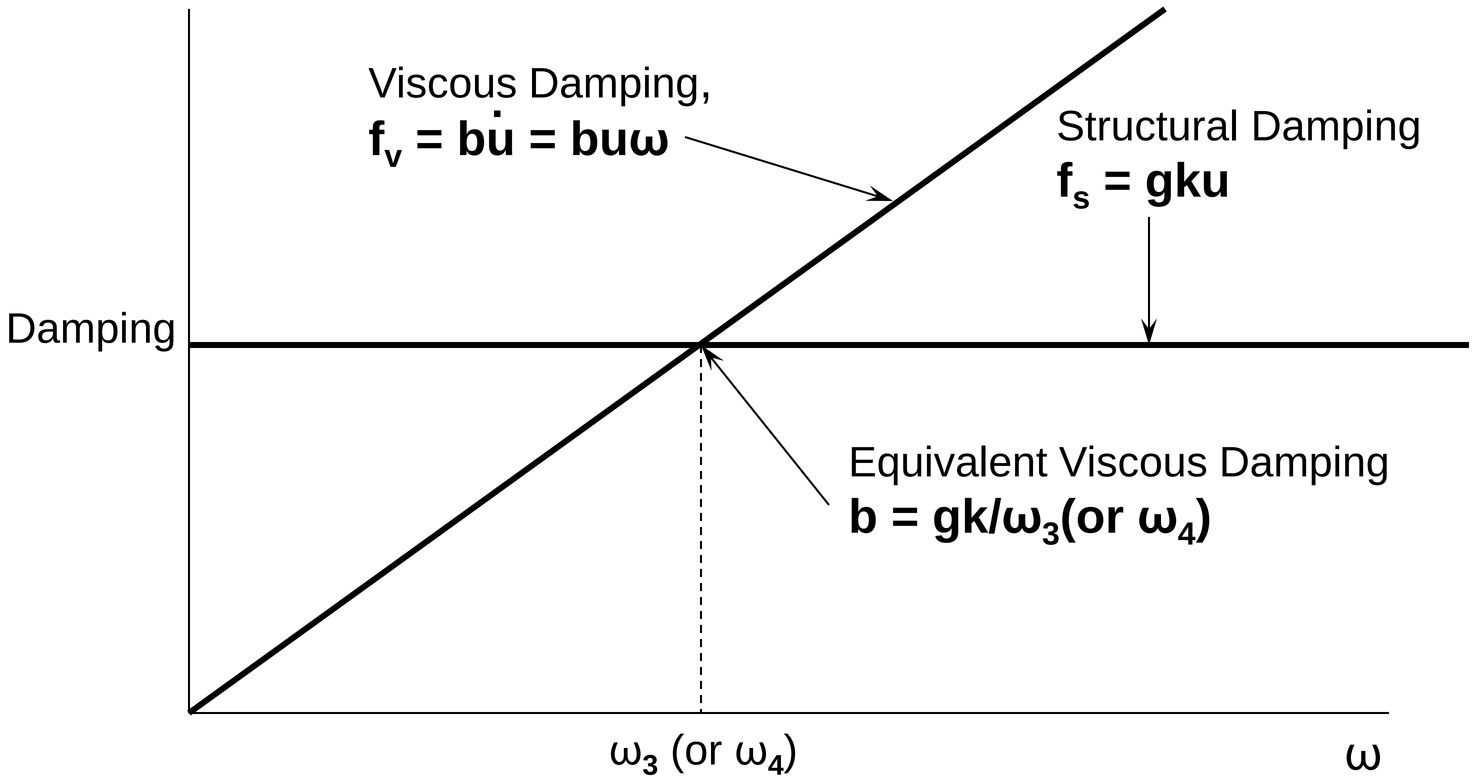
$$\xi = \frac{b}{b_c} = \frac{gm\omega_n}{2m\omega_n} = \frac{g}{2}$$

ξ = critical damping ratio (percent critical damping)

$g = \dfrac{1}{Q}$ = structural damping ratio

Q = quality factor or magnification factor

Structural Damping versus Viscous Damping (Constant Displacement)



- Viscous and structural damping are equivalent at frequency ω_3 (or ω_4)

Damping Summary

- Viscous damping force proportional to velocity
- Structural damping force proportional to displacement
- Critical damping ratio $b/b_c \equiv \xi$
- Quality factor Q inversely proportional to energy dissipated per cycle of vibration
- At resonance $\omega \cong \omega_n$
  - $\xi = g/2$
  - $Q = 1/(2\xi)$
  - $Q = 1/g$

Damping in NX Nastran for FEMAP

Damping Type	Direct Transient	Modal Transient	Direct Frequency	Modal Frequency
Viscous Element	$b \equiv \frac{\text{Force}}{\text{Velocity}}$	$b \equiv \frac{\text{Force}}{\text{Velocity}}$	$b \equiv \frac{\text{Force}}{\text{Velocity}}$	$b \equiv \frac{\text{Force}}{\text{Velocity}}$
Material	$b \equiv \frac{kg_e}{\omega_4}$	$b \equiv \frac{kg_e}{\omega_4}$	GE	GE
Overall Structural	$b \equiv \frac{kg}{\omega_3}$	$b \equiv \frac{kg}{\omega_3}$	PARAM,G	PARAM,G
Modal	N/A	TABDMP	N/A	TABDMP

Damping Input

- Structural Damping
  - MATi Bulk Data Entries

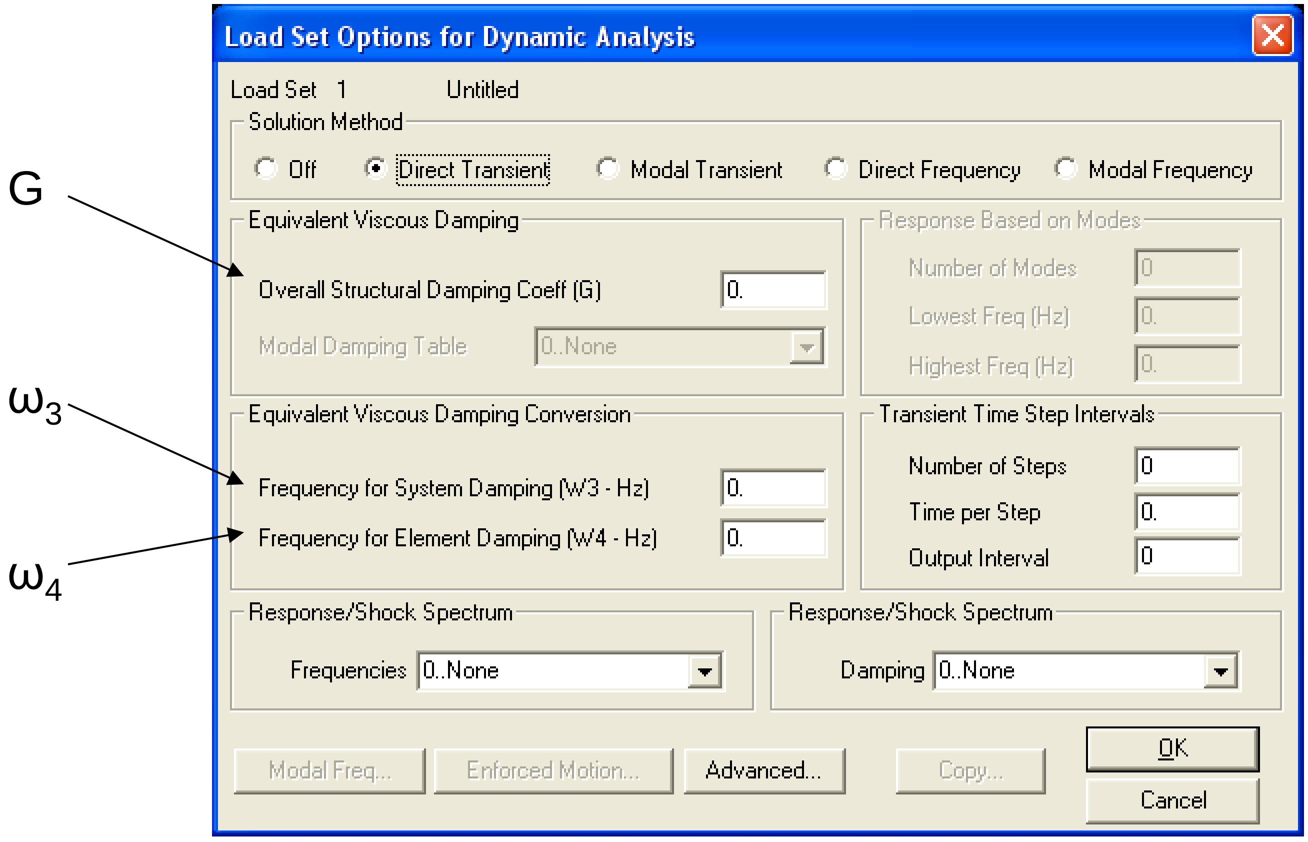
1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
MAT1	2	3.E+7		0.3	0.	0.	0.	0.1	

- PARAM,G,factor (default = 0.0)
  - Overall structural damping coefficient to multiply entire system stiffness matrix
- PARAM,W3,factor (default = 0.0)
  - Converts overall structural damping to equivalent viscous damping
- PARAM,W4,factor (default = 0.0)
  - Converts element structural damping to equivalent viscous damping
- Units for W3,W4 in NX Nastran for FEMAP are in Hertz (Hz)

- If PARAM,G is used, PARAM,W3 must be given a setting greater than zero; otherwise, PARAM,G is ignored in transient response analysis

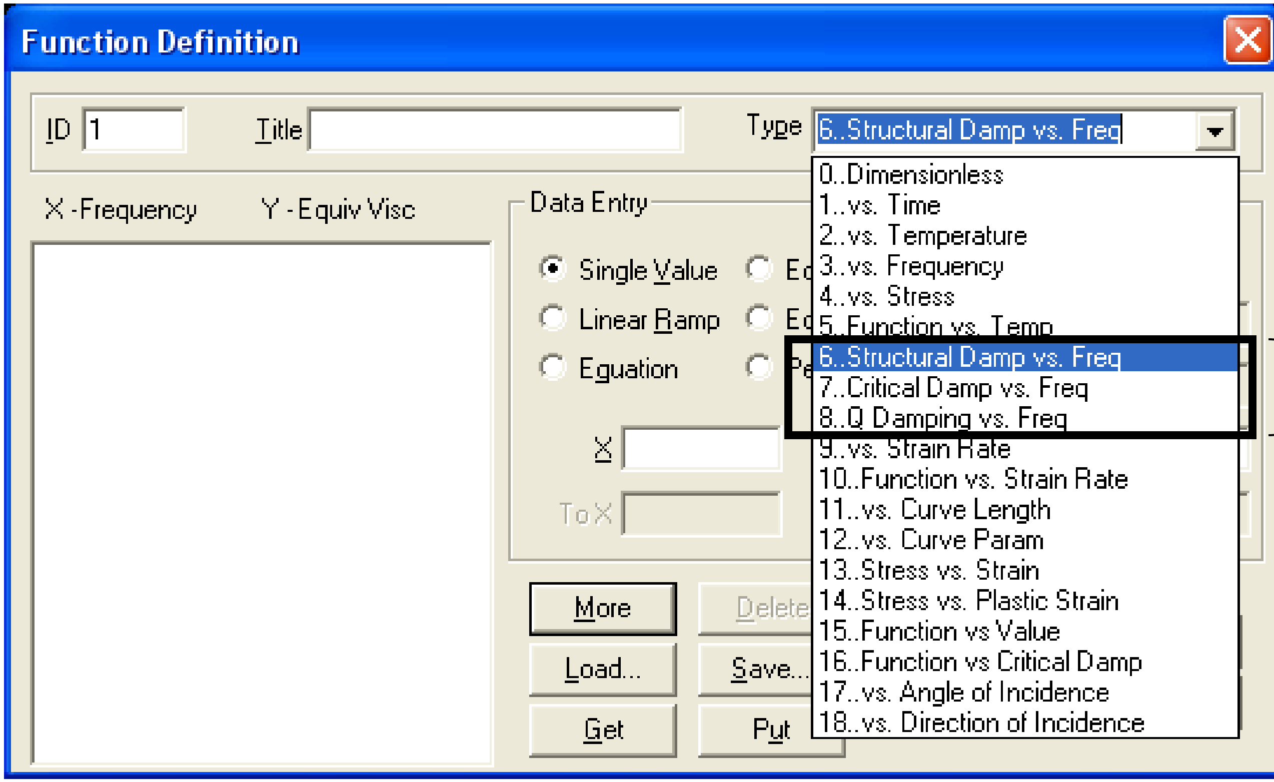
Damping Input

- Setting the PARAMs for G, ω_3, and ω_4 in NX Nastran for FEMAP is accomplished using the Model->Load->Dynamic Analysis command. The Load Set Options for Dynamic Analysis dialog box:



Damping Input

- Modal Damping can be set using the Model->Function command. Choose the type of damping desired from the Type drop-down menu in the Function Definition dialog box:



Damping function types:

6..Structural vs. Freq

7..Critical Damp vs. Freq

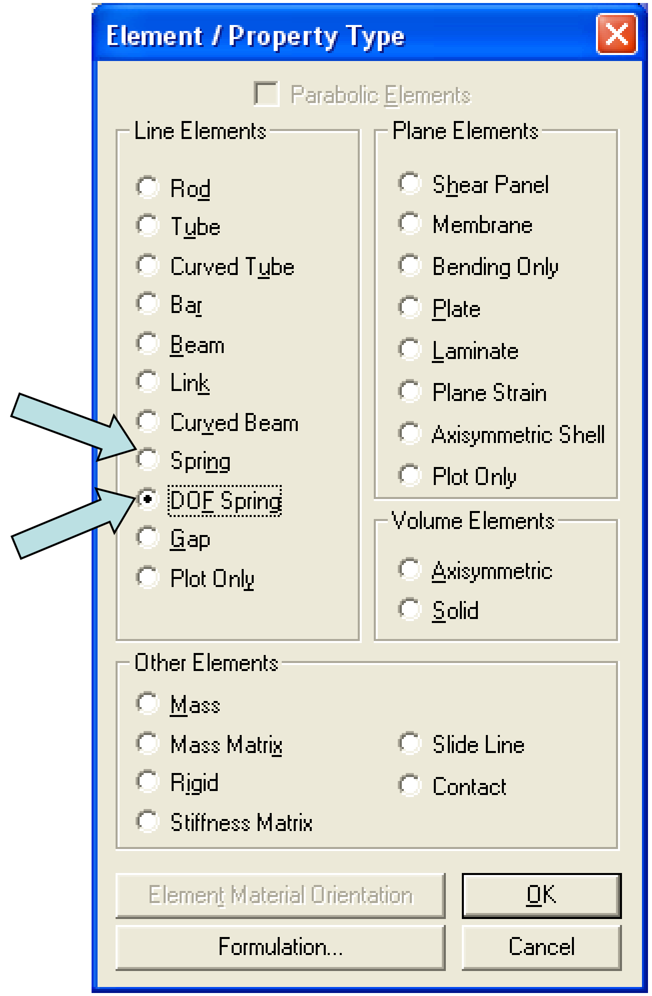
8..Q Damping vs. Freq

Damping Input

- Scalar viscous damping

CDAMP1 — Scalar damper between two DOFs with reference to a property entry

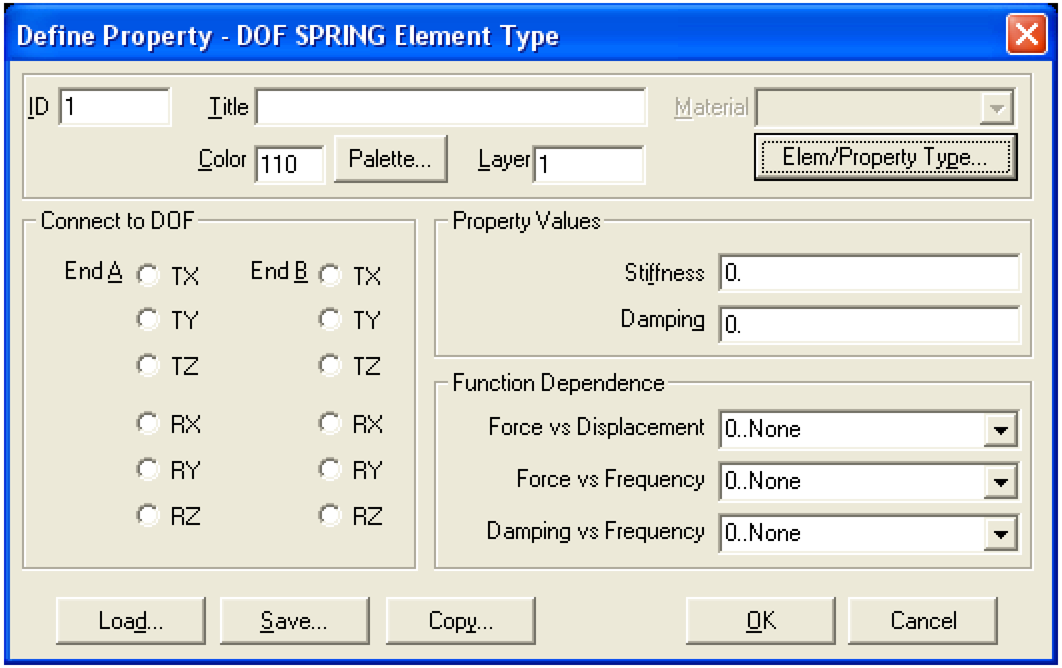
CVISC — Element damper between two grid points; references a property entry (PVISC)



Damping values are assigned through spring properties

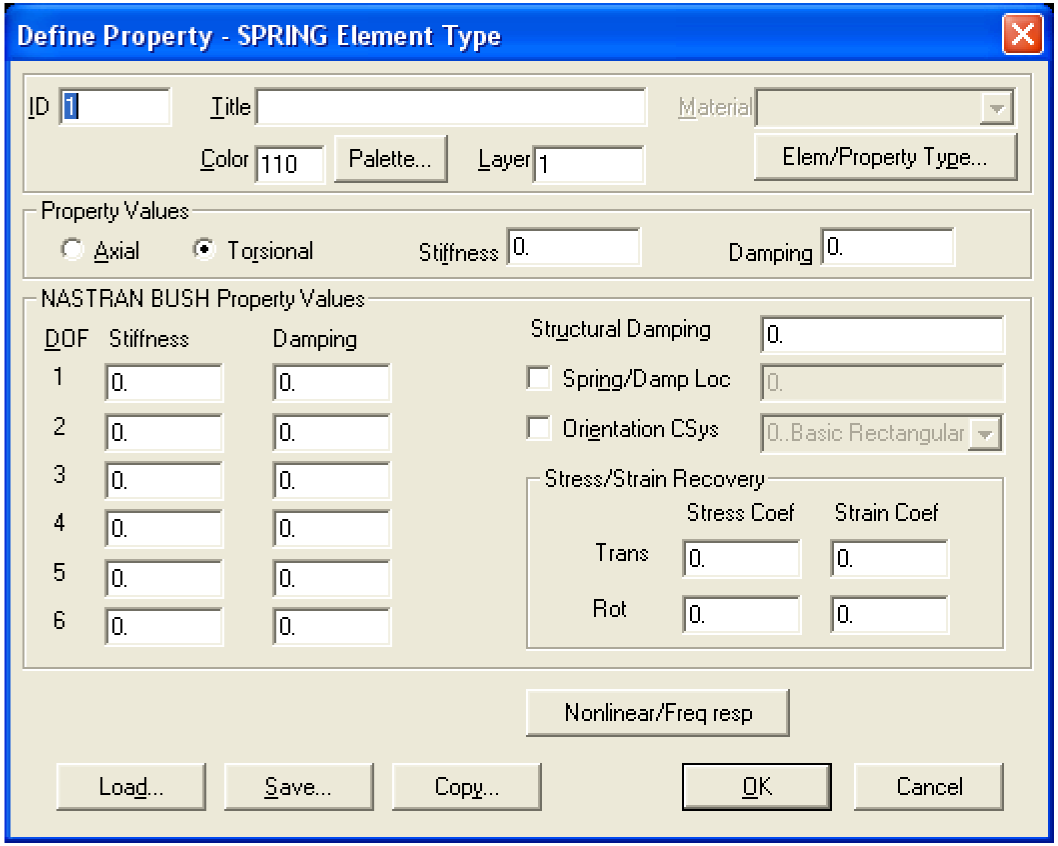
Damping Input

- DOF Spring Property creates CDAMP1



Damping Input

- SPRING Property creates PVISC Property card



Also, used to create the PBUSH property card for Nastran CBUSH elements

Direct Methods

- The general dynamic equation used in the direct method is:

$$[M_{dd}p^2 + B_{dd}p + K_{dd}]\{u_d\} = \{P_d\}$$

where p = a derivative operator

u_d = the union of the analysis set u_a and extra points u_e

- For frequency response and complex eigenvalue analysis, the dynamic matrices are:

$$[K_{dd}] = (1 + ig)[K^1_{dd}] + [K^2_{dd}] + i[K^4_{dd}]$$
$$[B_{dd}] = [B^1_{dd}] + [B^2_{dd}]$$
$$[M_{dd}] = [M^1_{dd}] + [M^2_{dd}]$$

- For transient response, the dynamic matrices are:

$$[K_{dd}] = [K^1_{dd}] + [K^2_{dd}]$$
$$[B_{dd}] = [B^1_{dd}] + [B^2_{dd}] + \frac{g}{\omega_3}[K^1_{dd}] + \frac{1}{\omega_4}[K^4_{dd}]$$
$$[M_{dd}] = [M^1_{dd}] + [M^2_{dd}]$$

FINITE ELEMENT ANALYSIS
Predictive Engineering

Dynamic Matrix Definitions

[K^1_{dd}] is the reduced structural stiffness matrix plus the reduced direct input K2GG (symmetric).

[K^2_{dd}] is the reduced direct input matrix K2PP plus the reduced transfer function input (symmetric or unsymmetric).

[K^4_{dd}] is the reduced structural damping matrix obtained by multiplying the stiffness matrix [K_e] of an individual structural element by an element damping factor g_e and combining results for all structural elements (symmetric).

[B^1_{dd}] is the reduced viscous damping matrix plus the reduced direct input B2GG (symmetric).

[B^2_{dd}] is the reduced direct input matrix B2PP plus the reduced transfer function input (symmetric or unsymmetric).

[M^1_{dd}] is the reduced mass matrix plus the reduced direct input M2GG (symmetric).

[M^2_{dd}] is the reduced direct input matrix M2PP plus the reduced transfer function input (symmetric or unsymmetric).

g, ω_3, ω_4 are the constants specified by the user.

Modal Methods

- The general dynamic equation used in the modal method is:

$$[M_{hh}p^2 + B_{hh}p + K_{hh}]\{u_h\} = \{P_h\}$$

where p = a derivative operator

u_h = the union of the modal coordinates ξ_I and extra points u_e

- The transformation between ξ_I and u_a is:

$$\{u_a\} = [\Phi_{ai}]\{\xi_i\}$$

where $[\Phi_{ai}]$ is the matrix of eigenvectors obtained in real eigenvalue analysis

- The transformation from uh to ud is obtained by augmenting $[\Phi_{ai}]$ to include the extra points.

$$\{u_d\} = [\Phi_{dh}]\{u_h\}$$

where $$[\Phi_{dh}] = \begin{bmatrix} \Phi_{ai} & 0 \\ 0 & I_{ee} \end{bmatrix}$$

$$\{u_h\} = \begin{bmatrix} \xi_i \\ u_e \end{bmatrix}$$

Modal Methods

- For frequency response and complex eigenvalue analysis, the dynamic matrices are:

$$[K_{hh}] = [k_i] + [\Phi_{dh}]^T(ig[K^1_{dd}] + [K^2_{dd}] + i[K^4_{dd}])\,[\Phi_{dh}]$$
$$[B_{hh}] = [b_i] + [\Phi_{dh}]^T([B^1_{dd}] + [B^2_{dd}])\,[\Phi_{dh}]$$
$$[M_{hh}] = [m_i] + [\Phi_{dh}]^T[M^2_{dd}][\Phi_{dh}]$$

where $[m_i]$ = a diagonal matrix with terms $m_{ii} = [\Phi_{ai}]^T[M_{aa}][\Phi_{ai}]$

$[b_i]$ = a diagonal matrix with terms $b_{ii} = \omega_i g(\omega_i) m_{ii}$ is the radian frequency of the i-th normal mode and $g(\omega_i)$ is a damping factor obtained from interpolation of a user-supplied table (TABDMP1)

$[k_i]$ = a diagonal matrix with terms kii = $\omega_i^2 m_{ii}$

- If parameter

KDAMP = -1, then

$m_{ii} = m_{ii}$

$b_{ii} = 0$

$k_{ii} = (1 + ig(\omega_i))k_{ii}$

g(ωi) is a damping factor obtained from the interpolation of a user-supplied table (TABDMP1)

Modal Methods

- $[m_i]$, $[b_i]$, and $[k_i]$ are expanded by the addition of zeros to the rows and columns corresponding to the extra points (u_e).

- For transient response the dynamic matrices are:

$$[K_{hh}] = [k_i] + [\Phi_{dh}]^T([K^2_{dd}][\Phi_{dh}]$$

$$[B_{hh}] = [b_i] + [\Phi_{dh}]^T([B^1_{dd}] + [B^2_{dd}] + \frac{g}{\omega_3}[K^1_{dd}] + \frac{1}{\omega_4}[K^4_{dd}])\,[\Phi_{dh}]$$

$$[M_{hh}] = [m_i] + [\Phi_{dh}]^T[B^2_{dd}][\Phi_{dh}]$$

- If only $[m_i]$, $[b_i]$, and $[k_i]$ are present in any modal dynamic analysis, then the modal dynamic equations are uncoupled.

Transient Response Analysis

NX Nastran Dynamic Analysis

Introduction to Transient Response Analysis

- Compute response to time-varying excitation.
- Excitation is explicitly defined in the time domain. All of the applied forces are known at each instant in time.
- Computed response usually includes nodal displacements and accelerations, and element forces and stresses.
- Two categories of analysis – direct and modal

Direct Transient Response

- Dynamic equation of motion

$$[M]\{\ddot{u}(t)\} + [B]\{\dot{u}(t)\} + [K]u(t) = \{p(t)\}$$

- Response solved at discrete times with fixed Δt

- Using central finite difference representation for $\{\dot{u}(t)\}$ and $\{\ddot{u}(t)\}$ at discrete times

$$\{\dot{u}_n\} = \frac{1}{2\Delta t}\{u_{n+1} - u_{n-1}\}$$

$$\{\ddot{u}_n\} = \frac{1}{\Delta t^2}\{u_{n+1} - 2u_n + u_{n-1}\}$$

Note: These equations are also used by NX Nastran to compute velocity and acceleration output.

Direct Transient Response

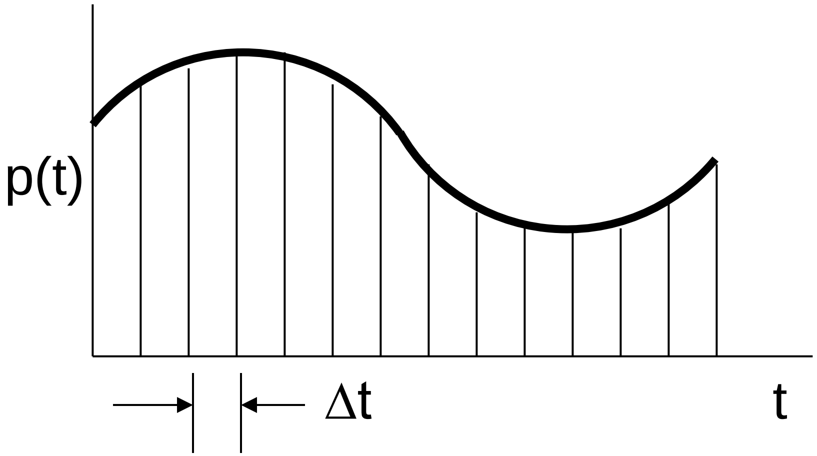
- Numerical integration (Newmark – Beta type method)

  (except "smear" force over 3 adjacent time points)

$$\left[\frac{m}{\Delta t^2}\right](u_{n+1} - 2u_n + u_{n-1}) + \left[\frac{b}{2\Delta t}\right](u_{n+1} - u_{n-1})\left[\frac{k}{3}\right](u_{n+1} + u_n + u_{n-1})$$

$$= \frac{1}{3}(P_{n+1} - P_n + P_{n-1})$$

$$\left.\begin{matrix} \dfrac{u_{n+1} + u_n + u_{n-1}}{3} \\[2ex] \dfrac{p_{n+1} + p_n + p_{n-1}}{3} \end{matrix}\right\} \text{Time Average}$$



- Alternate methods: Wilson – θ, Hughes – α, Bathe

Direct Transient Response

- Solution

$$[A_1]\{u_{n+1}\} = [A_2]^{\cdot} + [A_3]\{u_n\} + [A_4]\{u_{n-1}\}$$

Where

$$[A_1] = [M/\Delta t^2 + B/2\Delta t + K/3] \quad \text{Dynamic Matrix}$$

$$[A_2] = 1/3\,\{P_{n+1} + P_n + P_{n-1}\} \quad \text{Applied Force}$$

$$\left.\begin{array}{l}[A_3] = [2M/\Delta t^2 - K/3] \\ [A_4] = [-M/\Delta t^2 + B/2\Delta t - K/3]\end{array}\right\} \quad \text{Initial Conditions, from previous Time Step}$$

Except that {P(t)} is averaged over three time points and [K] is modified such that the dynamics equation of motion reduces to a $[K]\{u_n\} = \{P_n\}$ if no [M] or [B]

Direct Transient Response

- Solve by decomposing A_1 and applying it to the right-hand side of the above equation.
- Similar to classical Newmark-Beta method, except:
  - Applied force is averaged over 3 adjacent time points.
  - The stiffness K is similarly averaged to get the correct static solution for M,B = 0.
- M,B, and K do not change with time.
- A_1 needs to decomposed only once if Δt is unchanged throughout the entire solution. If Δt is changed, A_1 must be re-decomposed (which may increase run times dramatically).
- The output time interval may be greater than the solution time interval (example: use solution Δt of 0.001 second and output results every fifth time step or with output Δt of 0.005 second).

Damping in Modal Transient Response

- If damping matrix B exists, then the assumption is made that it is not diagonalized by Φ:

$$\Phi^T B \Phi \neq \text{diagonal}$$

- The coupled problem is solved using modal coordinates utilizing the direct transient response Newmark-Beta type numerical integration.

$$[A_1]\{\xi_{n+1}\} = [A_2] + [A_3]\{\xi_n\} + [A_4]\{\xi_{n-1}\}$$

where

$$[A_1] = [\Phi]^T[M/\Delta t^2 + B/2\Delta t + K/3][\Phi] \quad \text{Dynamic Matrix}$$

$$[A_2] = 1/3[\Phi]^T\{P_{n+1} + P_n + P_{n-1}\} \quad \text{Applied Force}$$

$$\left.\begin{aligned}[A_3] &= [\Phi]^T[2M/\Delta t^2 - K/3][\Phi] \\ [A_4] &= [\Phi]^T[-M/\Delta t^2 + B/2\Delta t - K/3][\Phi]\end{aligned}\right\} \quad \text{Initial Conditions, from previous Time Step}$$

FINITE ELEMENT ANALYSIS
Predictive Engineering

Damping in Modal Transient Response

- If modal damping is used, then each mode has damping b_i.

- The equations of motion become uncoupled

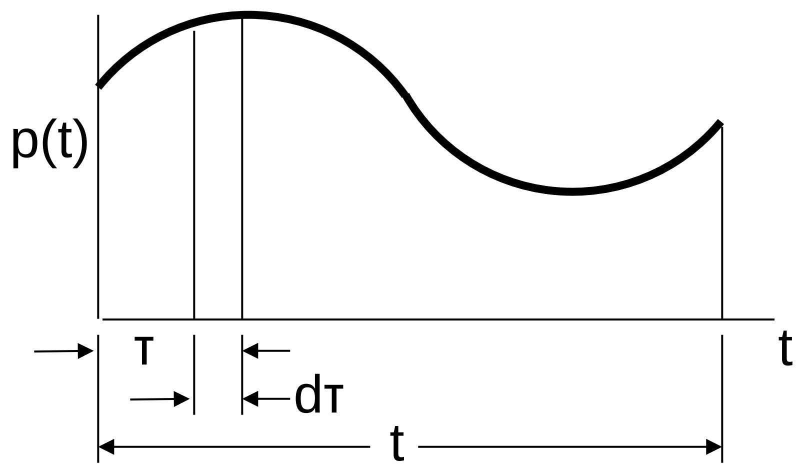
$$m_i\ddot{\xi} + b_i\dot{\xi} + k_i\xi = p_i(t)$$

Damping in Modal Transient Response

- Use Duhamel's integral to solve for modal response as decoupled SDOF systems.
- Duhamel's integral:

$$\xi(t) = e^{-bt/2m}\left[\xi_o \cos \omega_d t + \frac{\xi_o + (b/2m)\xi_o}{\omega_d}\right] +$$

$$e^{-bt/2m}\frac{1}{m\omega_d}\int_0^t e^{-bt/2m} p(\tau)\sin\omega_d(t - \tau)\ \tau d$$



Damping in Modal Transient Response

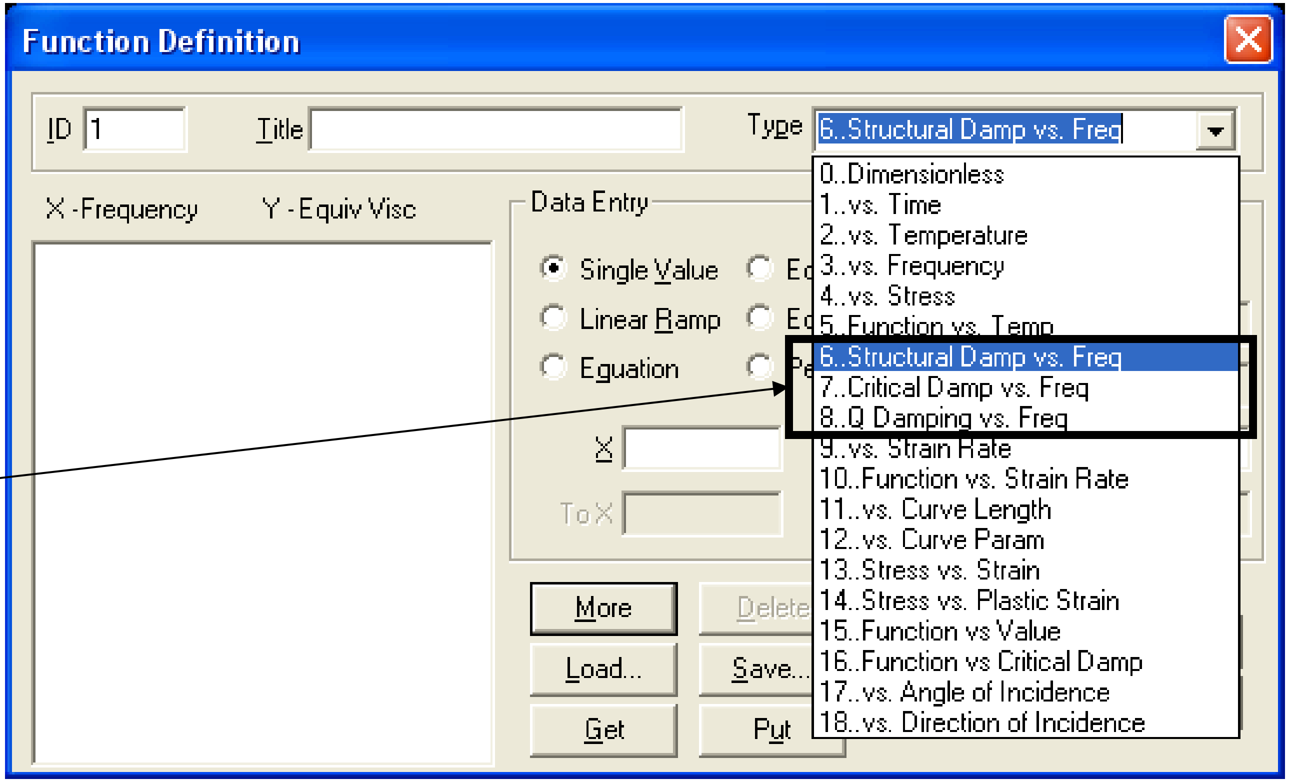
- Most efficient to use modal damping ratios since equations are decoupled
- TABDMP1 Bulk Data entry defines the modal damping ratios.
- Example: For 10% Critical Damping

g = 0.2 (6)

CRIT = 0.10 (7)

Q = 5.0 (8)

These are the three options for modal damping



Data Recovery in Modal Transient Response

- Recover physical response as the summation of the modal responses

$$u = [\Phi] \{\xi\}$$

- Not as large a computational penalty for changing Δt in modal transient response as in direct. However, the constant Δt is still recommended.

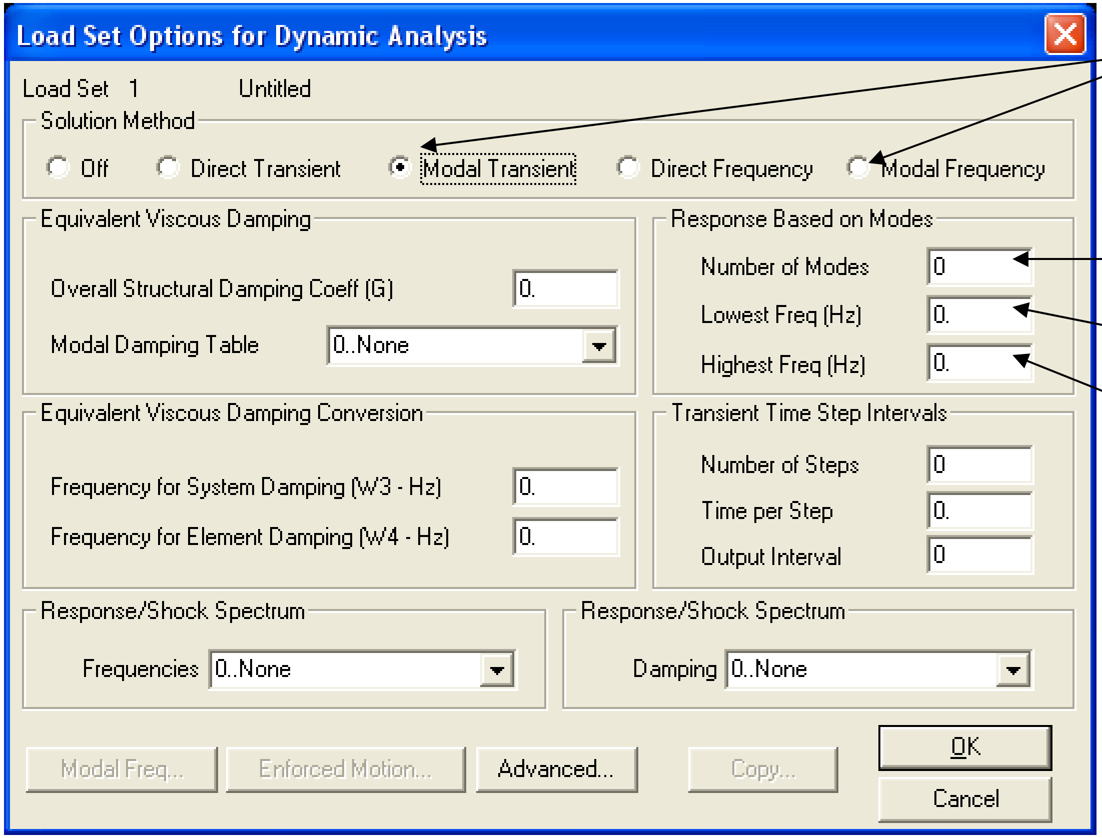
- The output interval may be greater than the solution time interval

Mode Truncation

- May not need all of the computed modes. Often only the lowest few will suffice for dynamic response calculation.
- PARAM,LFREQ gives the lower limit on the frequency range of retained modes.
- 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.
- Truncating high-frequency modes truncates high-frequency response.

Mode Truncation

- Mode truncation PARAMs can be entered in FEMAP using Model->Load->Dynamic Analysis command. The Load Set Options for Dynamic Analysis dialog box will appear (Modal Transient or Modal Frequency must be selected as the Solution Method):



Solution Methods (make Response Based on Modes fields active)

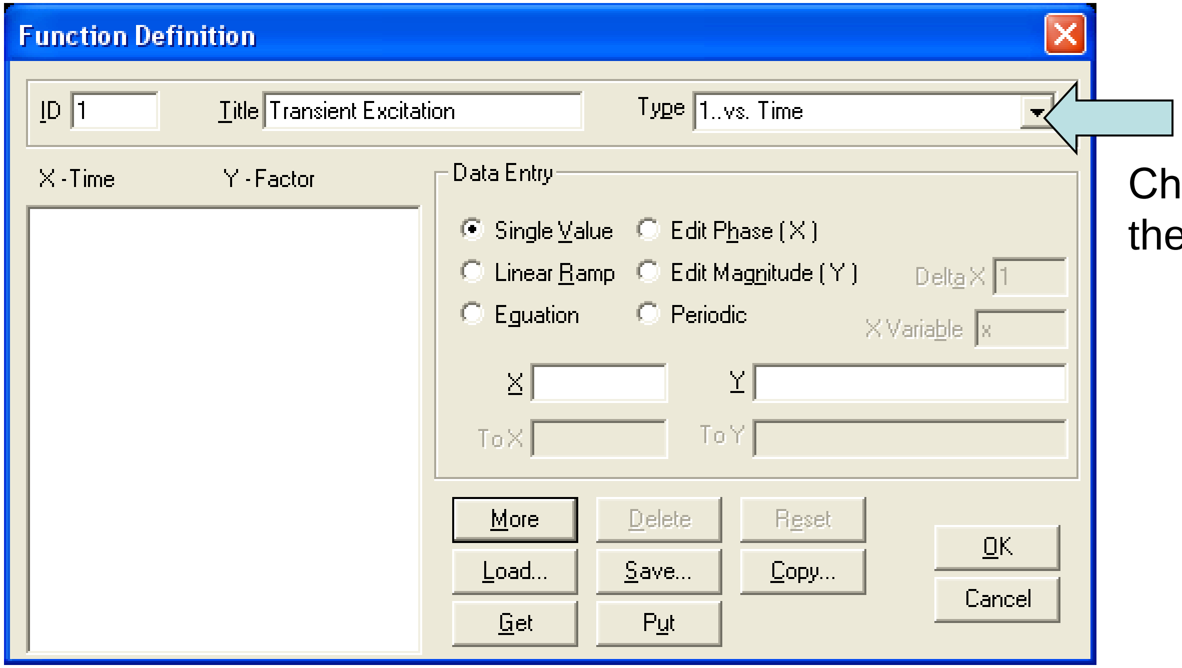
PARAM,LMODES

PARAM,LFREQ

PARAM,HFREQ

Transient Excitation

- Define force as a function of time using the Model->Function command and choosing “1..vs Time” as the Type



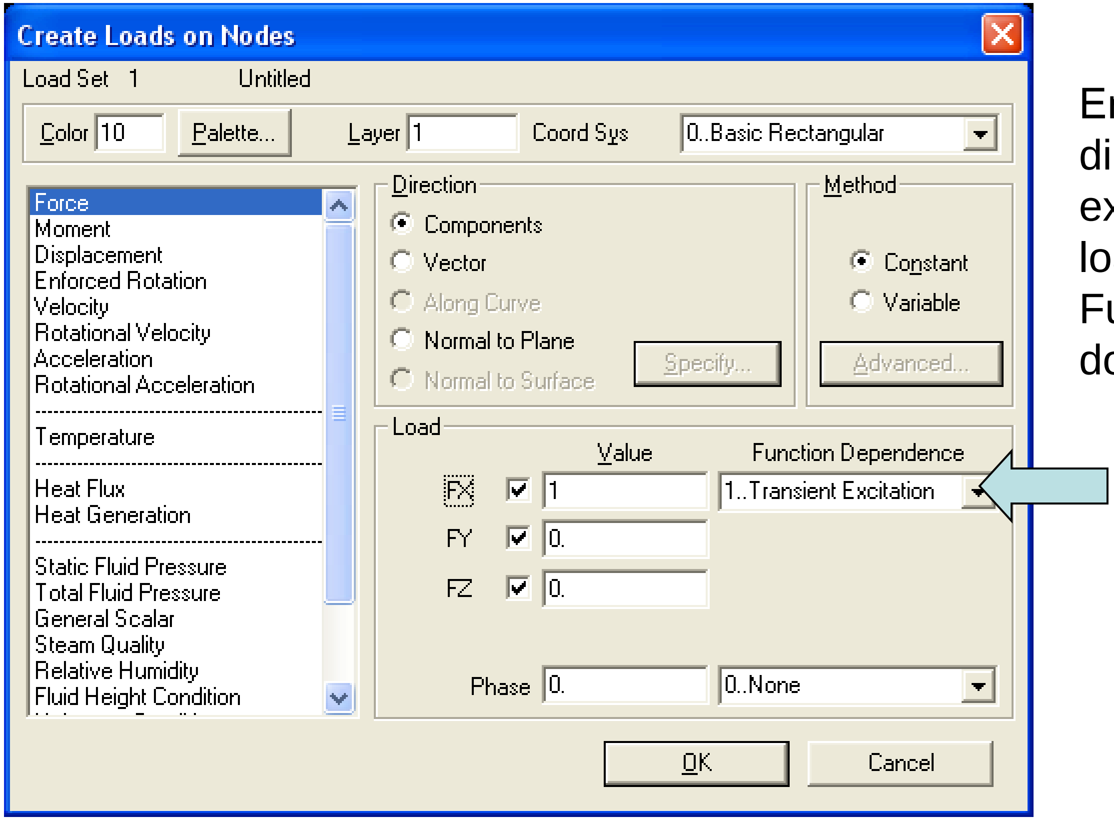
Choose “1..vs Time” from the Type drop-down menu

Enter values for the Excitation in the Y-field and values for Time in the X-field. Also, a linear ramp or other equation can be used to create values automatically. Once the values are input, Click OK.

Note: Tabular information in an Excel spreadsheet (2 columns maximum) or comma separated table can be pasted into FEMAP from the clipboard to create a function using the Get button

Transient Excitation

- Create a functionally dependent load representing dynamic excitation by using the Model->Load->(Nodal or Elemental) command. The Create Loads on (Nodes or Elements) dialog will appear:

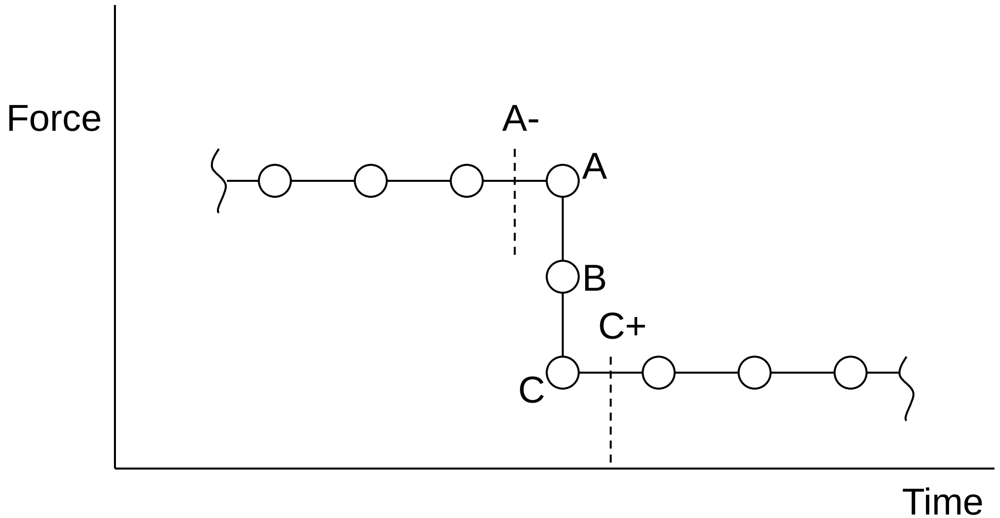


Enter a "unit" value in the direction of the dynamic excitation and choose a loading function from the Function Dependence drop-down menu.

FINITE ELEMENT ANALYSIS
Predictive Engineering

Transient Excitation Considerations

- The 1/3 “smearing” of applied loads must be taken into account. This will smooth the force and decrease apparent frequency content.
- Avoid discontinuous forces. These may cause different results on different computers.



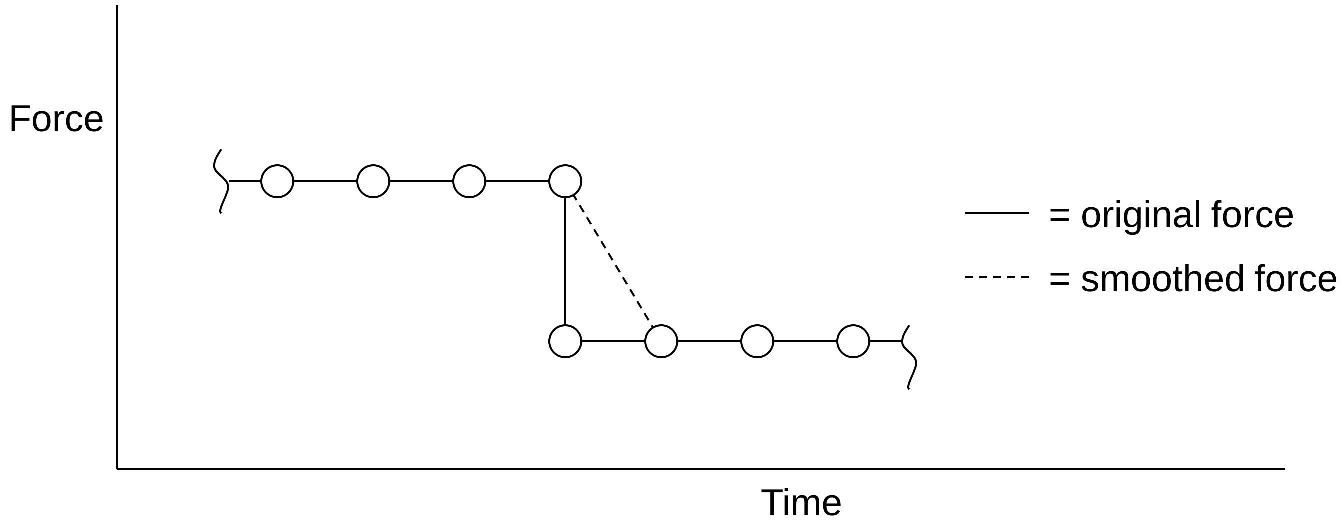
If NΔt causes a solution at ABC, then NX Nastran should select the average force B.

However, due to numerical round-off, NΔt on one computer may be at time A- and will give force A. On another computer, NΔt may be at time C+ and will give force C.

The integration results will differ depending on whether the force at NΔt is A, B, or C.

Transient Excitation Considerations

- Smooth a discontinuous force over one Δt



Initial Conditions

- May impose initial displacement and/or velocity in direct transient response via the TIC Bulk Data entry. Initial Conditions are not available in standard modal transient.
- The IC Case Control Command selects the TIC entry.
- Attention – initial conditions for unspecified DOFs are set to Zero.
- Initial conditions may be specified only for A-set DOFs.
- Initial conditions may be specified only in direct transient response. In modal transient response, all initial conditions are set to zero.
- Initial conditions are used to determine the values of $\{u_0\}$, $\{u_{-1}\}$, $\{P_0\}$, and $\{P_{-1}\}$ used in calculating $\{u_1\}$. The acceleration for all points is assumed to be zero for time, $t \leq 0$ (constant velocity).

$$\{u_{-1}\} = \{u_0\} - \{\dot{u}_0\}\Delta t$$

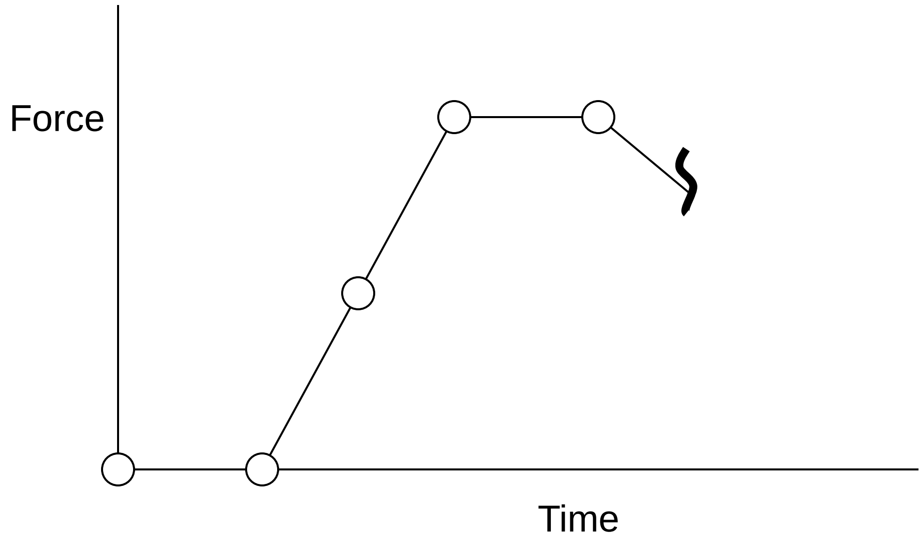
$$\{P_{-1}\} = [K]\{u_{-1}\} - [B]\{\dot{u}_0\}$$

The load specified by the user at $t = 0$ is replaced by:

$$\{P_0\} = [K]\{u_0\} - [B]\{\dot{u}_0\}$$

Initial Conditions

- The recommended practice for any type of dynamic excitation is to use at least one time step of zero excitation prior to applying the dynamic force



TSTEP Entry

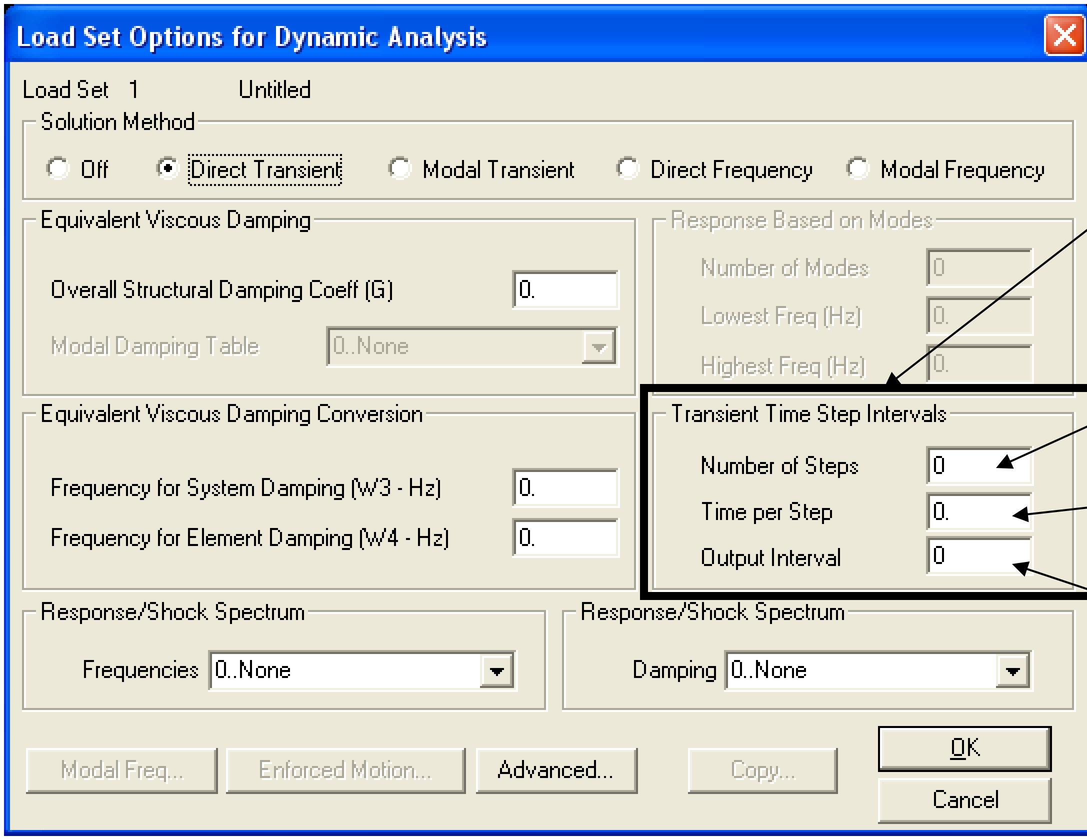
- Select integration time step for direct and modal transient response.
  - Integration errors increase with increasing natural frequency.
  - Recommended Δt is to use at least eight solution time steps per period (cycle) of response.
- The TSTEP Bulk Data entry controls solution and output Δt, and is selected by the TSTEP Case Control command.

- The cost of integration is directly proportional to the number of time steps when Δt is constant.

- Use adequate length of time to properly capture long-period (low-frequency) response.

TSTEP Entry

- Creating the TSTEP entry in FEMAP is accomplished using Model->Load->Dynamic Analysis command. The Load Set Options for Dynamic Analysis dialog box will appear:



Portion of dialog box for creating the Ni, DTi, and NOi, entries on the TSTEP Bulk data card.

Ni, Number of Time Steps of value DTi

DTi, Time increment

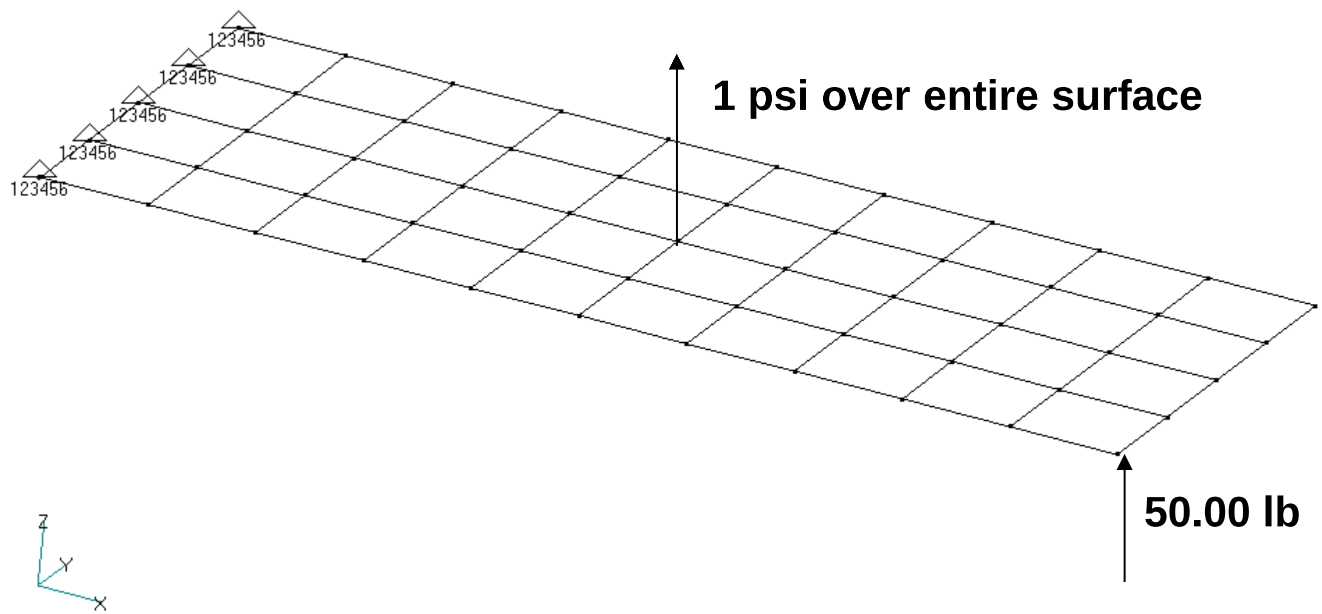
NOi, Skip factor for output. Every NOi-th step will be saved for output (default =1)

Problem #3

Direct Transient Response

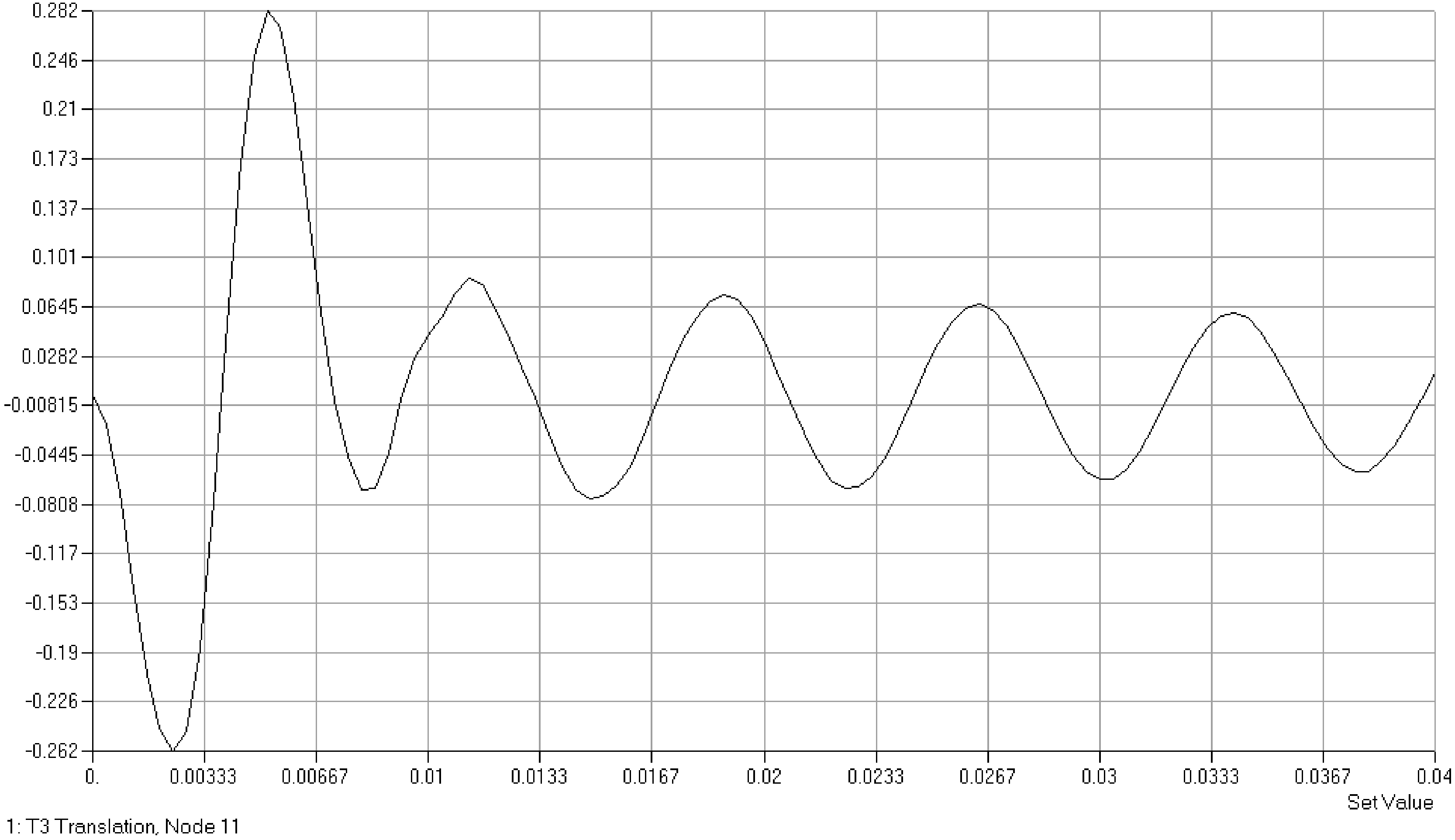
Problem #3: Direct Transient Response

For this problem, use the direct method to determine the transient response of the flat rectangular plate, created in problem #1, subject to a time-varying excitation. The structure is excited by a **1 psi pressure load** over the total surface of the plate varying at **250 Hz**. It is also excited with a **50 lb force** in the lower right corner (node **11**) which is also varying at **250 Hz, but 180 out-of-phase with the pressure load**. Both time-dependent dynamic loads are applied for a duration of **0.008 seconds** only. Use structural damping of **g = 0.06** and convert this damping to equivalent viscous damping at **250 Hz**. Continue the analysis to **0.04 seconds**.



Problem #3: Direct Transient Response

T3 Translation for Node 11 over complete duration of analysis



Problem #3: Direct Transient Response

Use these results for comparison:

Time 0.0024	T3 Displacement
Node 11	-0.26051 in
Node 33	-0.287 in
Node 55	-0.31076 in

Time 0.0052	T3 Displacement
Node 11	0.27895 in
Node 33	0.31823 in
Node 55	0.3528 in

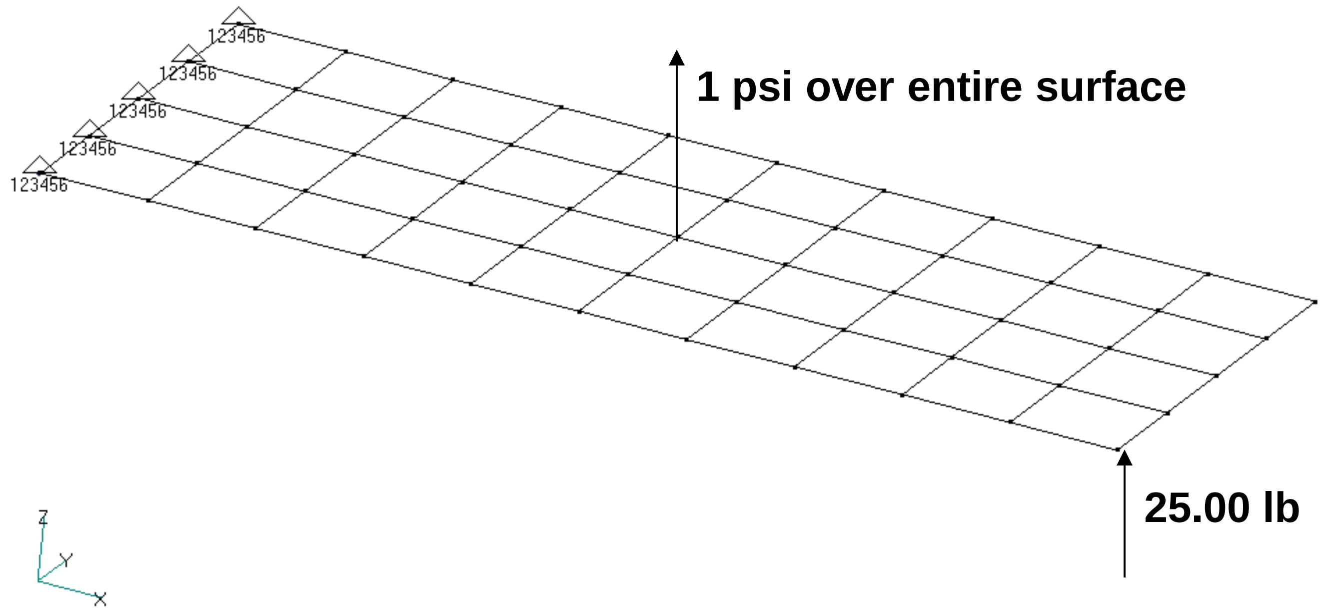
Time 0.02	T3 Displacement
Node 11	0.038693 in
Node 33	0.038876 in
Node 55	0.038945 in

Problem #4

Modal Transient Response

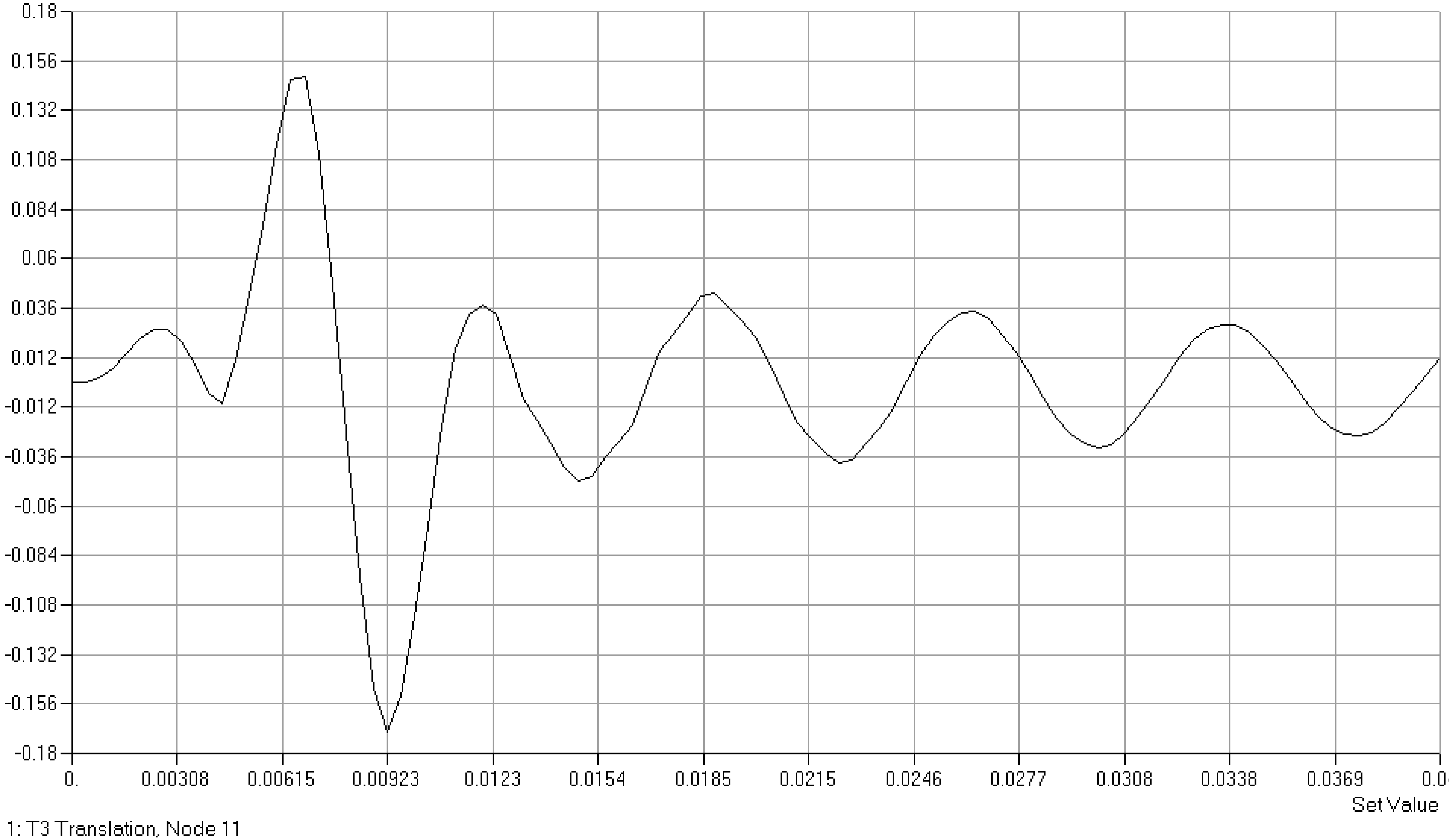
Problem #4: Modal Transient Response

For this problem, use the direct method to determine the transient response of the flat rectangular plate, created in problem #1, subject to a time-varying excitation. The structure is excited by a **1 psi pressure load** over the total surface of the plate varying at **250 Hz**. It is also excited with a **25 lb force** in the lower right corner (node **11**) which is also varying at **250 Hz, but starting 0.004 seconds after the pressure load**. Both time-dependent dynamic loads are applied for a duration of **0.008 seconds** only. Use a modal damping table of **ξ = 0.03** for all the modes. Continue the analysis to **0.04 seconds**.



Problem #4: Modal Transient Response

T3 Translation for Node 11 over complete duration of analysis



Problem #4: Modal Transient Response

Use these results for comparison:

Time .0068	T3 Displacement
Node 11	0.14765 in
Node 33	0.16062 in
Node 55	0.17344 in

Time .0092	T3 Displacement
Node 11	-0.16902 in
Node 33	-0.18492 in
Node 55	-0.20055 in

Time .022	T3 Displacement
Node 11	-0.038703 in
Node 33	-0.03752 in
Node 55	-0.036237 in

Frequency Response Analysis

NX Nastran Dynamic Analysis

Introduction to Frequency Response Analysis

- Compute response to oscillatory excitation.
- Excitation is explicitly defined in the frequency domain. All of the applied forces are known at each forcing frequency.
- Computed response usually includes nodal displacements and element forces and stresses.
- The computed responses are the complex numbers defined as magnitude and phase (with respect to forcing) or as real and imaginary components
- Two categories of analysis – direct and modal

Direct Frequency Response

- Dynamic equation of motion

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

- PARAM,G and GE on MATi entry do not form a damping matrix. They form a complex stiffness matrix

$$K = (1 + iG)K^1 + i\sum G_E k_E$$

where K^1 = global stiffness matrix

G = overall structural damping coefficient (PARAM,G)

k_E = element stiffness matrix

G_E = element structural damping coefficient (GE on MATi entry)

- Solve the equation by inserting to form a complex left-hand side, and then solve it similar to a Statics problem (using complex arithmetic)

Modal Frequency Response

- Convert to modal coordinates and solve as decoupled SDOF systems

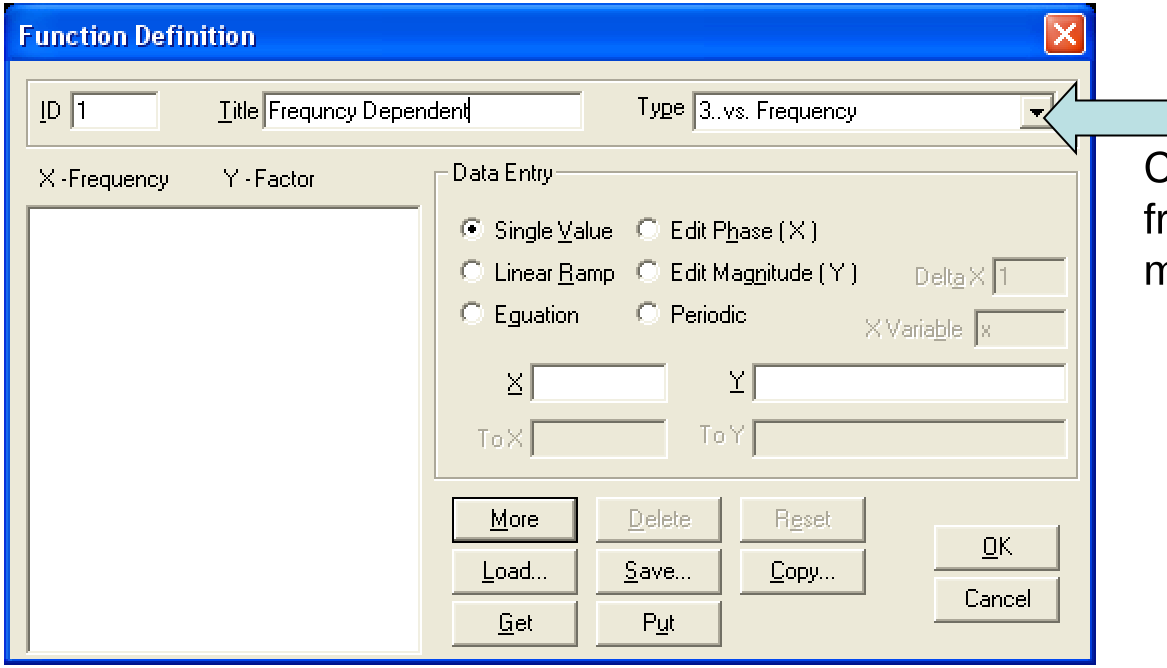
$$\xi_i = \frac{P_i}{-m_i\omega + ib_i\omega + k_i}$$

- Much quicker to solve this equation than in direct method

- Decoupled procedure can be used only if either no damping is present or if modal damping alone (via TABDMP1) is used. Otherwise, use the less efficient direct approach (on smaller modal coordinate matrices) if non-modal damping (VISC,DAMP) is present.

Excitation Definition

- Define force as a function of time using the Model->Function command and choosing “3..vs Frequency” as the Type

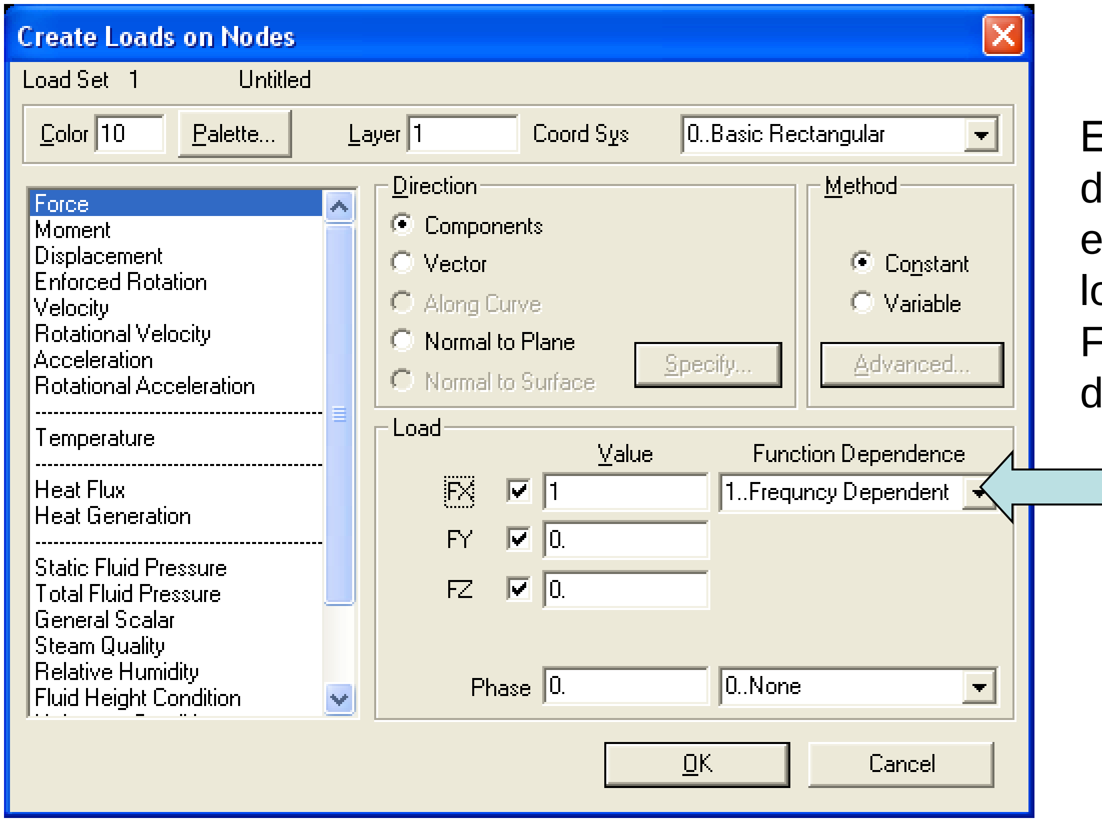


Choose “3..vs Frequency” from the Type drop-down menu

Enter values for the Excitation in the Y-field and values for Frequency in the X-field. Also, a linear ramp or other equation can be used to create values automatically. Once the values are input, Click OK.

Excitation Definition

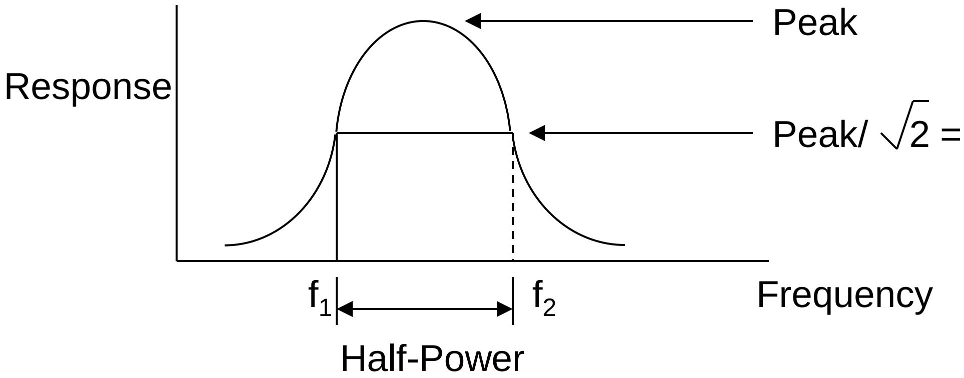
- Create a functionally dependent load representing dynamic excitation by using the Model->Load->(Nodal or Elemental) command. The Create Loads on (Nodes or Elements) dialog will appear:



Enter a “unit” value in the direction of the dynamic excitation and choose a loading function from the Function Dependence drop-down menu.

Frequency Response Considerations

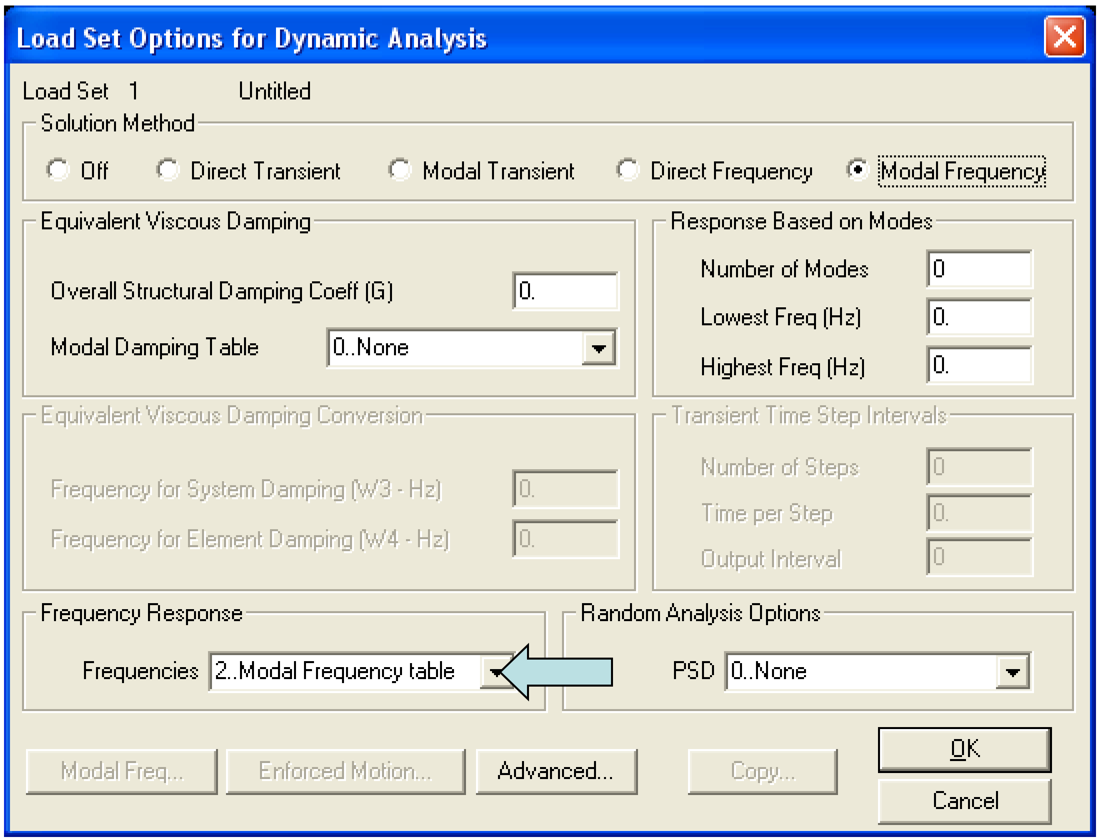
- Exciting an undamped (or modal damped) system at 0.0 Hz gives the same results as a static analysis. Therefore, if the maximum excitation frequency is much less than the lowest resonant frequency of the system, a static analysis is sufficient.
- Very lightly damped structures exhibit large dynamic responses for excitation frequencies near resonant frequencies. A small change in the model (or running it on another computer) may give large changes in such response.
- Use a fine enough frequency step size (Δf) to adequately predict peak response. Use at least 5 points per half-power bandwidth.



- For maximum efficiency, use an uneven frequency step size. Use smaller Δf in regions of resonant frequencies and larger Δf in regions removed from resonant frequencies

Solution Frequencies

- Define a table of solution frequencies using the Model->Function command. Then use the Model-> Load->Dynamic Analysis command to choose the Modal Frequency Table. The Load Set Options for Dynamic Analysis dialog box will appear:



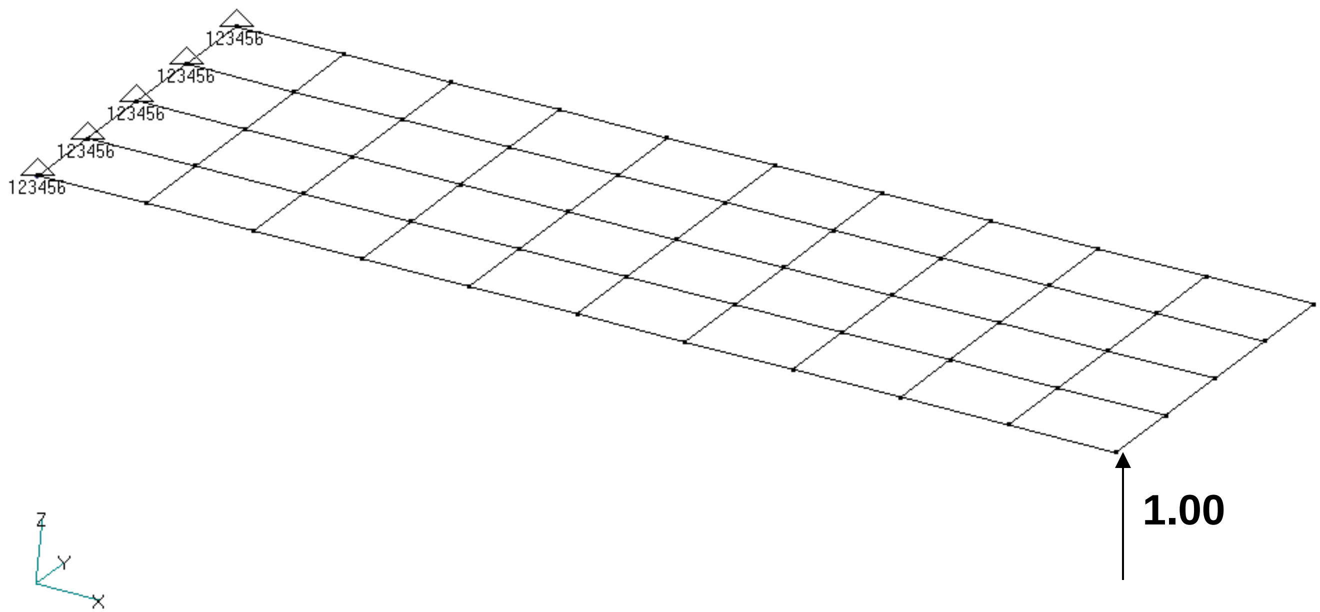
If a normal modes analysis has already been performed and the results read into FEMAP, then Modal Frequency Table can be created automatically from those modal results by pressing the Modal Freq…button. Some additional factors need to be specified to create the table but in general the defaults in FEMAP are usable.

Problem #5

Direct Frequency Response

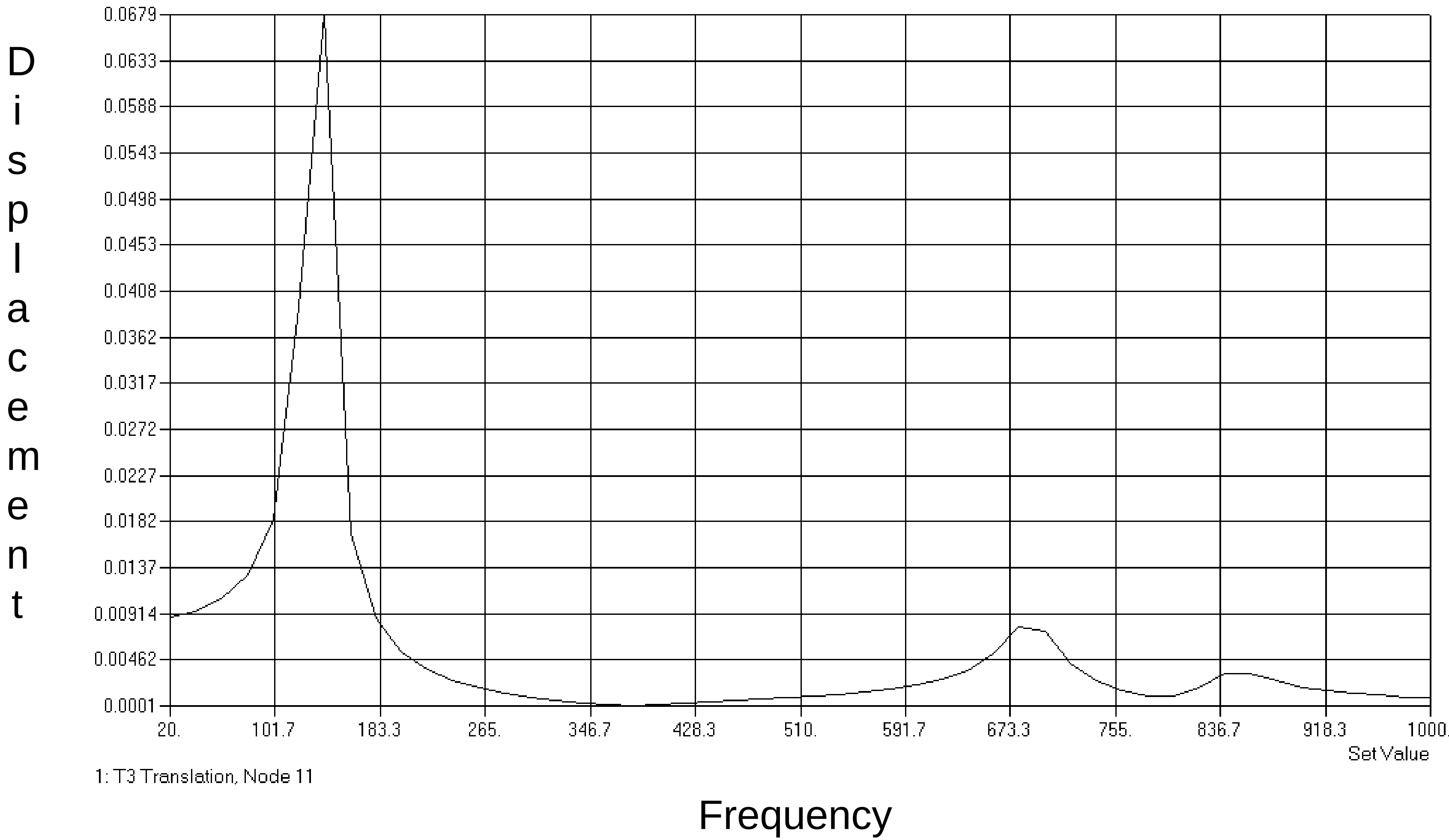
Problem #5: Direct Frequency Response

For this problem, use the direct method to determine the frequency response of the flat rectangular plate, created in problem #1, subject to a frequency-varying excitation. The structure is excited by a **unit load (1.0)** at the lower right corner (node **11**). Use a **frequency step (∆f) of 20 Hz** between a **range of 20 and 1000Hz**. Use Structural damping **g = 0.06**.



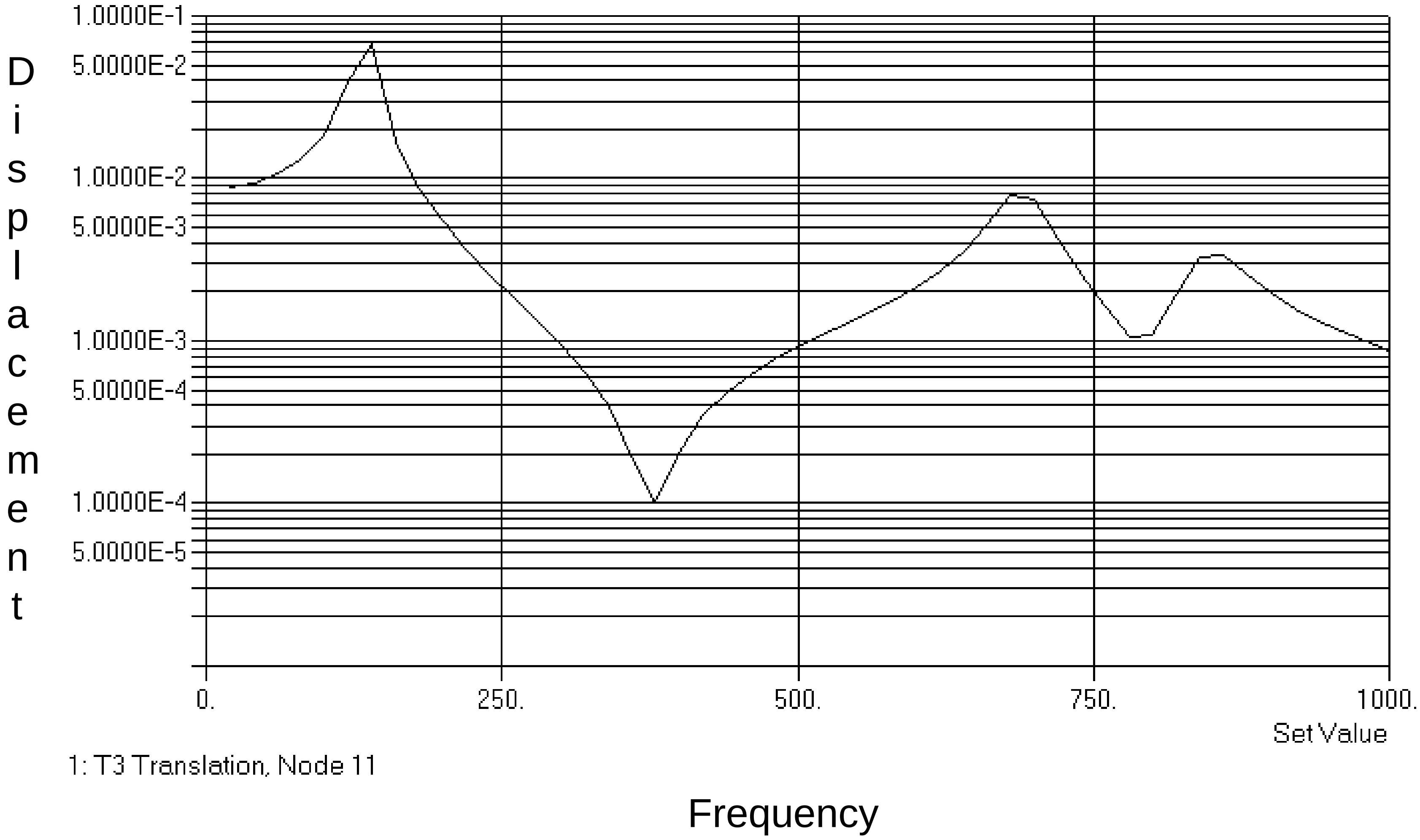
Problem #5: Direct Frequency Response

T3 Translation for Node 11 (Magnitude) over complete duration of analysis (Rectilinear graph)



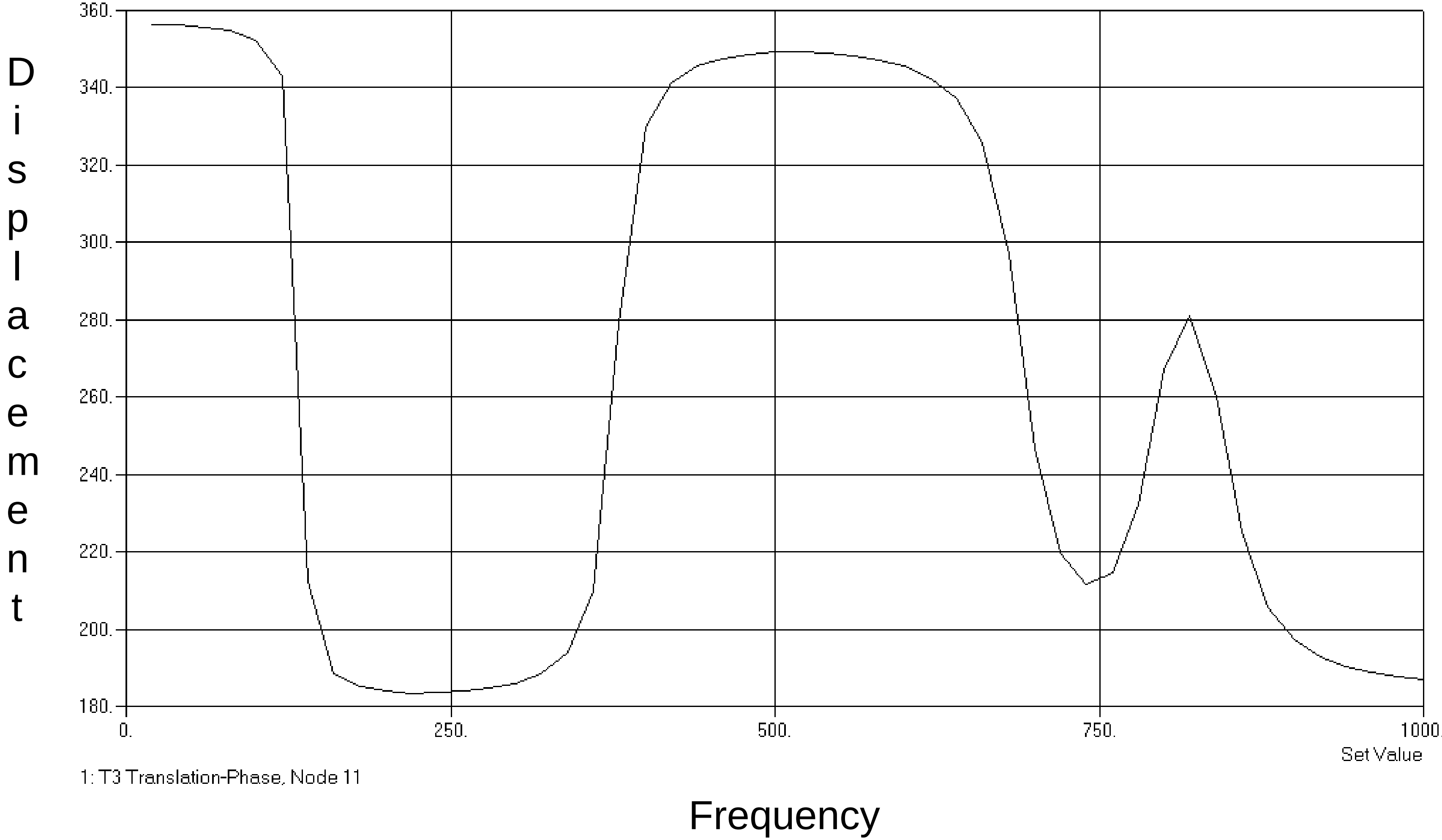
Problem #5: Direct Frequency Response

T3 Translation for Node 11 (Magnitude) over complete duration of analysis (Y-axis log scale)



Problem #5: Direct Frequency Response

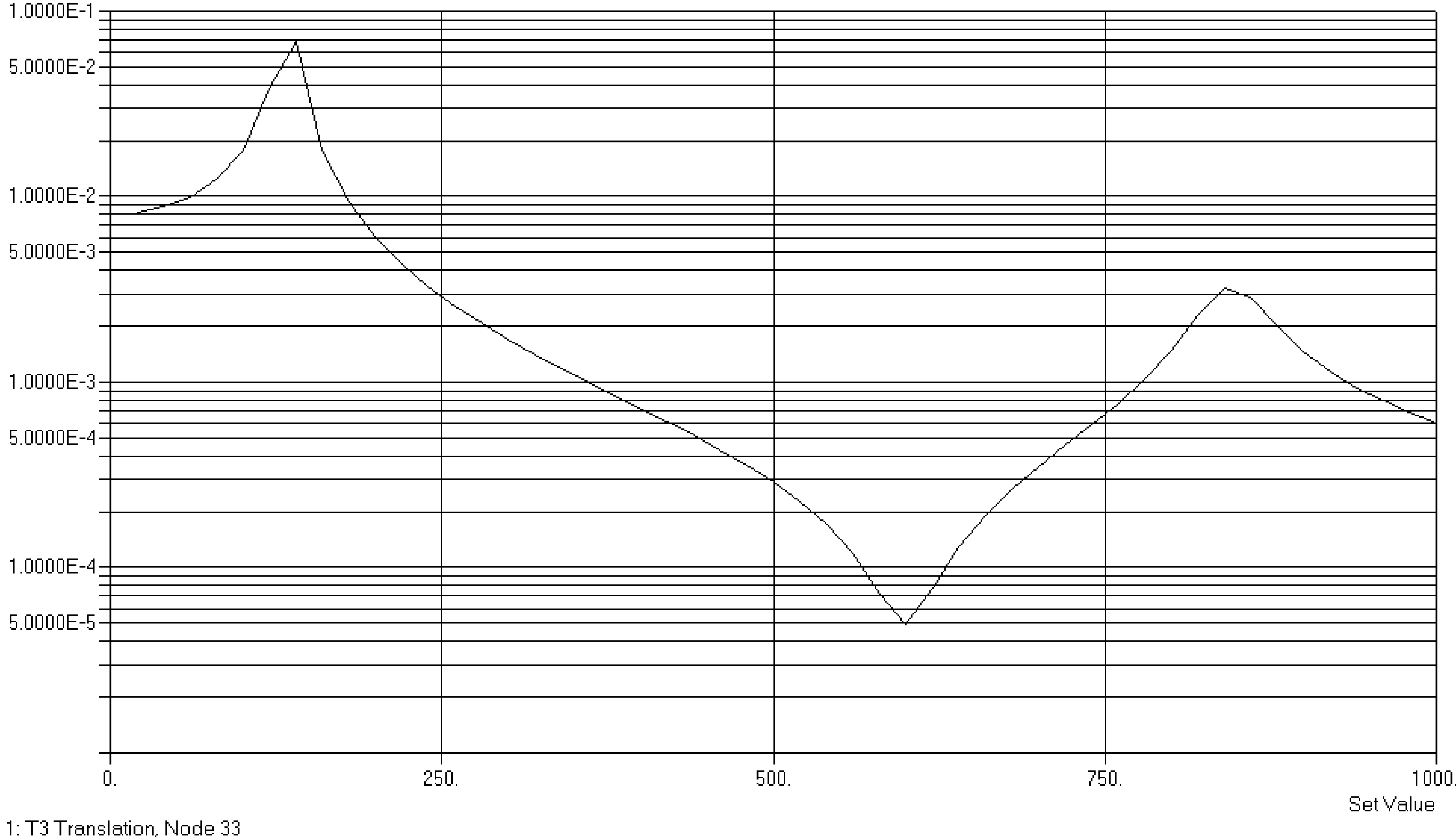
T3 Translation for Node 11 (Phase) over complete duration of analysis (Rectilinear graph)



Problem #5: Direct Frequency Response

T3 Translation for Node 33 (Magnitude) over complete duration of analysis (Y-axis log scale)

Displacement

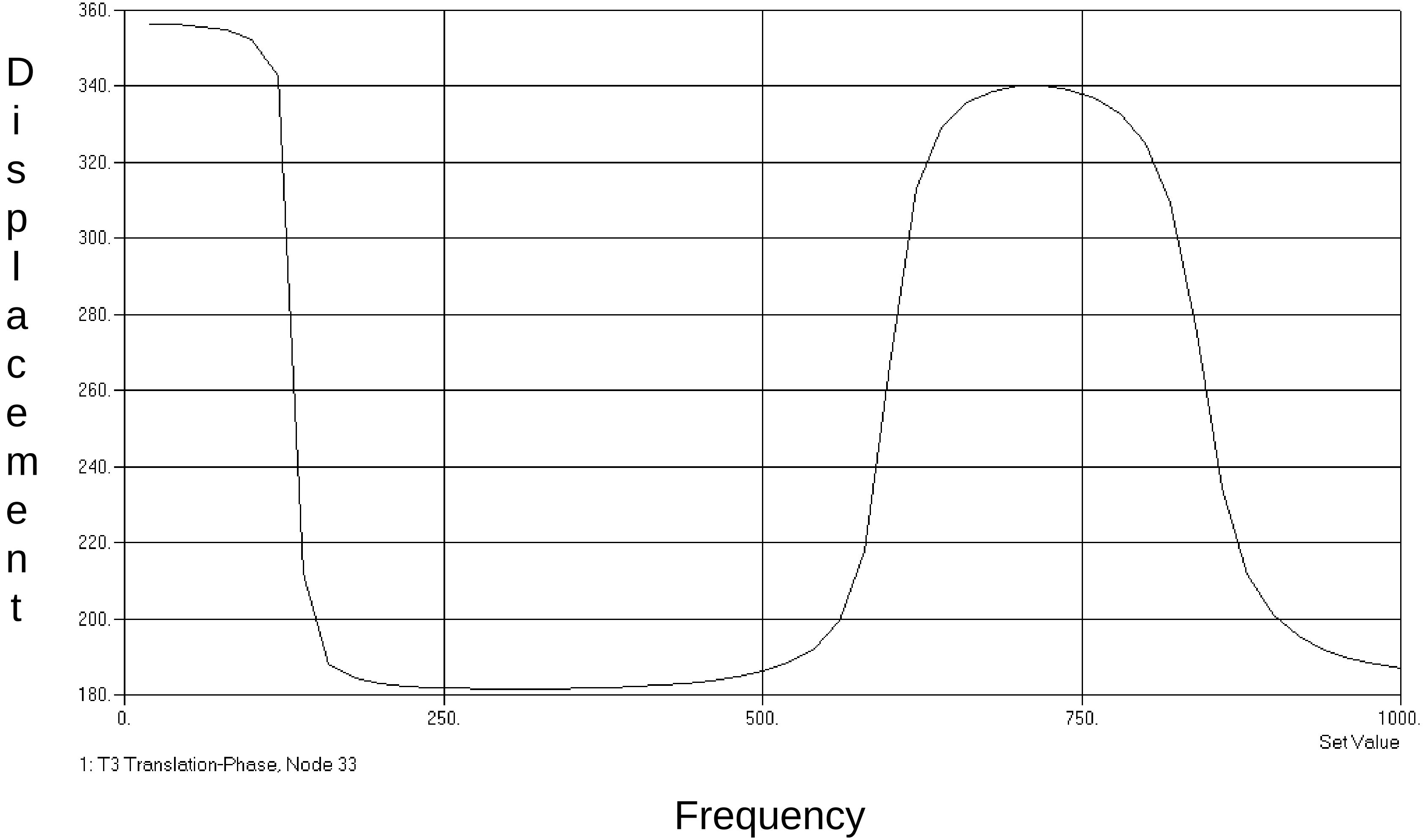


Frequency

Finite Element Analysis Predictive Engineering

Problem #5: Direct Frequency Response

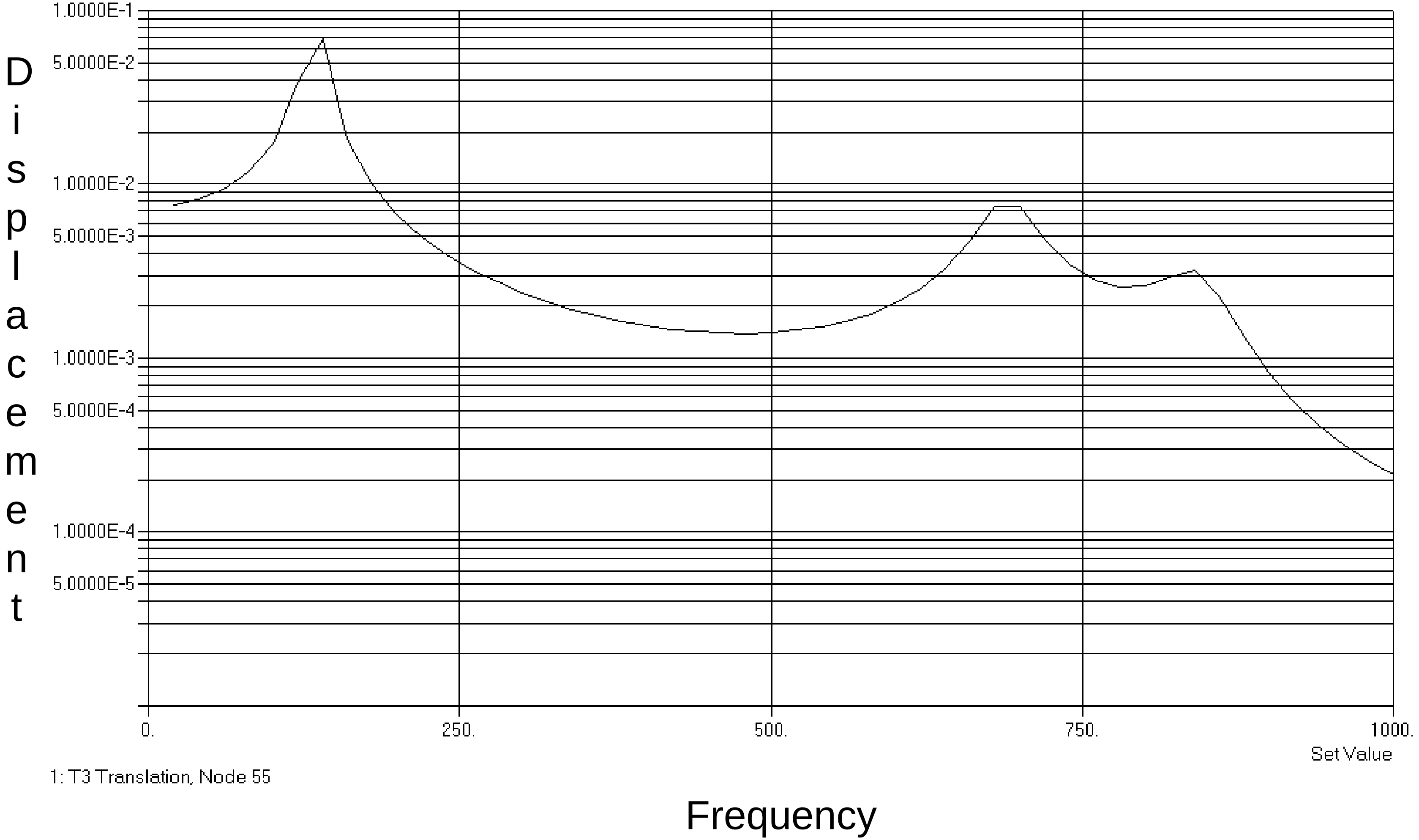
T3 Translation for Node 33 (Phase) over complete duration of analysis (Rectilinear graph)



FINITE ELEMENT ANALYSIS Predictive Engineering

Problem #5: Direct Frequency Response

T3 Translation for Node 55 (Magnitude) over complete duration of analysis (Y-axis log scale)

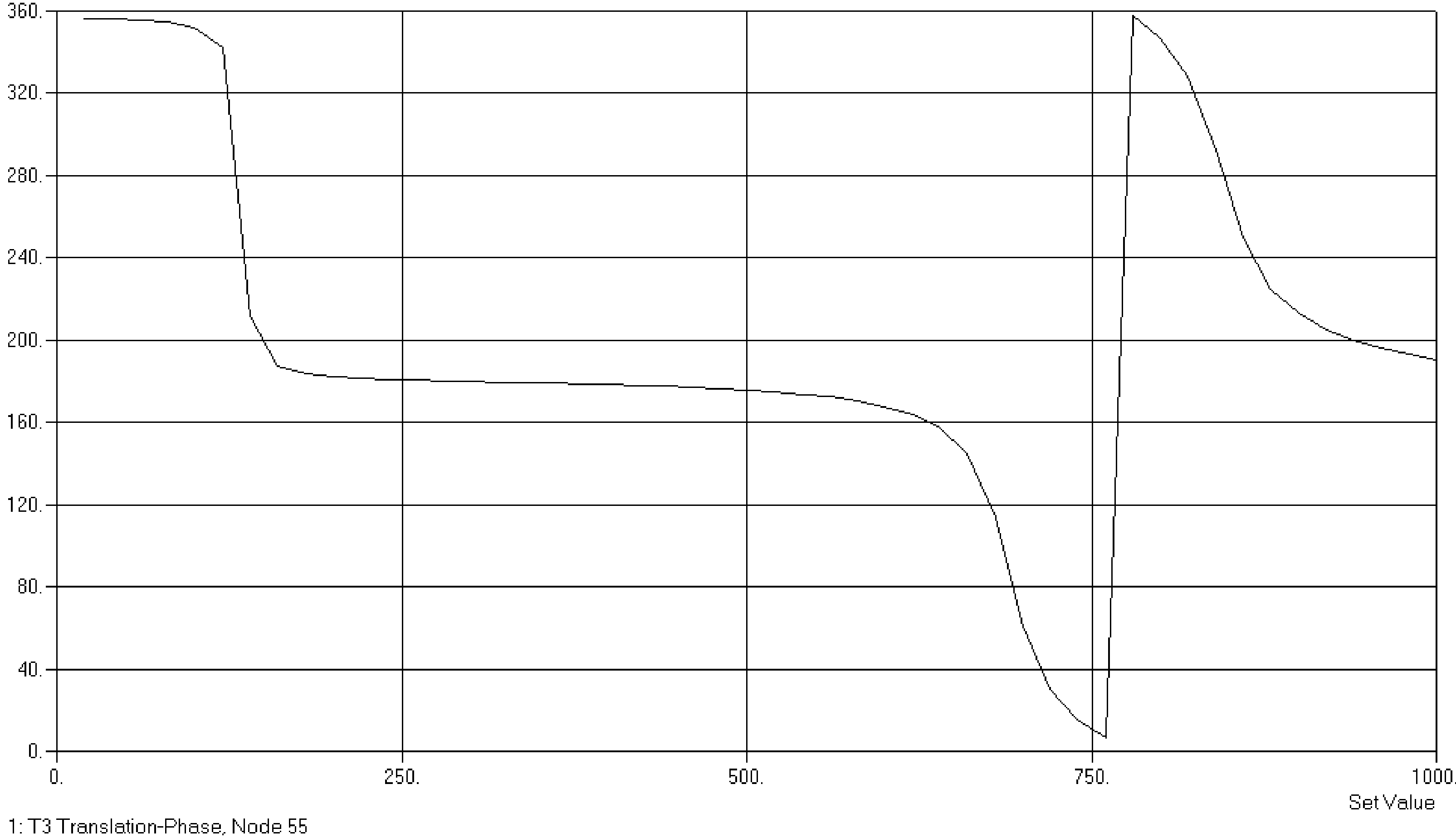


Finite Element Analysis
Predictive Engineering

Problem #5: Direct Frequency Response

T3 Translation for Node 55 (Phase) over complete duration of analysis (Rectilinear graph)

Displacement



Frequency

Problem #5: Direct Frequency Response

Use these results for comparison (Magnitude):

Frequency 140	T3 Displacement
Node 11	0.067867 in
Node 33	0.068504 in
Node 55	0.068888 in

Frequency 480	T3 Displacement
Node 11	0.00076559 in
Node 33	0.0003570 in
Node 55	0.0013882 in

Frequency 680	T3 Displacement
Node 11	0.0078558 in
Node 33	0.00026242 in
Node 55	0.0074482 in

Problem #5: Direct Frequency Response

Use these results for comparison (Phase):

Frequency 140	T3 Displacement
Node 11	212.265
Node 33	211.938
Node 55	211.655

Frequency 480	T3 Displacement
Node 11	348.688
Node 33	184.776
Node 55	176.419

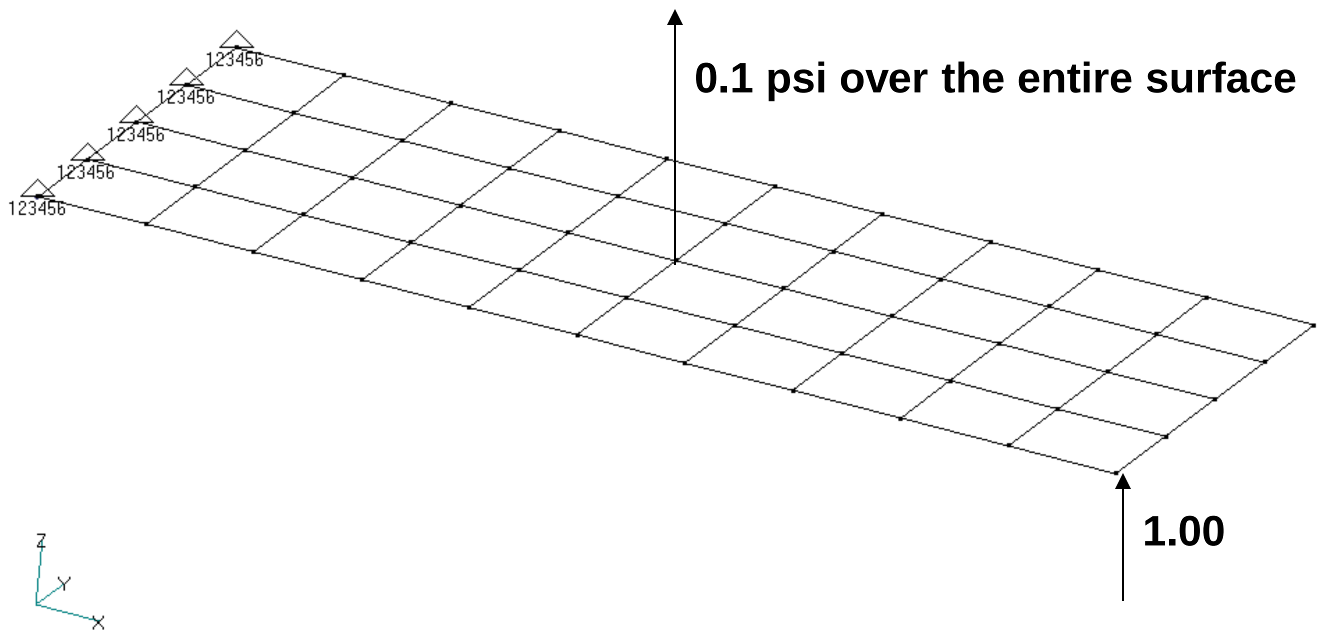
Frequency 680	T3 Displacement
Node 11	296.914
Node 33	338.782
Node 55	113.105

Problem #6

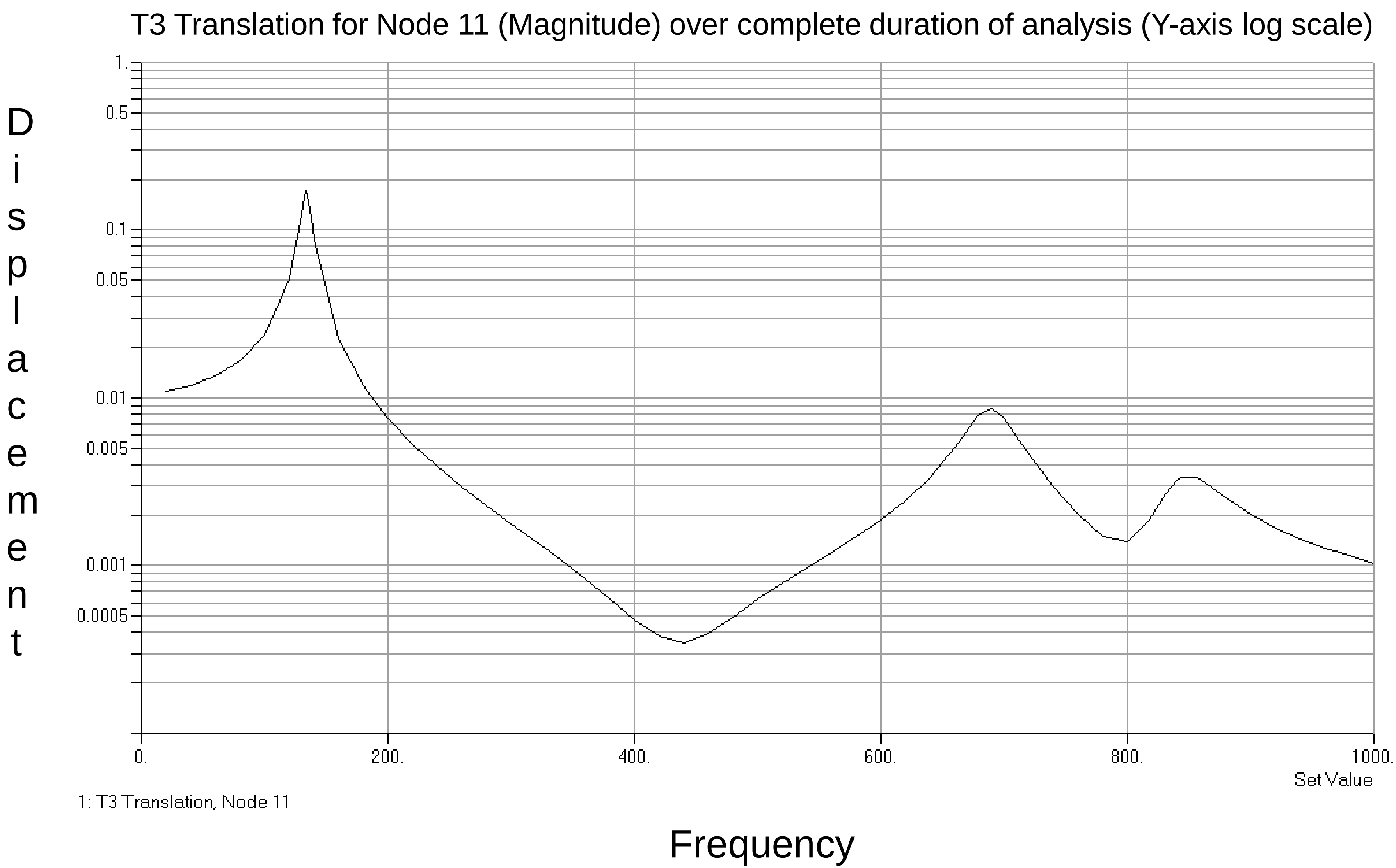
Modal Frequency Response

Problem #6: Modal Frequency Response

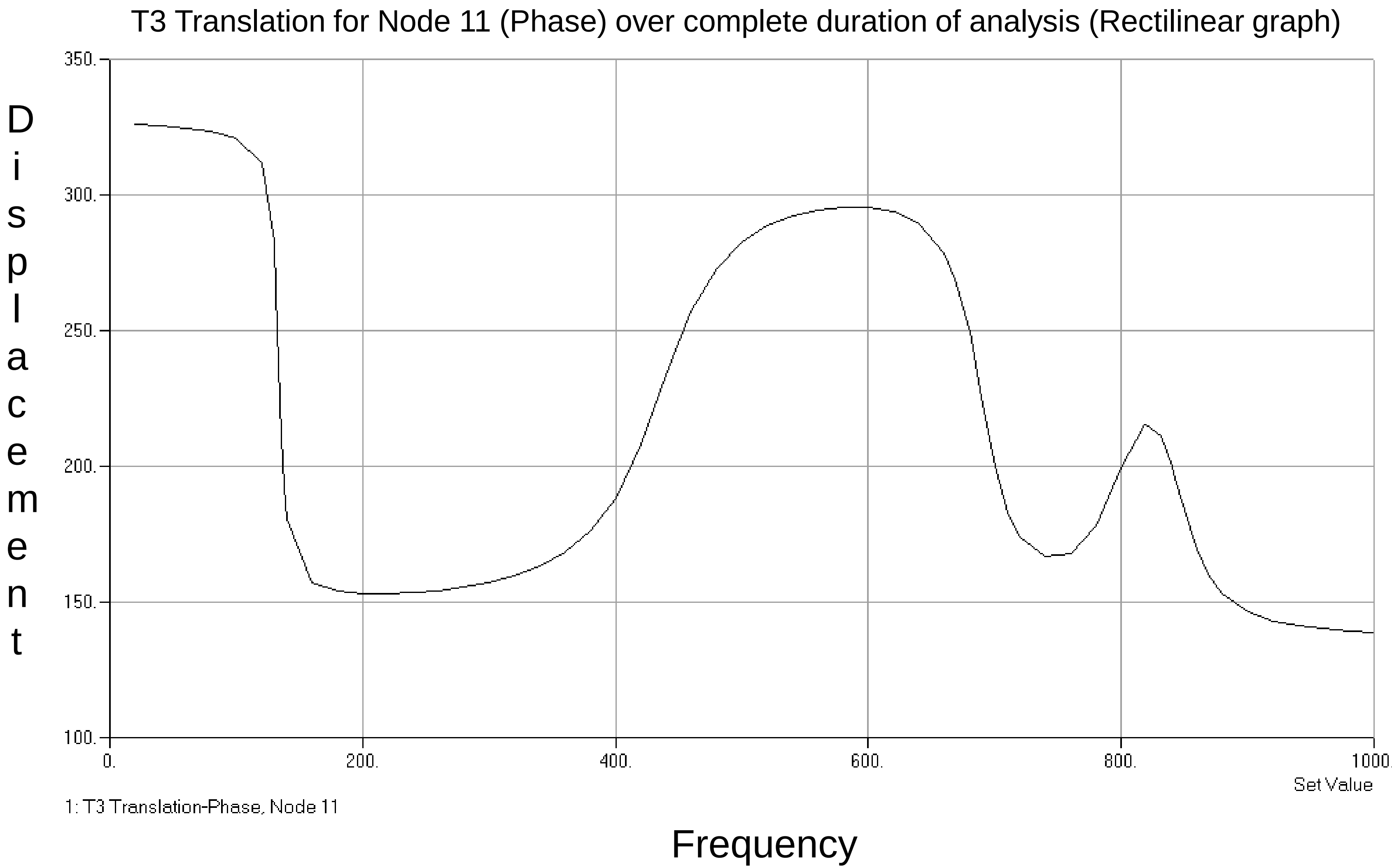
For this problem, use the modal method to determine the frequency response of the flat rectangular plate, created in problem #1, subject to a **0.1 psi pressure load** over the entire surface and a **unit load (1.0lb)** at the lower right corner (node **11**) lagging **45°**. Use a **modal damping of ξ = 0.03**. Use a **frequency step (Δf) of 20 Hz** between a **range of 20 and 1000Hz**. In addition, specify **five evenly spaced excitation frequencies between the half-power points of each resonant frequency between the range of 20 – 1000 Hz** (Modal Frequency table). Run modal analysis first to determine resonant frequencies between 20 -1000 Hz.



Problem #6: Modal Frequency Response



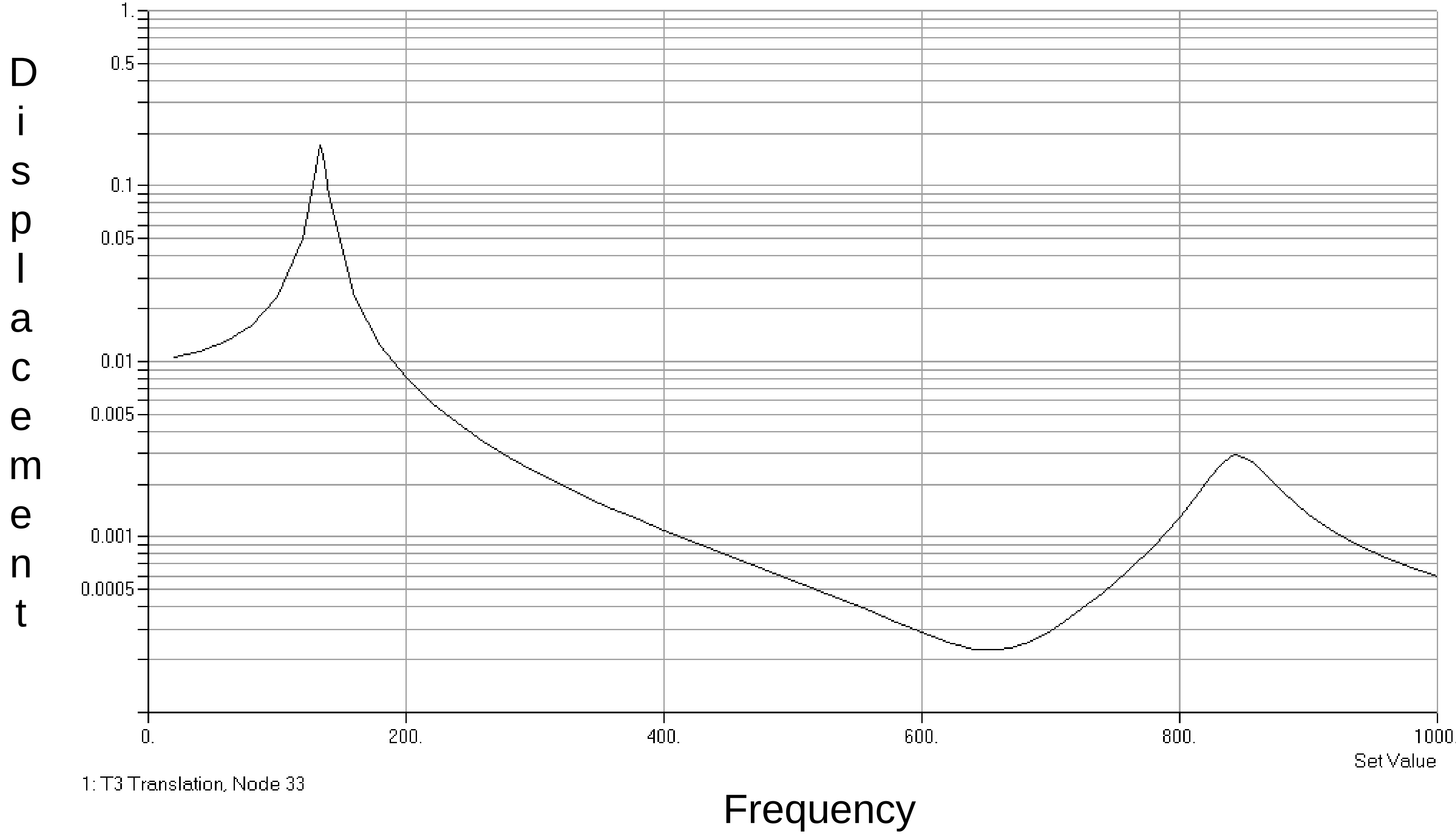
Problem #6: Modal Frequency Response



Finite Element Analysis Predictive Engineering

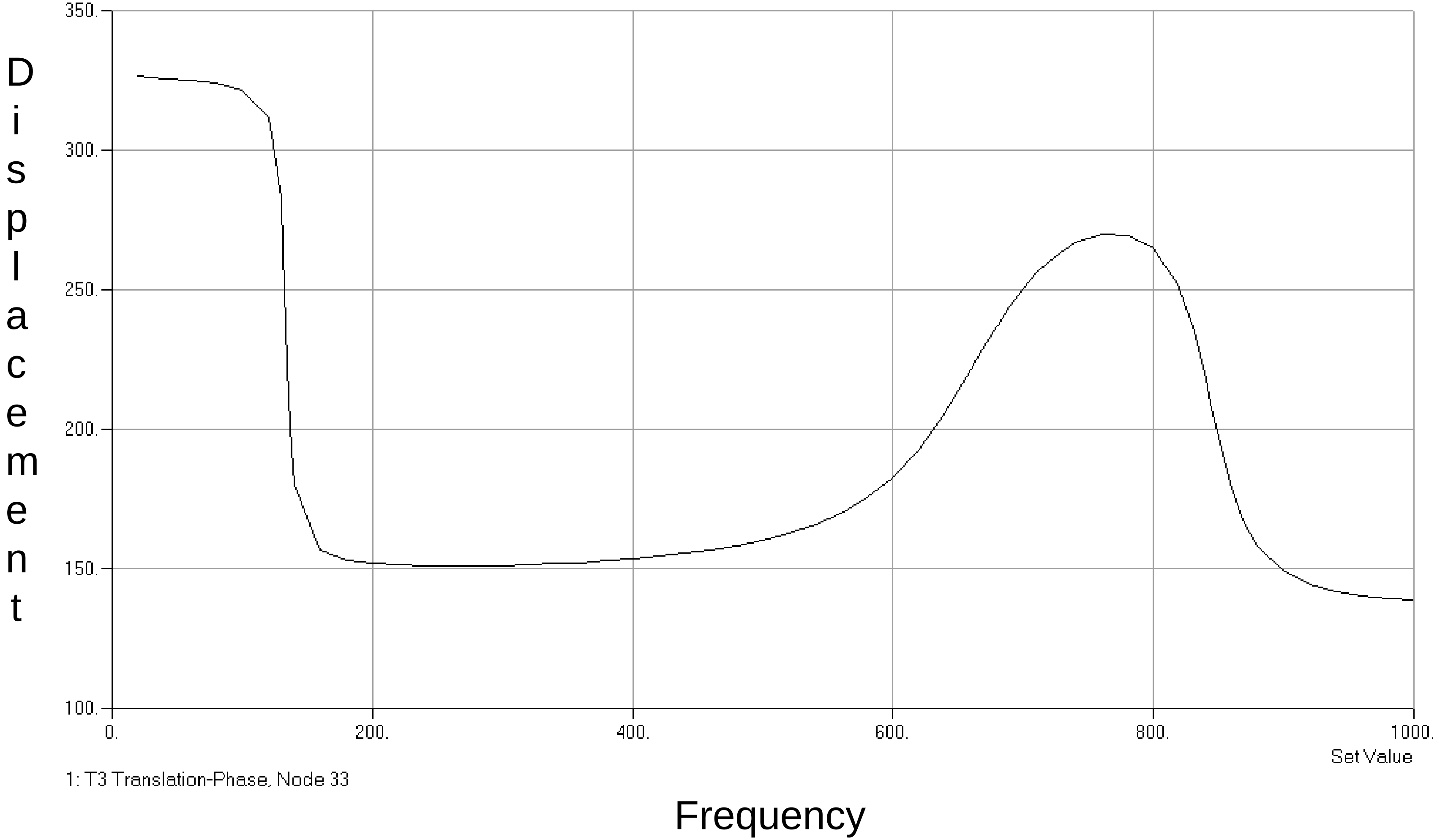
Problem #6: Modal Frequency Response

T3 Translation for Node 33 (Magnitude) over complete duration of analysis (Y-axis log scale)



Problem #6: Modal Frequency Response

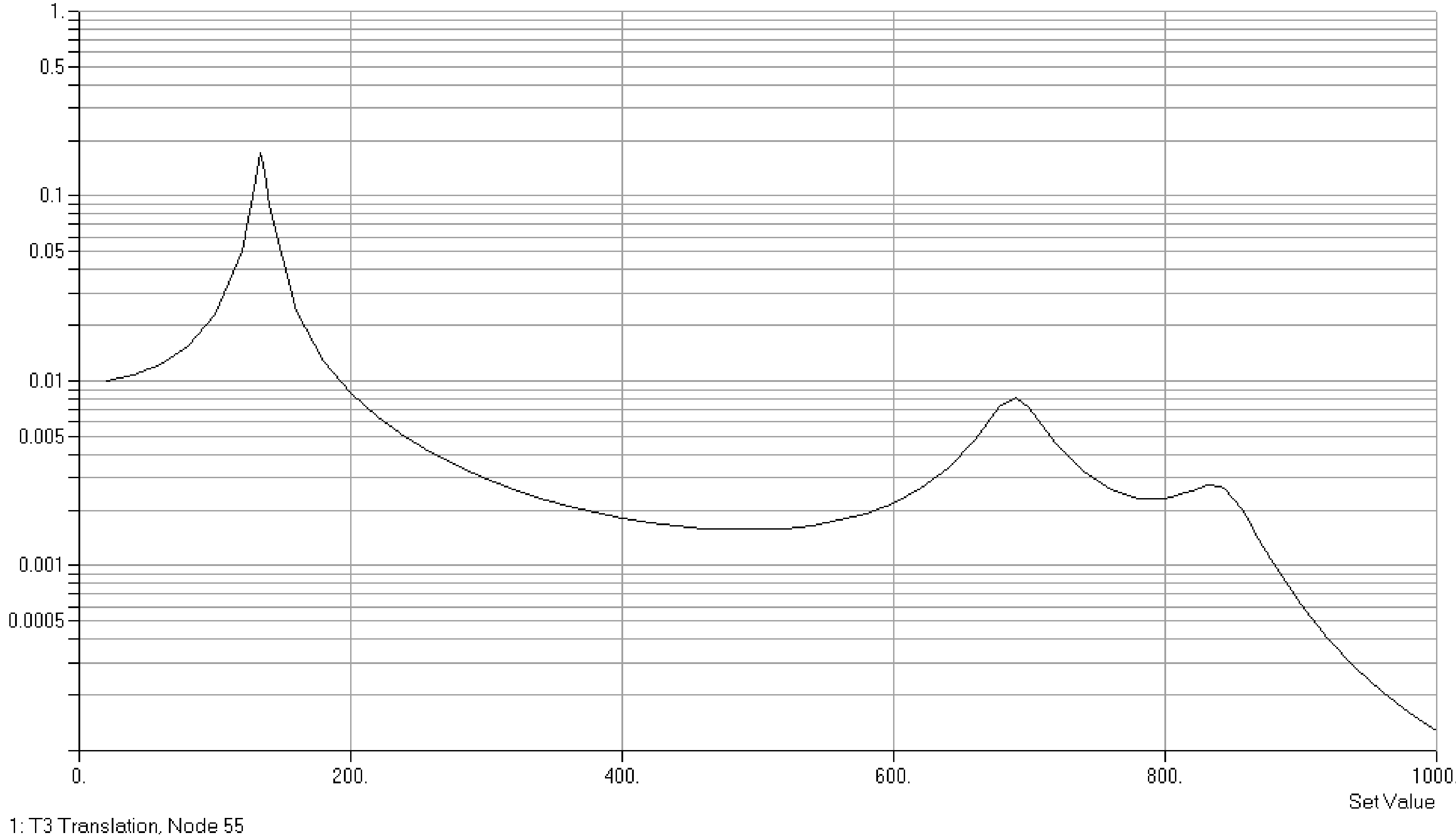
T3 Translation for Node 33 (Phase) over complete duration of analysis (Rectilinear graph)



Problem #6: Modal Frequency Response

T3 Translation for Node 55 (Magnitude) over complete duration of analysis (Y-axis log scale)

Displacement

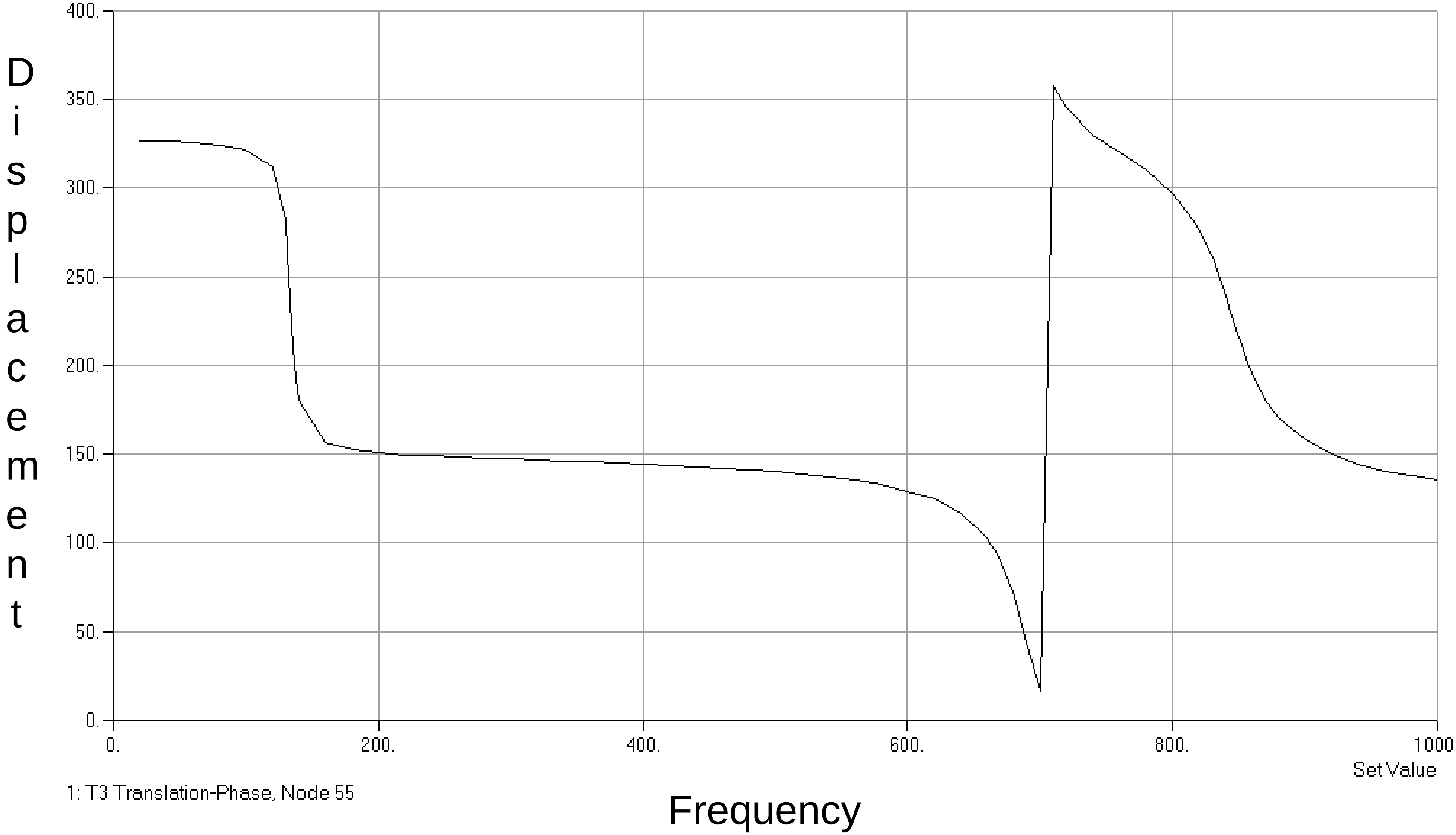


Frequency

Finite Element Analysis
Predictive Engineering

Problem #6: Modal Frequency Response

T3 Translation for Node 55 (Phase) over complete duration of analysis (Rectilinear graph)



Problem #6: Modal Frequency Response

Use these results for comparison (Magnitude):

Frequency 180	T3 Displacement
Node 11	0.011813 in
Node 33	0.012332 in
Node 55	0.012829 in

Frequency 440	T3 Displacement
Node 11	0.00034835 in
Node 33	0.00083596 in
Node 55	0.0016477 in

Frequency 720	T3 Displacement
Node 11	0.0045846 in
Node 33	0.00036933 in
Node 55	0.0045729 in

Problem #6: Modal Frequency Response

Use these results for comparison (Phase):

Frequency 180	T3 Displacement
Node 11	154.131
Node 33	153.283
Node 55	152.48

Frequency 440	T3 Displacement
Node 11	233.878
Node 33	155.416
Node 55	143.213

Frequency 720	T3 Displacement
Node 11	174.215
Node 33	260.31
Node 55	345.321

Enforced Motion

NX Nastran Dynamic Analysis

Introduction to Enforced Motion

- Used to analyze constrained structures with base input acceleration, velocity, and displacement.

- Common examples include earthquakes (for transient analysis) and swept-sine shaker test simulation (for frequency response analysis).

- For many years, no automatic method existed in Nastran for applying displacements, velocities, or accelerations to the “base”. At that time, there was a need to convert applied forces on equivalent unconstrained structure to enforced motion of constrained structure. Now accelerations, velocities, and displacements can be directly applied to the base node in what is known as the “Direct Method”. The Direct Method will be explained in a later chapter.

- Several methods of Enforced Motion exist: large mass and large stiffness.

Enforced Motion in Transient Response (Large Mass and Large Stiffness methods)

- The utilization of the Large Mass or Large Stiffness methods was necessary to specify acceleration, velocity, or displacement for dynamic analysis that involved enforced

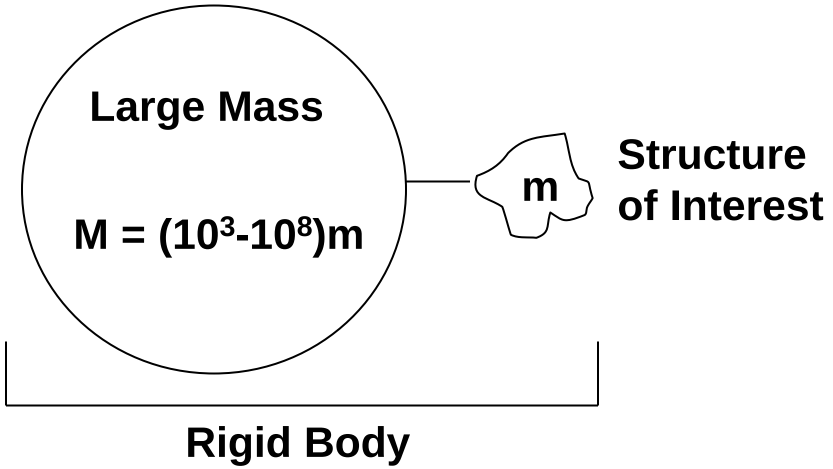
- For the indirect method, Nastran can only apply forces {P(t)} to the structure. If enforced motion is selected, for this section, it is assumed that the user is imposing the motion on a “large mass”. Therefore, the force to move the large mass is proportional to acceleration.

$$F = m\ddot{u}$$

or

$$\ddot{u} = \frac{F}{m}$$

Using "Large Mass" Method in Transient Response



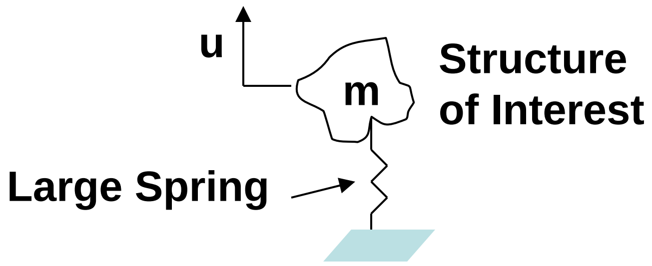
- For enforced acceleration $\ddot{u}_b$

$$\ddot{u}_b \approx \frac{1}{M_L} P \qquad M_L = \text{large mass}$$

- Larger mass should be 10^3 to 10^8 times the "Structure of Interest" mass

FINITE ELEMENT ANALYSIS
Predictive Engineering

Using "Large Stiffness" Method in Transient Response



For enforced motion displacement u_b

$$u_b = \frac{1}{K_L} P$$

K_L = large spring

- Apply P to spring/structure interface to obtain desired u_b. Spring K_L is entered on spring entries. Scale factor K_L is entered on force entries.
- The large spring should be 10^2 to 10^4 times $\omega_c{}^2 m$ where ω_c is the cutoff frequency of the excitation.

Using “Large Stiffness” Method in Transient Response

- Advantages in special cases: Avoids round-off errors in differentiation and avoids “rigid body drift” when enforced motion is applied to statically indeterminate points.

- Disadvantages in all cases: difficult to estimate a good value for large stiffness, and the required modes (in a modal formulation) are high frequency ones that are not likely to be included in retained modes.

Enforced Motion in Frequency Response (Large Mass and Large Stiffness methods)

- The utilization of the Large Mass or Large Stiffness methods was necessary to specify acceleration, velocity, or displacement for dynamic analysis that involved enforced motion.
- For these methods, Nastran can only apply forces {P(t)} to the structure. If enforced motion is selected, for this section, it is assumed that the user is imposing the motion on a “large mass” unless otherwise stated. Therefore, the force to move the large mass is proportional to acceleration.
- The easiest method to use to define enforced motion with Frequency Response is the Large Mass method

Using “Large Mass” Method in Frequency Response

- In frequency response the input and response are assumed to be sinusoidal functions

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

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

resulting in a simplified dynamic equation of motion:

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

Using “Large Mass” Method in Frequency Response

- As in transient response, the force required to move a “large mass” is:

$$P = ma$$

but the acceleration is assumed to be a sinusoidal function:

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

Therefore, to impose an acceleration, apply:

$$P(\omega) = ma(\omega)$$

Using “Large Mass” Method in Frequency Response

- To impose displacement:

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

A force needs to be applied that results in the desired displacement. Differentiating the displacement twice produces an acceleration:

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

The applied force is:

$$P(\omega) = ma(\omega) = -m\omega^2 u(\omega)$$

Using "Large Stiffness" Method in Frequency Response

- Using the Large Stiffness method is similar to the large mass method except the applied force is:

$$P = Ku$$

- To impose a displacement:

$$P(\omega) = Ku(\omega)$$

- To impose an acceleration:

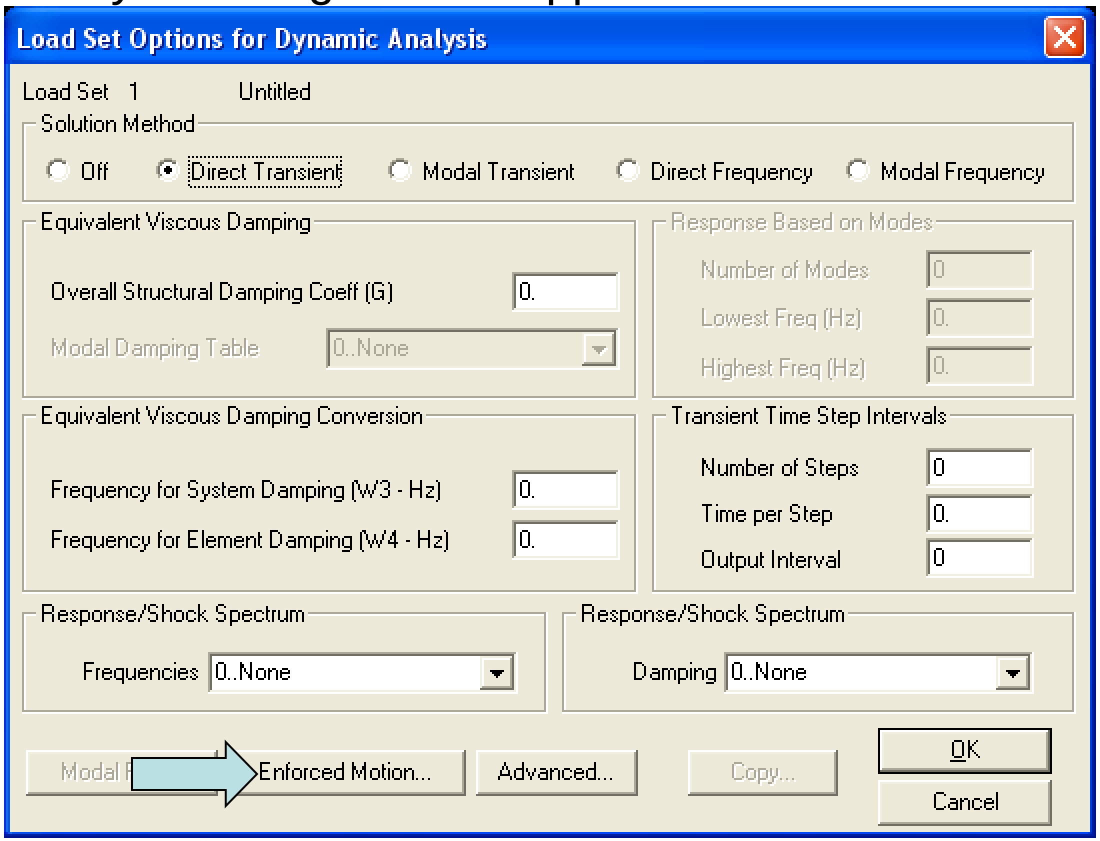
$$P(\omega) = \frac{1}{-\omega^2} Ka(\omega)$$

Recommendations for Enforced Motion (Large Mass and Large Stiffness methods)

- Use Large Mass method.
- Large mass should be at least 10^3 times structure mass for accuracy, but no more than 10^8 times structure mass (any higher causes numerical errors).
- Retain rigid body modes for analysis.
- Be careful with units – many times enforced acceleration is specified in terms of g (acceleration constant) rather than in direct units (such as in/sec^2).
- Use a small model to verify solution procedure.

Specifying Enforced Motion (Large Mass Method)

- Using the Model-> Load->Dynamic Analysis command to choose the **Enforced Motion** button. This will guide the user through creating enforced motion via the Large Mass method. The Load Set Options for Dynamic Analysis dialog box will appear:



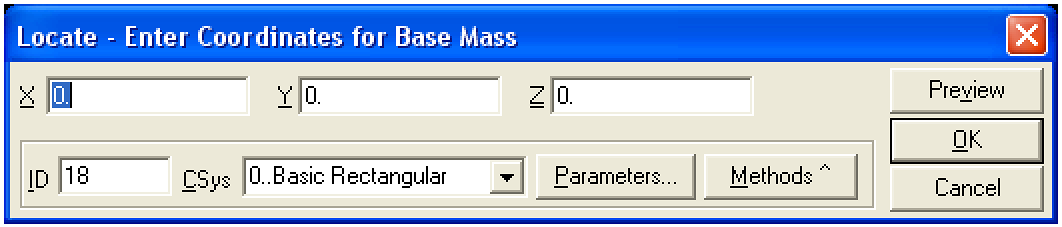
Click the Enforced Motion button after selected the desired Solution Method at the top of the dialog box.

The other parameters can be filled in after the enforced motion process has been completed.

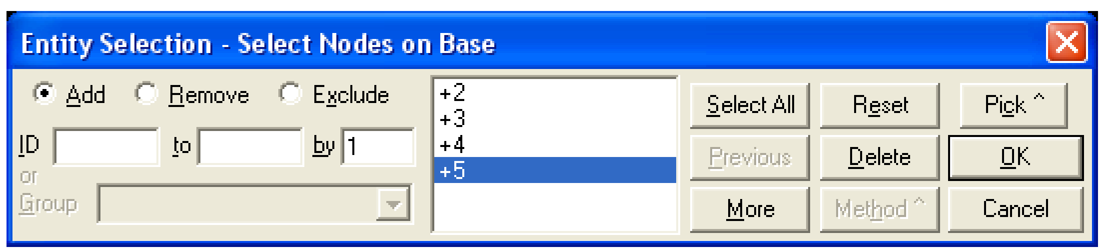
Finite Element Analysis
Predictive Engineering

Specifying Enforced Motion (Large Mass Method)

Select Coordinates for Base Mass. Any point in 3-D space can be chosen and existing node or point potions may be used as well. Click OK

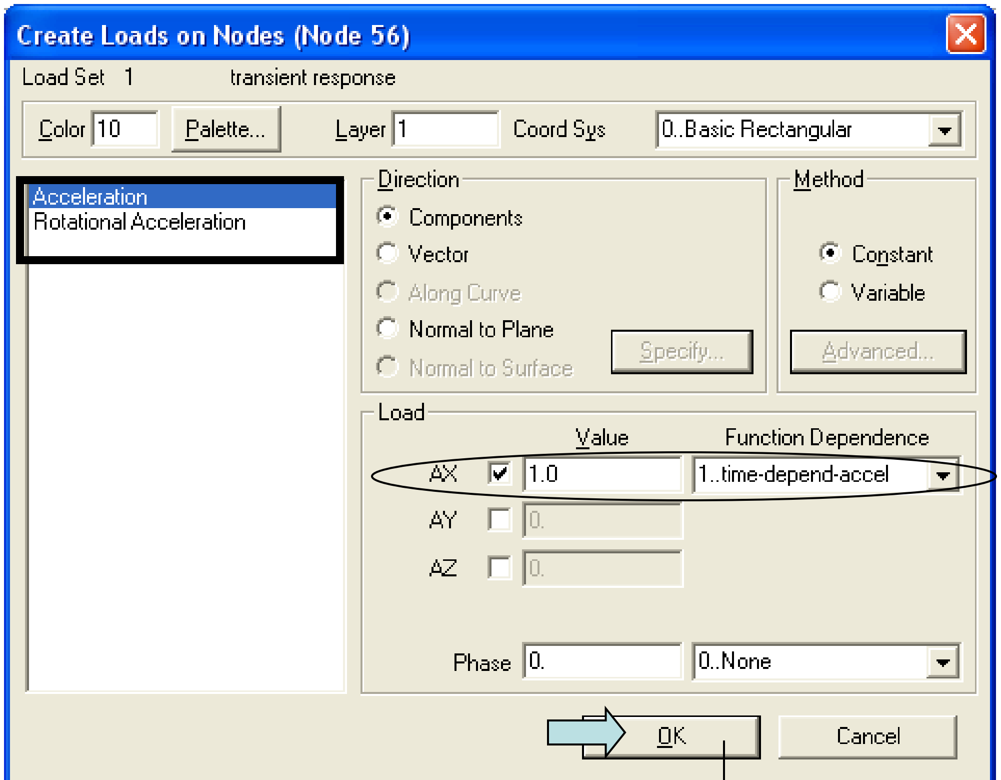


Select Nodes on the Base. At least one node must be chosen to connect the structure to the Base Mass. Once these nodes are chosen, FEMAP will automatically create a rigid element from the newly created node at the Base Mass location (independent node) to the nodes chosen to represent the base of the structure (dependent nodes). An example of proper selection of “Base Nodes” would be the points at the bottom of a High-Voltage tower that attach the tower to the ground or its foundation. Click OK



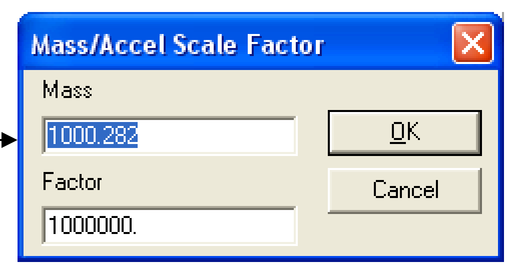
Specifying Enforced Motion (Large Mass Method)

Select acceleration or rotational acceleration from the list of available loads. Give a value for the selected load in the forcing direction, and a forcing function.



With Large Mass Method, this acceleration will be applied as a force which is exciting the base mass (a = F/M). Click OK.

The mass size and mass factor will be selected automatically in the next dialog box, but can also be altered for a customized enforced motion effect. Click OK



Enforced Motion in Transient Response (Direct method)

- While enforced motion with Nastran was accomplished for years by either the Large Mass approach or the Lagrange Multipliers technique, direct enforced motion capabilities are available in all versions of NX Nastran.

- Both the Large Mass and Lagrange Multipliers methods are both theoretically valid, they can also be very cumbersome to implement.

- Disadvantages:
  - Large Mass Method – often leads to computational and numerical problems due to round off errors and pseudo-rigid body modes.

  - Lagrange Multipliers Method – require a version specific DMAP ALTER command.

Enforced Motion in Transient Response (Direct method)

- The Direct method allows for direct specification of displacements, velocities, or accelerations using SPC/SPC1/SPCD data, eliminating the need to employ Large Mass or Lagrange Multipliers.

- NX Nastran directly utilizes this enforced motion information in the equations of motion, partitioning and integrating them (i.e., transient analysis) in accordance with the type of motion specified.

- Direct enforced motion is available in direct and modal frequency analysis (Solutions 108 and 111), direct and modal transient analysis (Solutions 109 and 112), and design optimization (Solution 200)

- Field 3 of the TLOADi and RLOADi cards has been changed from DAREA to EXCITEID.

Direct Method in Frequency Response

- In frequency response the transient effects are assumed to be negligible, and the time-dependent nature of the loading can be expressed in terms of harmonic forcing functions.
- Force response of the structure to these harmonic loads occurs at the same frequency, and in proportion to the magnitude of the applied loads.
- When an enforced motion is applied instead of a harmonic force, the effect is similar, creating a response with proportional forces of constraint at the same frequency as that of the enforcing motion.
- Any one of the enforced displacement, velocity, or acceleration must uniquely determine the other two (differ only by multiples of frequency), with resultant forces of constraint derived from a solution of governing equations.

Direct Method in Frequency Response

- To illustrate this, an applied harmonic forcing function of the form:

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

will lead to the in-plane displacement:

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

With the corresponding velocity and acceleration:

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

and

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

yielding the equations of frequency response.

Direct Method in Frequency Response

- Going into further detail, the frequency response equations are written after multipoint constraint partitioning operations have been performed leaving just the free (f-set) and constrained (s-set) degrees of freedom:

$$\left[-\omega^2 \begin{bmatrix} M_{ff} & M_{fs} \\ M_{sf} & M_{ss} \end{bmatrix} + i\omega \begin{bmatrix} B_{ff} & B_{fs} \\ B_{sf} & B_{ss} \end{bmatrix} + \begin{bmatrix} K_{ff} & K_{fs} \\ K_{sf} & K_{ss} \end{bmatrix} \right] \begin{Bmatrix} U_f \\ U_s \end{Bmatrix} = \begin{Bmatrix} P_f \\ P_s + q_s \end{Bmatrix} \quad \text{Eq. 9-1}$$

Where P_s are the external loads applied to the S-Set and q_s are the corresponding forces of constraint.

If the constraints specify zero motion ($U_s = \{0\}$), the solution for the free degrees-of-freedom may be obtained directly from the upper part of this equation:

$$(-\omega^2 M_{ff} + i\omega B_{ff} + K_{ff})\, U_f = P_f \qquad \text{Eq. 9-2}$$

And the corresponding constraint forces from the lower part:

$$q_s = -P_s + (-\omega^2 M_{sf} + i\omega B_{sf} + K_{sf})\, U_f \qquad \text{Eq. 9-3}$$

Direct Method in Frequency Response

If enforced displacements, velocities, or accelerations are applied, $U_s \neq \{0\}$ and the free degrees-of-freedom, from **Eq. 9-1** are:

$$(-\omega^2 M_{ff} + i\omega B_{ff} + K_{ff})\, U_f = P_f - (-\omega^2 M_{sf} + i\omega B_{sf} + K_{sf})\, U_s \quad \text{Eq. 9-4}$$

which corresponds with constraint forces:

$$q_s = -P_s + (-\omega^2 M_{sf} + i\omega B_{sf} + K_{sf})\, U_f + (-\omega_2 M_{ss} + i\omega B_{ss} + K_{ss})\, U_s \quad \text{Eq. 9-5}$$

Comparing 9-4 and 9-5 with 9-2 and 9-3 shows that the enforced motion modifies the force applied to the f-set degrees-of-freedom.

Had an enforced velocity been applied instead, the resultant applied displacement would differ by a factor of **(1/(iω)).**

Had it been an applied acceleration, it would differ by **–(1/ω²).**

Direct Method in Transient Response

- When the transient dynamic equations of motion are written in terms of displacement and its higher order derivatives, the solution of these equations must be performed in a stepwise integral fashion for very time step, t.

- As with Frequency response, any enforced displacement, velocity, or acceleration must uniquely define the other two quantities for that degree-of-freedom with resultant forces of constraint derived from a solution of governing equations of motion at that particular time step(s) of interest.

- Unlike Frequency response, higher-order displacement derivatives for the enforced degrees-of-freedom must be determined by finite difference, with lower-order quantities (for example, displacements and velocities for an applied acceleration) determined by numerical integration.

Direct Method in Frequency Response

- Going into further detail, the transient dynamic equations of motion are written after multipoint constraint partitioning operations have been performed leaving just the free (f-set) and constrained (s-set) degrees of freedom:

$$\begin{bmatrix} M_{ff} & M_{fs} \\ M_{sf} & M_{ss} \end{bmatrix} \begin{Bmatrix} \ddot{u}_f \\ \ddot{u}_s \end{Bmatrix} + \begin{bmatrix} B_{ff} & B_{fs} \\ B_{sf} & B_{ss} \end{bmatrix} \begin{Bmatrix} \dot{u}_f \\ \dot{u}_s \end{Bmatrix} + \begin{bmatrix} K_{ff} & K_{fs} \\ K_{sf} & K_{ss} \end{bmatrix} \begin{Bmatrix} u_f \\ u_s \end{Bmatrix} = \begin{Bmatrix} P_f(t) \\ P_s(t) + q_s(t) \end{Bmatrix} \quad \text{Eq. 9-6}$$

In the case of zero constrained motion, $u_s = u_s = u_s = \{0\}$ and the solution for the free degrees-of-freedom may be obtained directly from this equation:

$$M_{ff}\,\ddot{u}_f + B_{ff}\,\dot{u}_f + K_{ff}\,u_f = P_f(t) \qquad \text{Eq. 9-7}$$

with corresponding forces of constraint from:

$$q_s(t) = -P_s(t) + (M_{sf}\,\ddot{u}_f + B_{sf}\,\dot{u}_f + K_{sf}\,u_f) \qquad \text{Eq. 9-8}$$

Direct Method in Frequency Response

If enforced displacements, velocities, or accelerations are applied, u_s, $\dot{u}_s$, $\ddot{u}_s \neq \{0\}$, and the solution for the free degrees-of-freedom from **Eq. 9-6**, are:

$$M_{ff}\,\ddot{u}_f + B_{ff}\,\dot{u}_f + K_{ff}\,u_f = P_s(t) + (M_{sf}\,\ddot{u}_s + B_{sf}\,\dot{u}_s + K_{sf}\,u_s) \quad \text{Eq. 9-9}$$

with constraint forces:

$$q_s(t) = -P_s(t) + [M_{sf}\ \ M_{ss}]\begin{Bmatrix}\ddot{u}_f \\ \ddot{u}_s\end{Bmatrix} + [B_{sf}\ \ B_{ss}]\begin{Bmatrix}\dot{u}_f \\ \dot{u}_s\end{Bmatrix} + [K_{sf}\ \ K_{ss}]\begin{Bmatrix}u_f \\ u_s\end{Bmatrix} \quad \text{Eq. 9-10}$$

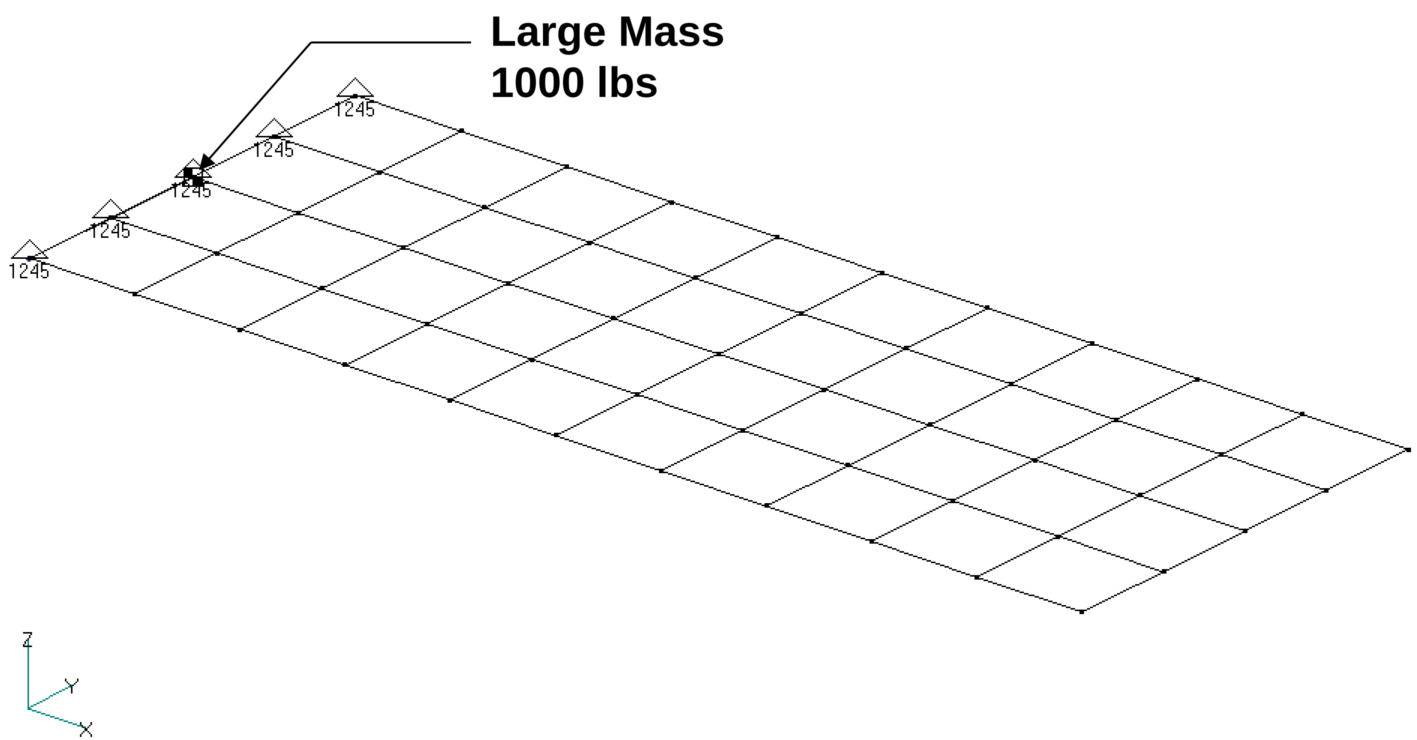
As with frequency response, the effect of enforced motion is to modify the loads on the f-set, and the s-set forces of constraint

Problem #7

Direct Transient Response with Enforced Acceleration

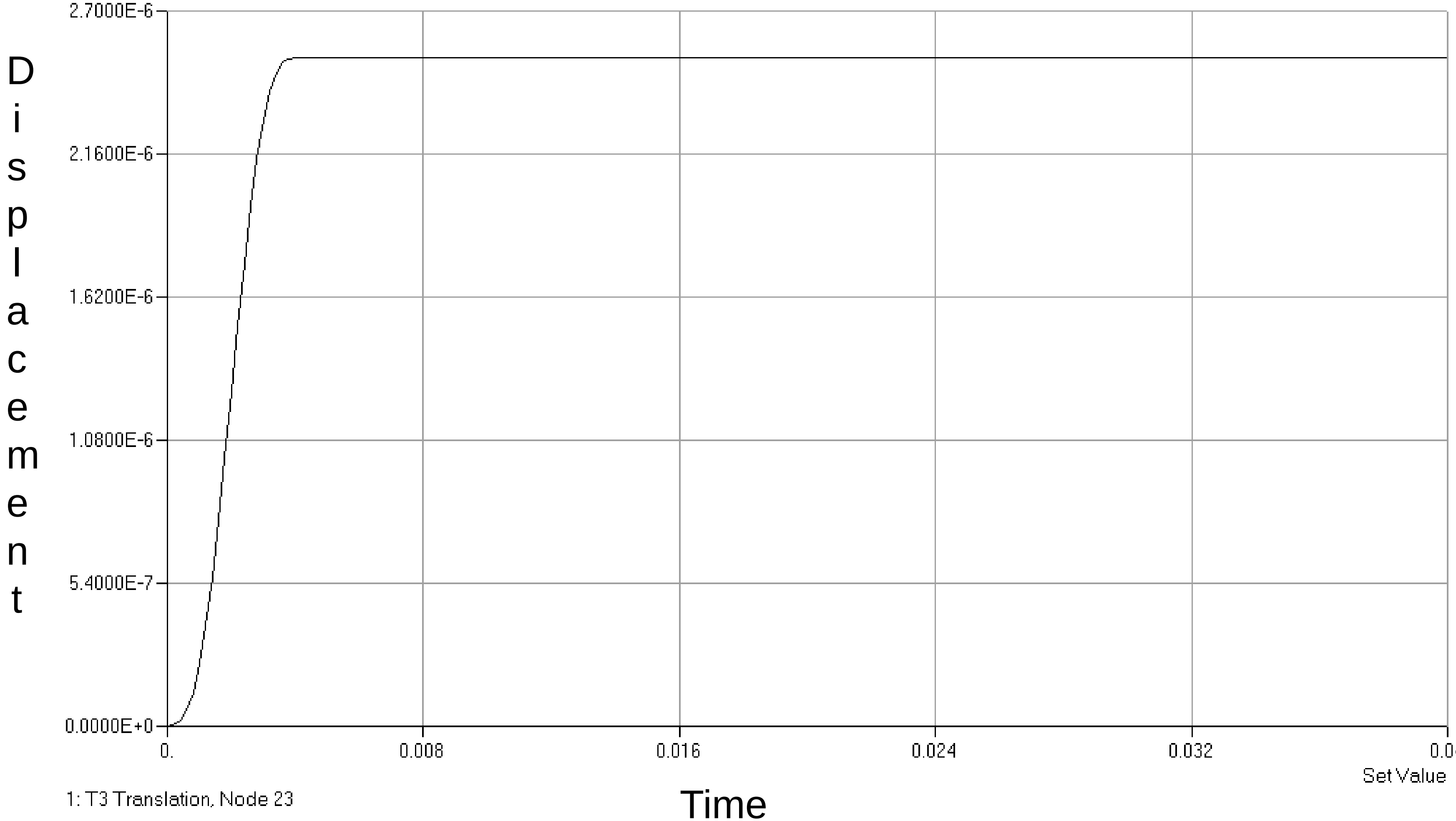
Problem #7: Direct Transient Response with Enforced Acceleration

For this problem, use the direct method to determine the transient response of the flat rectangular plate, created in problem #1, subject to a **unit acceleration sine pulse of 200 HZ** applied to the base (**node 23**) in the z-direction. A **large mass of 1000 lb** is applied to the base. Use a structural damping coefficient of **g = 0.05** and convert this damping to **equivalent viscous damping at 200 Hz**.



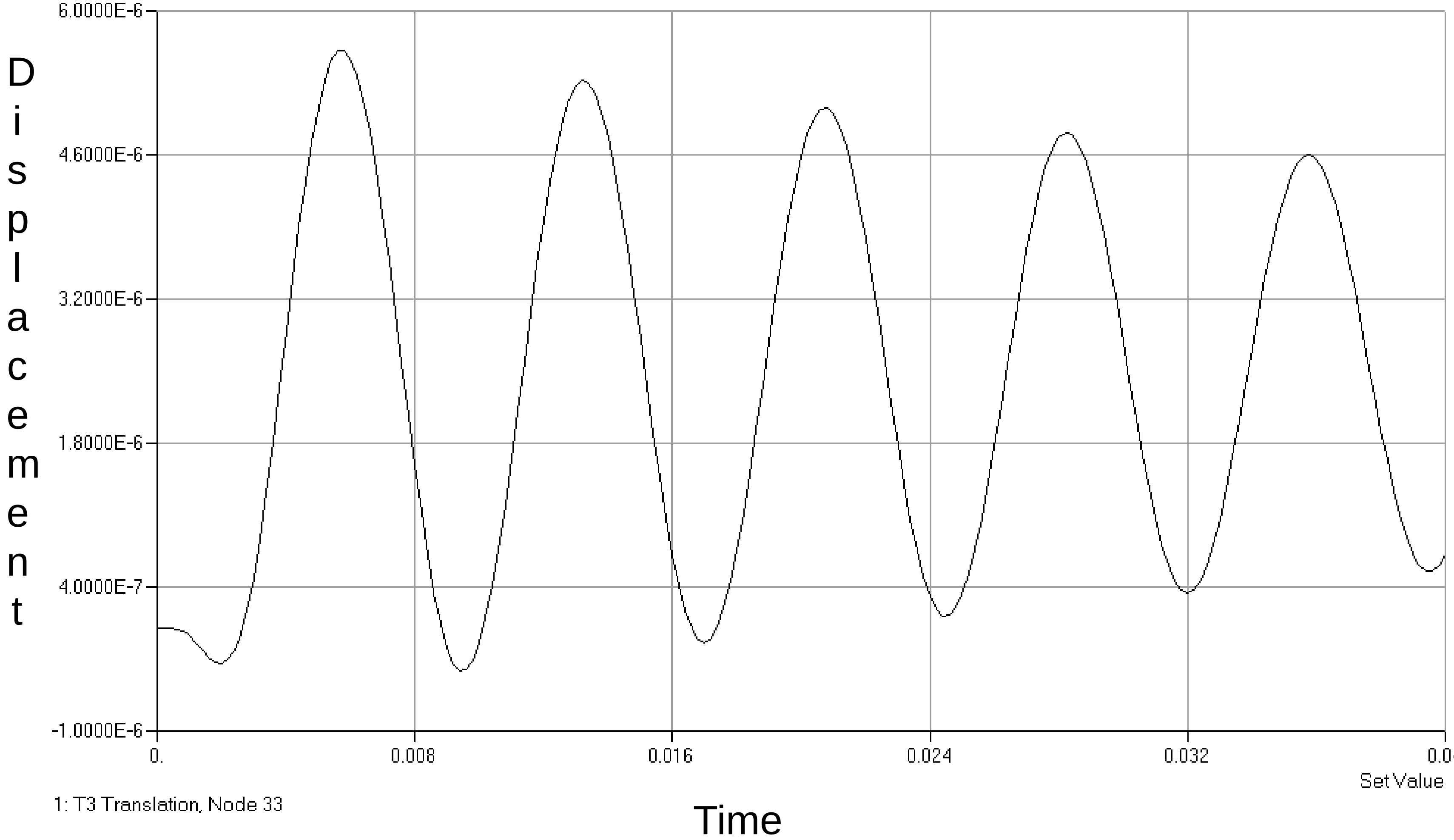
Problem #7: Displacement Results

T3 Translation for Node 23 (Base node) over duration of analysis



Problem #7: Displacement Results

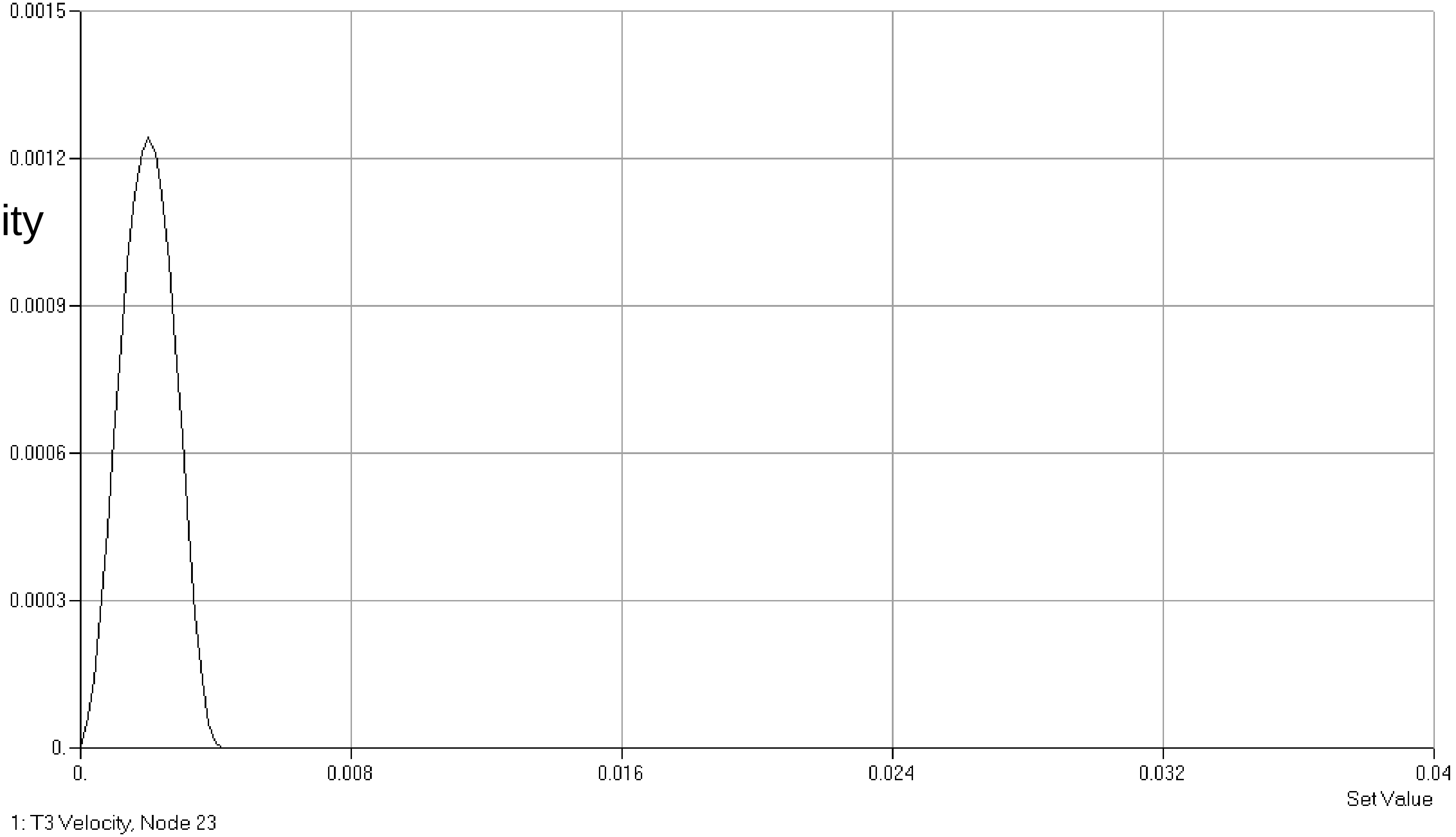
T3 Translation for Node 33 over complete duration of analysis



Problem #7: Velocity Results

T3 Velocity for Node 23 (Base node) over duration of analysis

Velocity

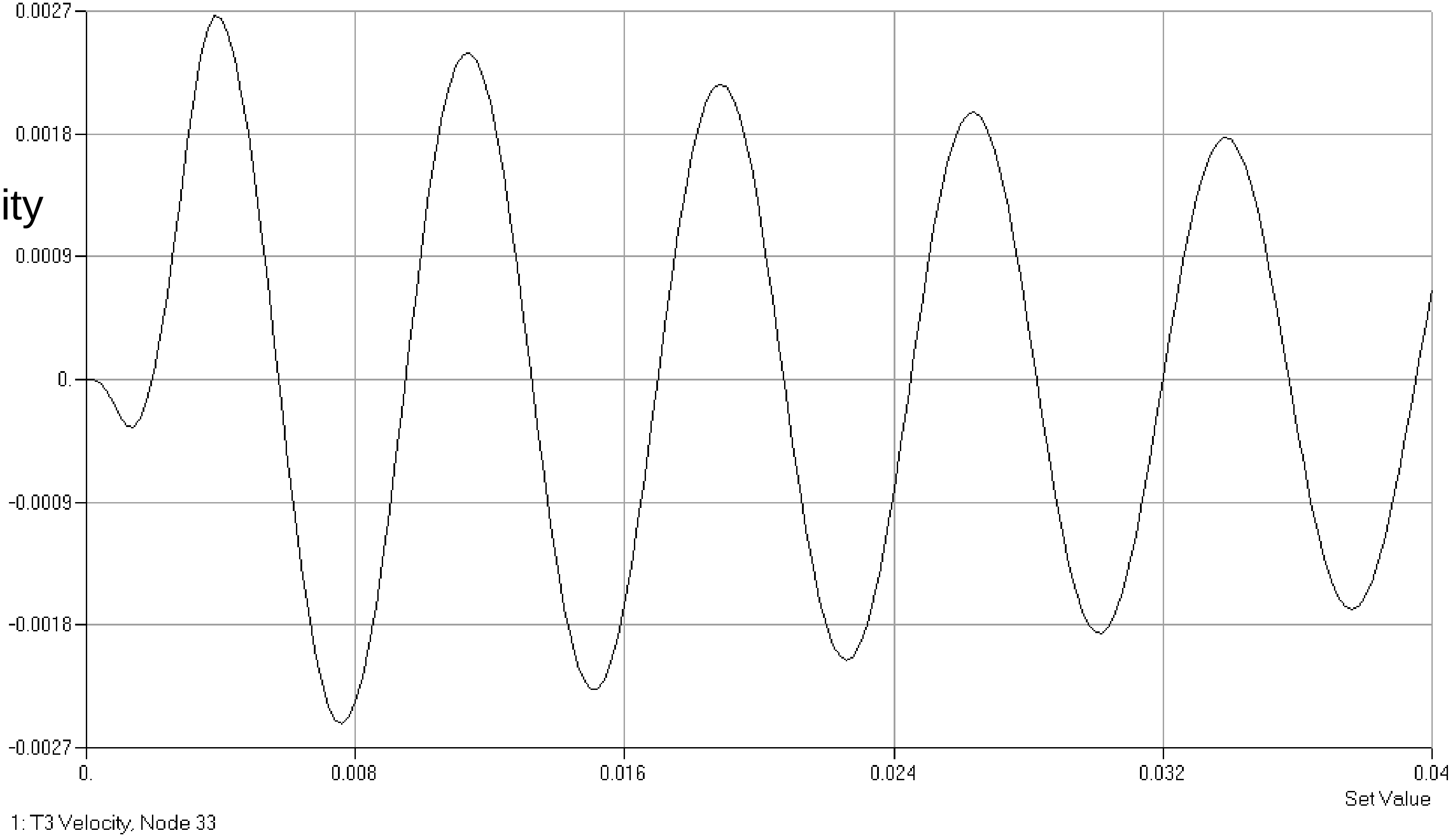


Time

Problem #7: Velocity Results

T3 Velocity for Node 33 over complete duration of analysis

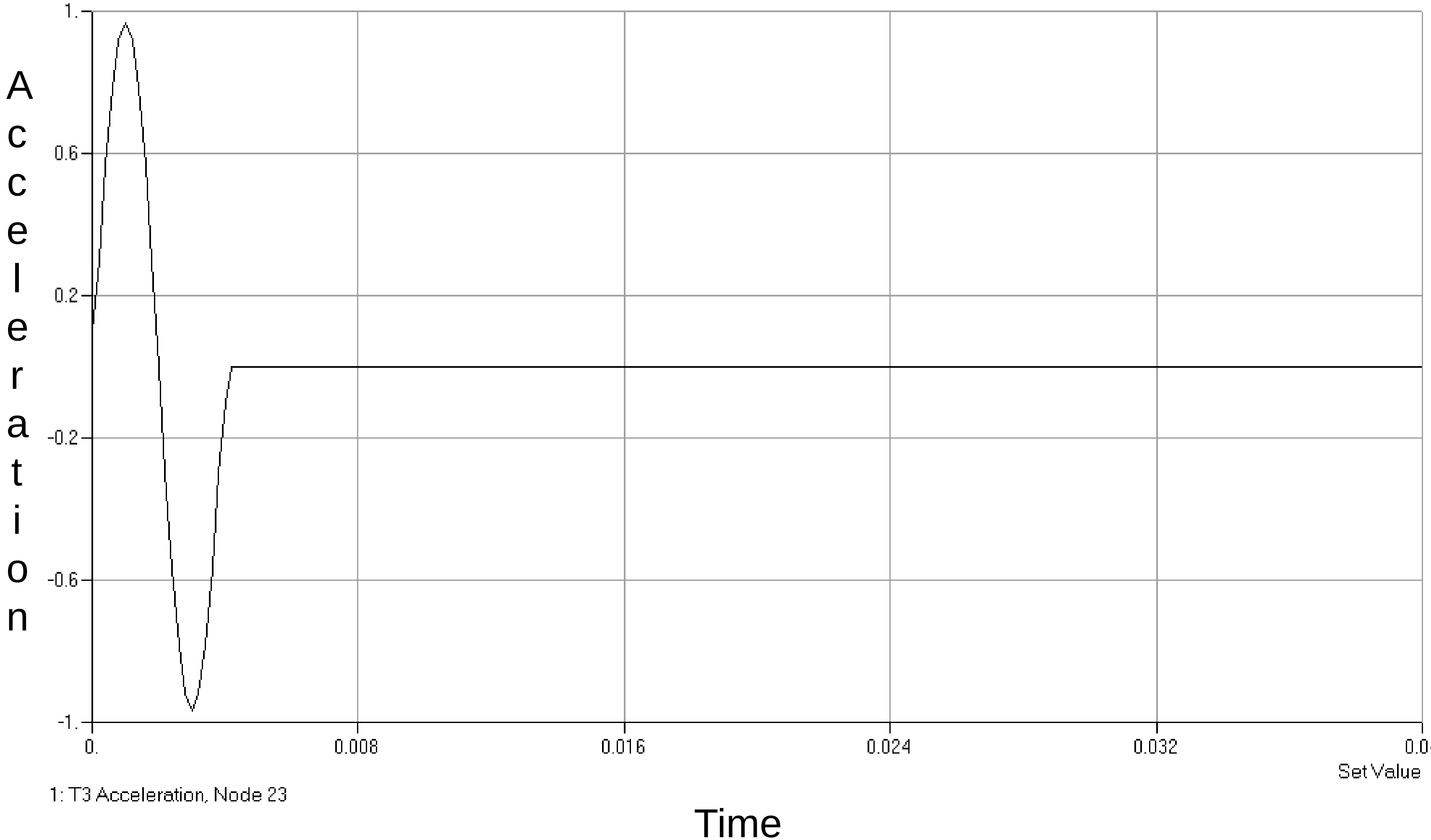
Velocity



Time

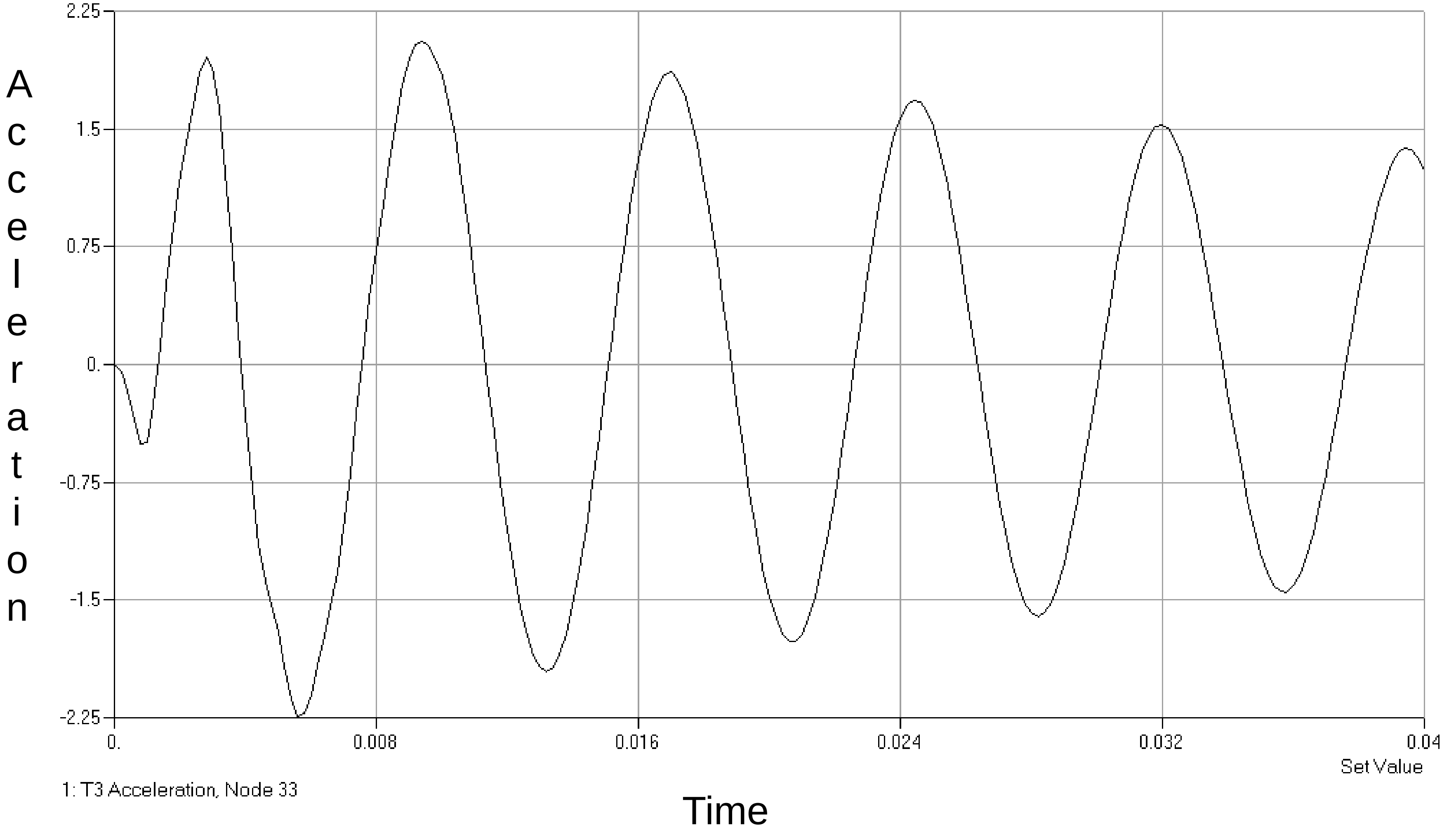
Problem #7: Acceleration Results

T3 Acceleration for Node 23 (Base node) over duration of analysis



Problem #7: Acceleration Results

T3 Acceleration for Node 33 over complete duration of analysis



Problem #7: Direct Transient with Enforced Motion

Use these results for comparison (Node 23):

Time	T3 Displacement
0	0.0 in
0.02	2.525E-6 in
0.04	2.525E-6 in

Time	T3 Velocity
0	0.0
0.02	-1.35554E-7
0.04	-7.25978E-8

Time	T3 Acceleration
0	0.103
0.02	0.00016233
0.04	-0.00013677

Problem #7: Direct Transient with Enforced Motion

Use these results for comparison (Node 33):

Time	T3 Displacement
0	0.0 in
0.02	4.59E-6 in
0.04	7.1984E-7 in

Time	T3 Velocity
0	0.0
0.02	0.0012328
0.04	0.00065997

Time	T3 Acceleration
0	-0.0043797
0.02	-1.47664
0.04	1.24428

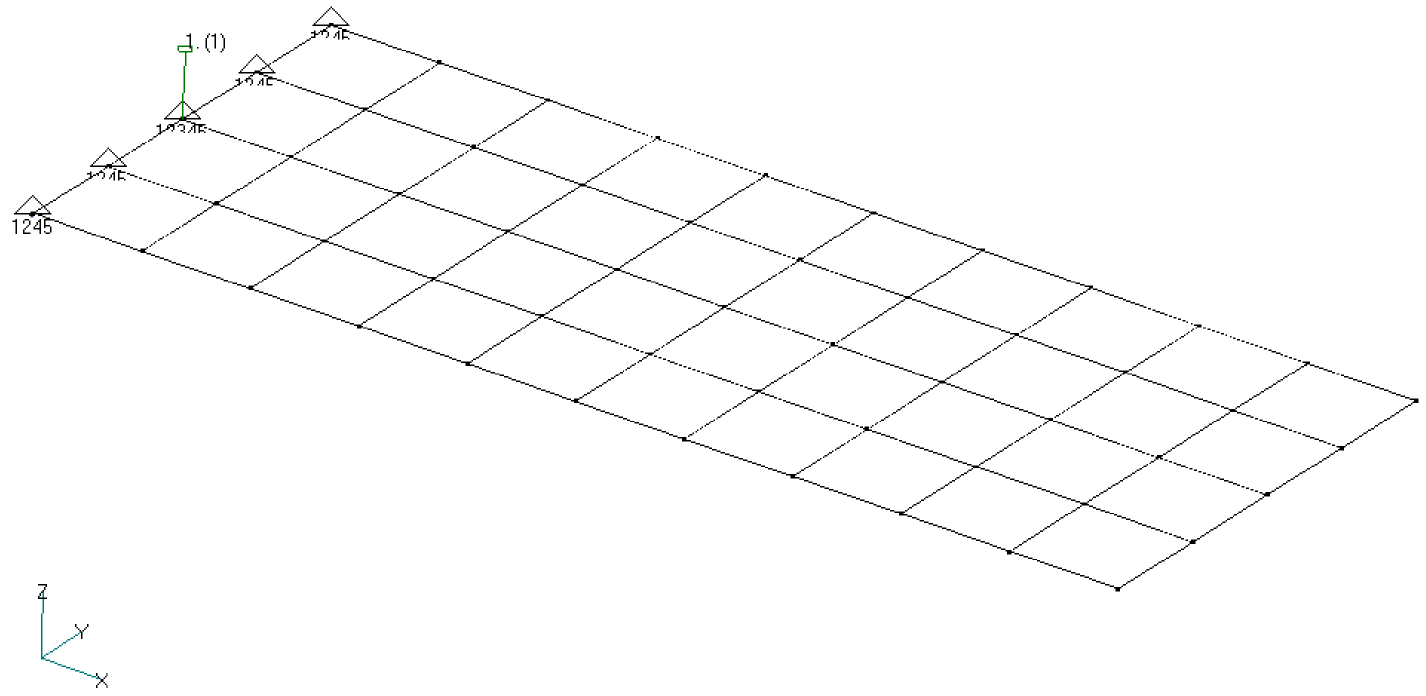
Problem #7a

Direct Transient Response with Enforced Acceleration

(Direct application of acceleration)

Problem #7a: Direct Transient Response with Enforced Acceleration (Direct)

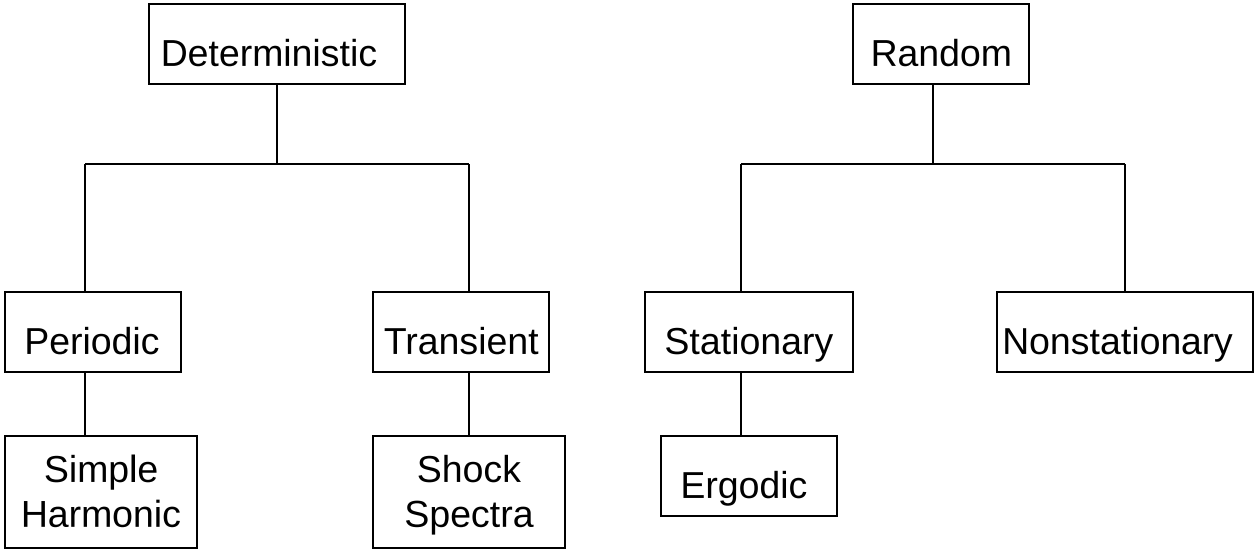
For this problem, use the direct method to determine the transient response of the flat rectangular plate, created in problem #1, subject to a **unit acceleration sine pulse of 250 HZ** applied to the base (**node 23**) in the z-direction. No large mass is not required for this example. Use a structural damping coefficient of **g = 0.06** and convert this damping to **equivalent viscous damping at 250 Hz**. Compare with results of **Ex.7**.



Random Response Analysis

NX Nastran Dynamic Analysis

Classification of Dynamic Environments



Introduction to Random Response

- Random Vibration is vibration that can be described only in a statistical sense. Its instantaneous magnitude at any time is not known; rather, the probability of its magnitude exceeding a certain value is given.

- Examples 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.

- NX Nastran performs random response as post-processing to frequency response. Inputs include the output from frequency response, as well as, user-supplied loading conditions in the form of auto- and cross-spectral densities. Outputs are response power spectral densities (PSDs), autocorrelation functions, number of zero crossings with positive slope per unit time, and the RMS values of response.

- Reference: Random Vibration in Mechanical Systems, by S. H. Crandall and W. D. Mark, Academic Press, 1963

Introduction to Random Response

- There are several conventions used to define random analysis quantities. Care must be taken to use NX Nastran random response capabilities properly.

- NX Nastran random analysis assumes ergodic processes, which means the excitations are stationary with respect to time.

- The concepts of autocorrelation, auto-spectrum (power spectrum), cross-correlation, and cross-spectrum must be defined.

- The mean square value and apparent frequency are the key statistical quantities to be gotten from Random Response.

Autocorrelation and Autospectrum

- Autocorrelation function:

$$R_j(\tau) = \lim_{T \to \infty} \frac{1}{T} \int_0^T u_j(t) u_j(t - \tau) d\tau$$

$R_j(o)$ is the mean-square value of

- Autospectrum function:

$$S_j(\omega) = \lim_{T \to \infty} \frac{2}{T} \left| \int_0^T u_j(t) e^{-i\omega t} d\tau \right|^2$$

Autocorrelation and Autospectrum

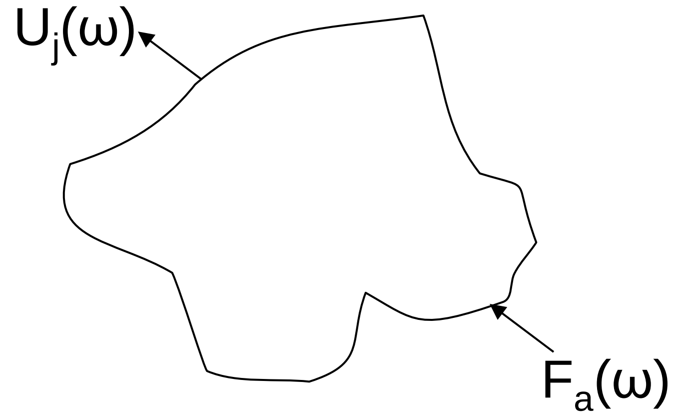
- Mean square value:

$$u_j(t)^2 = R_j(o) = \frac{1}{2\pi}\int_0^{\infty} S_j(\omega)d\omega$$

- Apparent frequency N_0 (zero crossings):

$$N_0{}^2 = \frac{\int_0^{\infty}(\omega/2\pi)^2 S_j(\omega)d\omega}{\int_0^{\infty} S_j(\omega)d\omega}$$

Calculation of Linear System Response to Ergodic Random Excitation



- From frequency response analysis:

$$u_j(\omega) = H_{ja}(\omega) \cdot F_a(\omega)$$

where $H_{ja}(\omega)$ is the frequency response or transfer function relating output u_j to input F_a.

- If there are several inputs, then:

$$u_j(\omega) = H_{ja}(\omega)F_a(\omega) + H_{jb}(\omega)F_b(\omega) + \ldots$$

Definition of Multiple Input-Output Spectral Relationship for a Linear System

- In matrix form:

$$u_j(\omega) = [\ H_{ja}(\omega)\ H_{jb}(\omega) \ldots\ ] \begin{Bmatrix} F_a(\omega) \\ F_b(\omega) \\ \ldots \end{Bmatrix}$$

- The output autospectrum is:

$$S_{ujuj} = T[\ H_{ja}\ H_{jb}\ \ldots\ ] \overline{\begin{Bmatrix} F_a(\omega) \\ F_b(\omega) \\ \ldots \end{Bmatrix} [F^*_a(\omega)\ F^*_b(\omega)\ \ldots\ ]} \begin{bmatrix} H^*_{ja}(\omega) \\ H^*_{jb}(\omega) \\ \ldots \end{bmatrix}$$

- The individual input spectra are:

$$T\overline{F_a(\omega)\ F^*_a}(\omega) = S_{aa}(\omega)$$

$$T\overline{F_a(\omega)\ F^*_b}(\omega) = S_{ab}(\omega)$$

$$T\overline{F_b(\omega)\ F^*_b}(\omega) = S_{bb}(\omega)$$

Definition of Multiple Input-Output Spectral Relationship for a Linear System

- The multiple input-output spectral relationship is therefore:

$$S_{ujuj}(\omega) = [\ H_j]^{\mathbf{T}} \begin{bmatrix} S_{aa} & S_{ab} & \cdots \\ S_{ba} & S_{bb} & \cdots \\ \cdots & \cdots & \cdots \end{bmatrix} [\ H^*_j]$$

where $[H_j]^{\mathbf{T}} = [\ H_{ja}\ H_{jb} \ldots\ ]$

$$[H^*_j] = \begin{Bmatrix} H^*_{ja} \\ H^*_{jb} \\ \cdots \end{Bmatrix}$$

Definition of Multiple Input-Output Spectral Relationship for a Linear System

- The input cross-spectral matrix is:

$$[S]_{\mathbf{IN}} = [\,H_j]^{\mathbf{T}} \begin{bmatrix} S_{aa}(\omega) & S_{ab}(\omega) & \ldots \\ S_{ba}(\omega) & S_{bb}(\omega) & \ldots \\ \ldots & \ldots & \ldots \end{bmatrix}$$

It has the special properties:

$$S_{ab}(\omega) = S^*_{ab}(\omega)$$

$$S_{aa}(\omega),\ S_{bb}(\omega) = \text{real} \geq 0$$

Definition of Multiple Input-Output Spectral Relationship for a Linear System

- Commonly used special cases:
  - Single input analysis (fully correlated inputs)

$$S_{ujuj}(\omega) = \left|H_{ja}(\omega)\right|^2 S_{aa}(\omega)$$

  - Uncorrelated multiple inputs

$$S_{ujuj}(\omega) = \left|H_{ja}(\omega)\right|^2 S_{aa}(\omega) + \left|H_{jb}(\omega)\right|^2 S_{bb}(\omega) + \ldots$$

Random Analysis as Implemented in NX Nastran

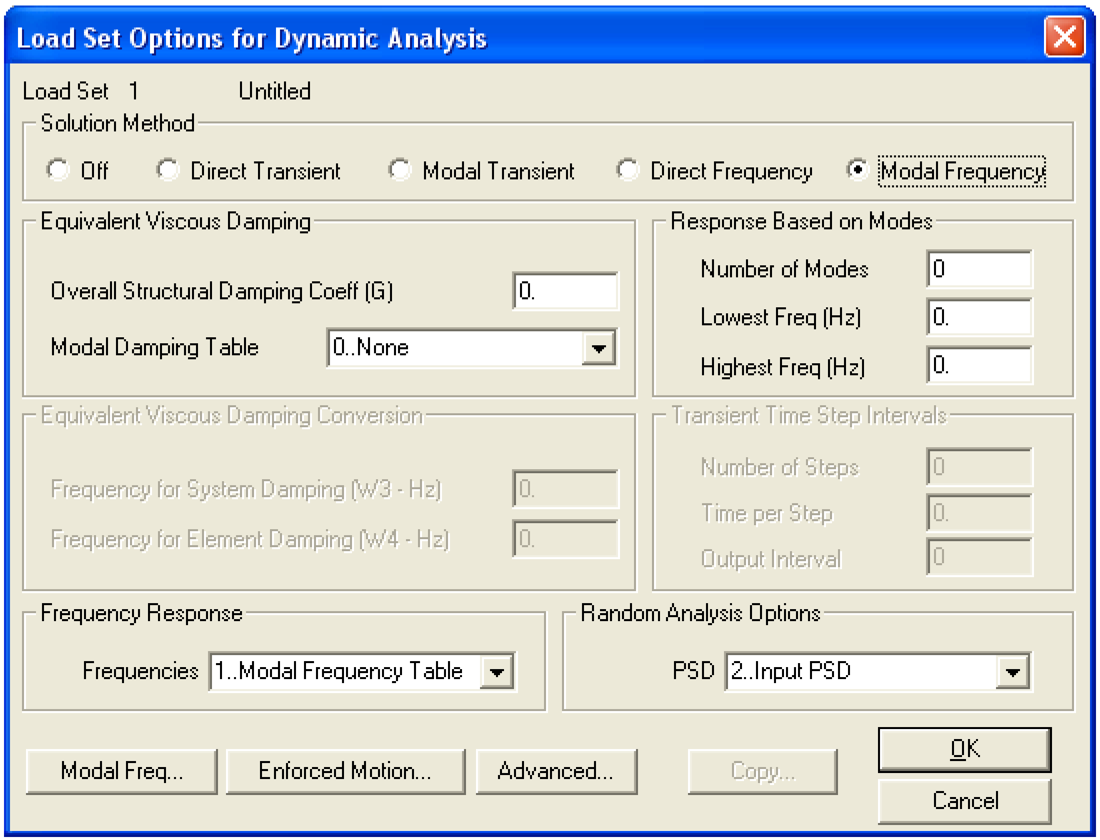
- It is assumed that the output from the frequency response calculations is $H_{ja}(\omega)$. It does not calculate:

$$H_{ja}(\omega) = u_j(\omega)/F_a(\omega)$$

If $H_{ja}(\omega)$ is desired, use $F(\omega) = 1.0$.

Random Analysis as Implemented in NX Nastran

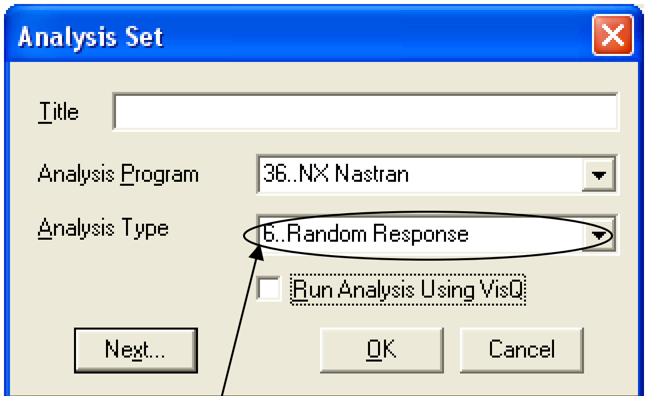
- Use the Model->Load->Dynamic Analysis command, the Load Set Options for Dynamic Analysis dialog box will appear:



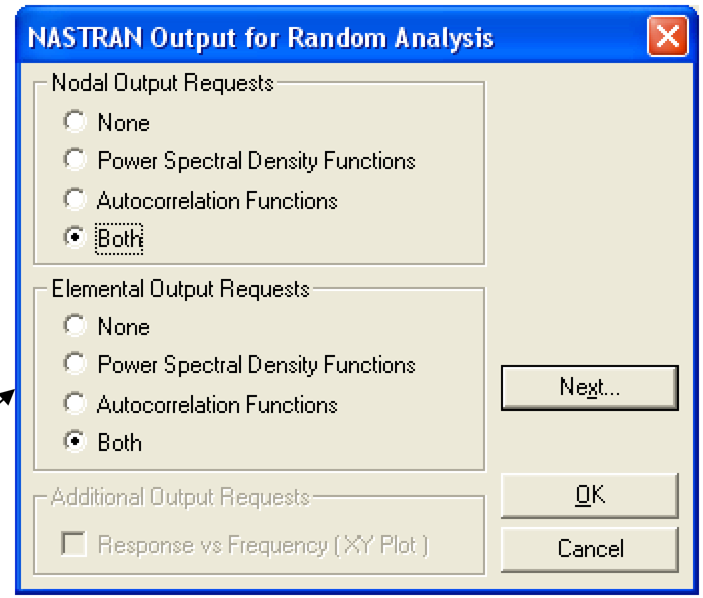
The Power Spectral Density data should be created in functional form and then selected from the PSD drop-down menu in the Random Analysis Options portion of the dialog box.

Random Analysis as Implemented in NX Nastran

- Use the Model->Analysis command to create a Random Response analysis set using the Analysis Set Manager :



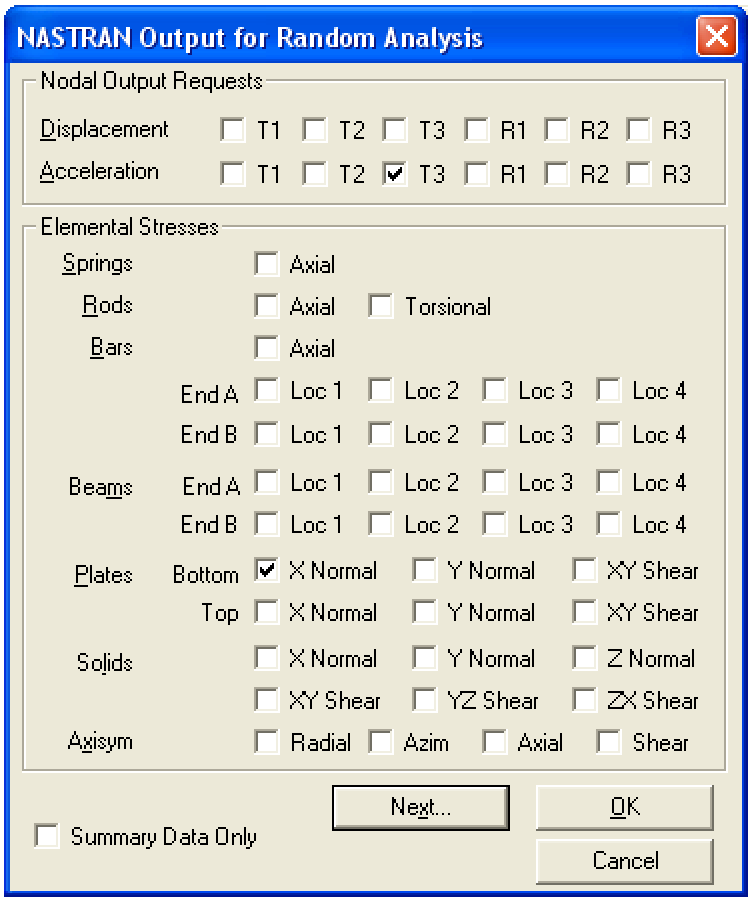
Choose 6..Random Response as the analysis type.



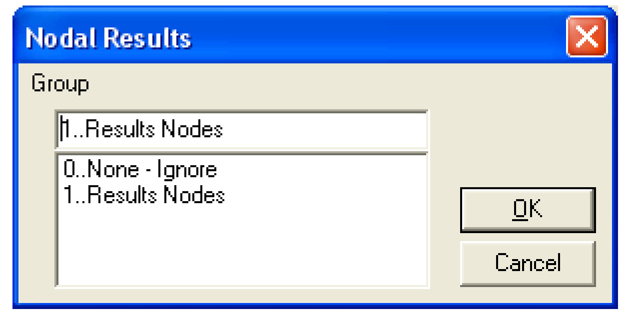
Determine what type of output is desired for Nodes and/or Elements. Choose Power Spectral Density Functions or Autocorrelation Functions or Both.

Random Response XY Output

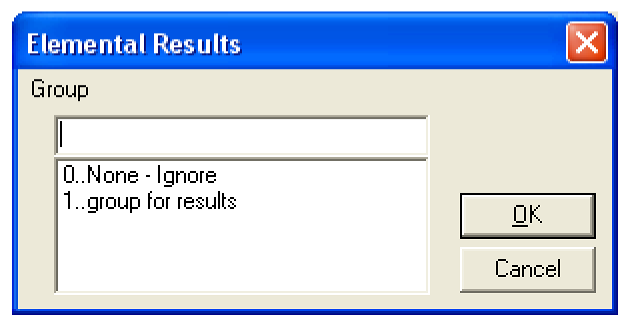
Once the type of Output is determined, the specific Output requests are made in this dialog box.



Choose existing group of nodes to retrieve requested output. (If Nodal results have been requested)

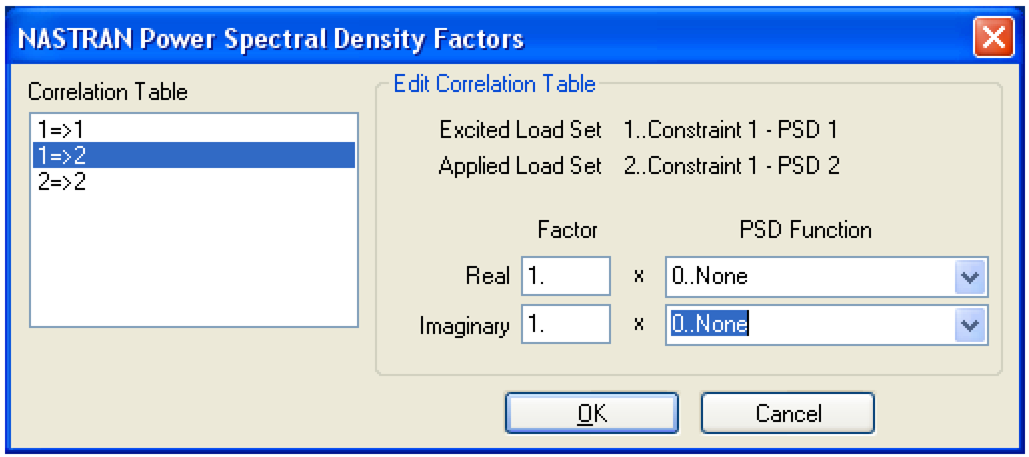


Choose existing group of elements to retrieve requested output. (If Elemental results have been requested)



Correlation Table

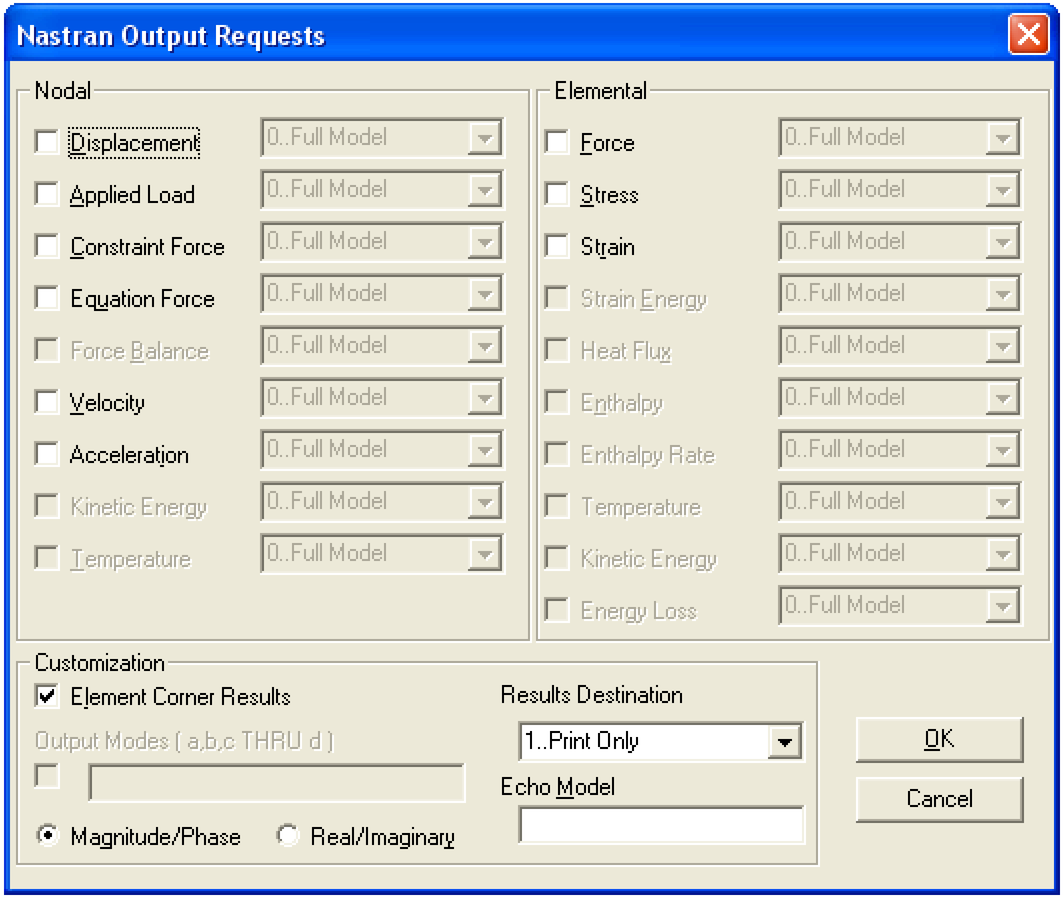
Only available when there are multiple subcases with multiple Load Sets. Choose real and imaginary component factors of specific PSD functions to create a correlation table between the Excited Load Set and the Applied Load Set. This Dialog box can only be found by in the tree structure of the Analysis Set Manager



An Example of when this might be useful is when correlating between PSD load sets from four wheels of a vehicle riding on a rough surface.

Random Analysis as Implemented in NX Nastran

Many times, no other Output needs to be requested other than the PSD functions and the Autocorrelation Functions for Random Response Analysis.



Random Analysis Recommendations

- Most spectra are given as a log function. Use equation features on the function form if the PSD is given in log scale.
- Always generate the output PSD at the input location if possible
- Plot the output PSD. Do not use the summery output blindly.
- Use several frequencies in the vicinity of each mode.
- For low frequencies (<20 Hz), use many frequencies since the displacement spectra is changing rapidly for a constant input acceleration.

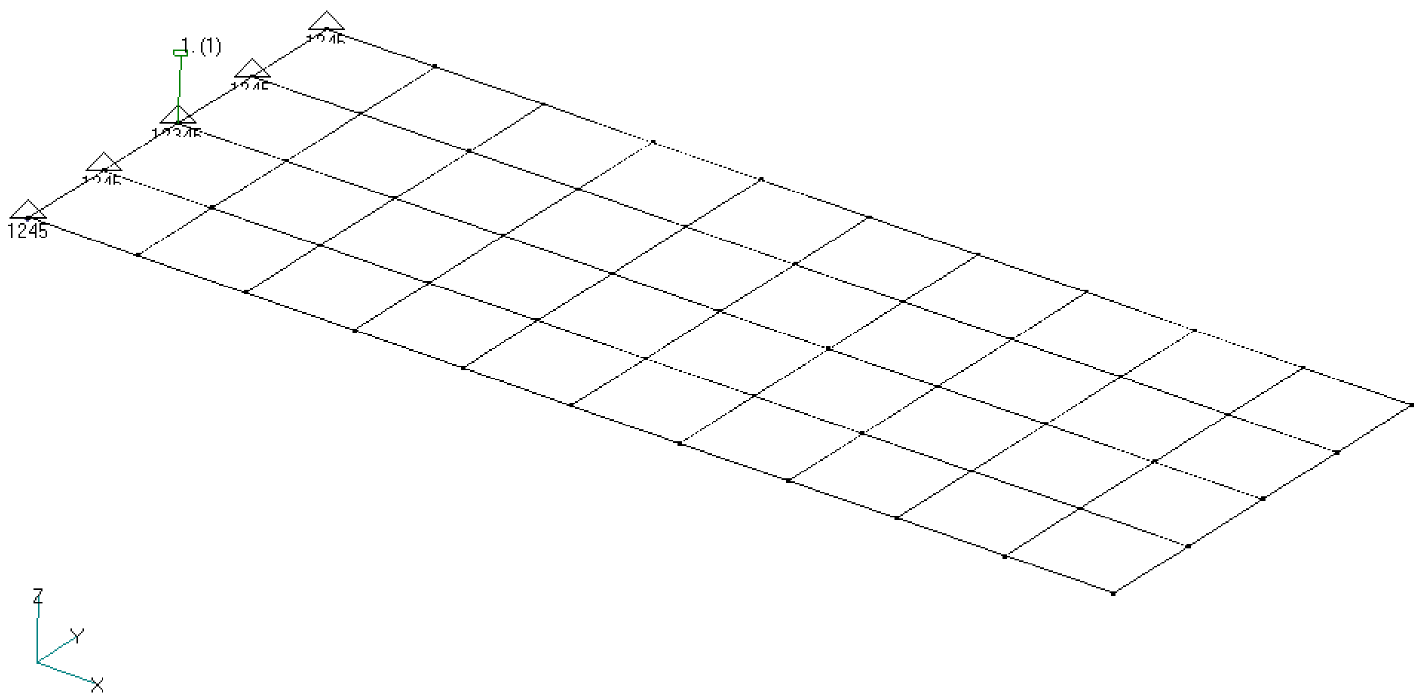
Problem #7a

Direct Transient Response with Enforced Acceleration

(Direct application of acceleration)

Problem #7a: Direct Transient Response with Enforced Acceleration (Direct)

For this problem, use the direct method to determine the transient response of the flat rectangular plate, created in problem #1, subject to a **unit acceleration sine pulse of 250 HZ** applied to the base (**node 23**) in the z-direction. No large mass is not required for this example. Use a structural damping coefficient of **g = 0.06** and convert this damping to **equivalent viscous damping at 250 Hz**. Compare with results of **Ex.7**.



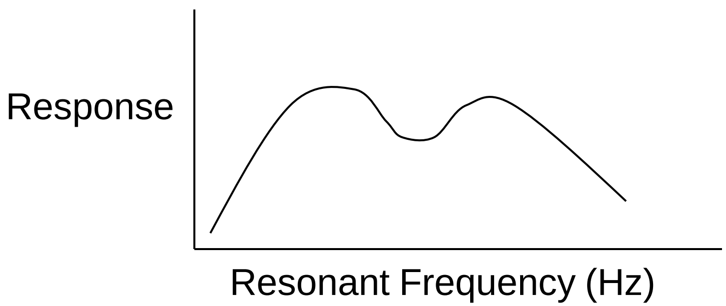
Shock and Response Spectrum

NX Nastran Dynamic Analysis

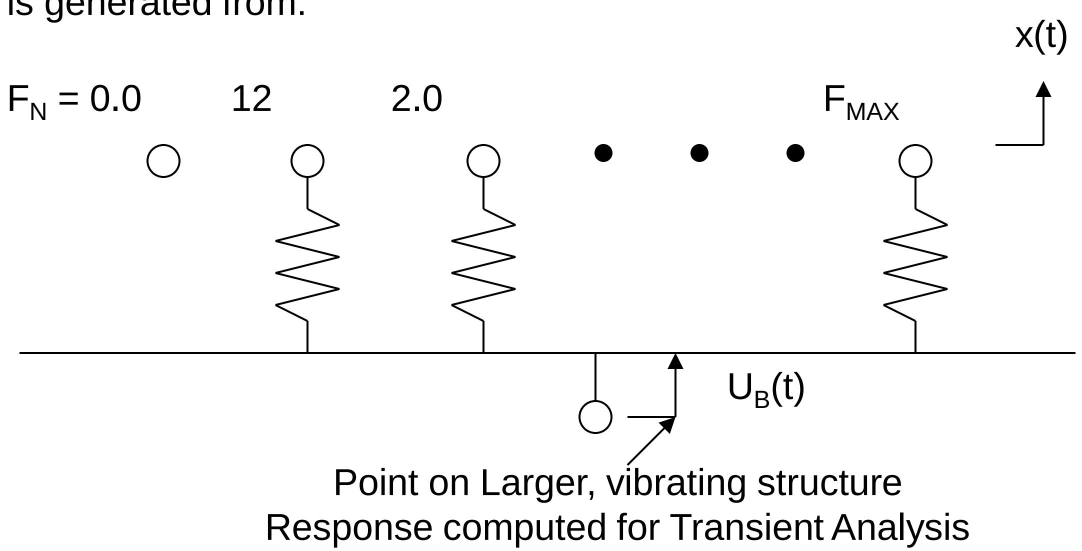
Response Spectrum

- Response spectrum depicts the maximum response of a SDOF system as a function of its resonant frequency for base excitation.

This Graph:



is generated from:



Response Spectrum

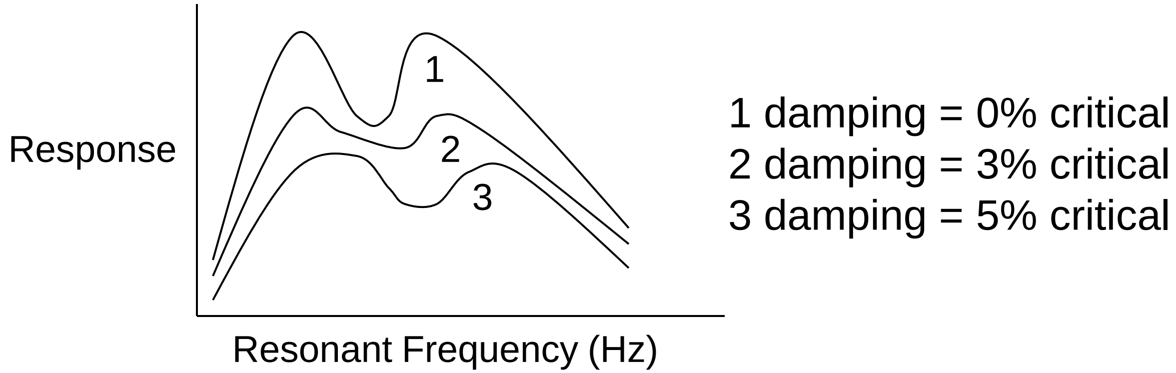
- The peak of each SDOF oscillator is calculated from its X(t). The oscillator base motion UB is derived from the force or base excitation applied to a larger structure.

- An example: An earthquake time history is applied to a power plant. Response Spectra are calculated at the locations of the floors to be used in the design of components (machinery, piping systems, etc.)

- An implicit assumption is that the oscillator's mass is very small relative to the larger, vibrating mass. Therefore, no dynamic interaction occurs between the two. (Consequently, the response spectrum analysis is decoupled from the transient analysis).

Response Spectrum

- Analysis is repeated for several damping values to generate a family of curves.



Damping applies to each oscillator, not the vibrating structure

Response Spectrum

- Maximum displacement response from X(t) is calculated for each oscillator. The maximum relative displacement between each oscillator and its base (a point on the vibrating structure) is also computed.

  X = maximum inertial (absolute)

  X_r = maximum relative

- Relative velocity and absolute acceleration are approximately related to the relative displacements by

$$\dot{X}_r = \omega X_r$$
$$\ddot{X} = \omega^2 X_r$$

- For design, useful variables are X_r, $\dot{X}_r$, and $\ddot{X}$. Design spectra are usually in terms of these variables.

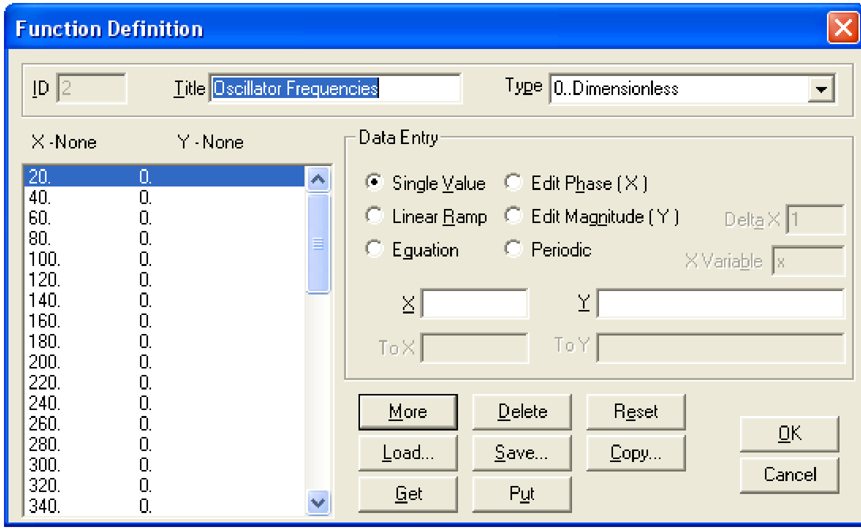
Response Spectrum

- Response Spectra may be generated in any transient solution (SOLs 109, 112).

- The transient response for selected DOFs in model is used as the input time history for the generation of the response spectra curves.

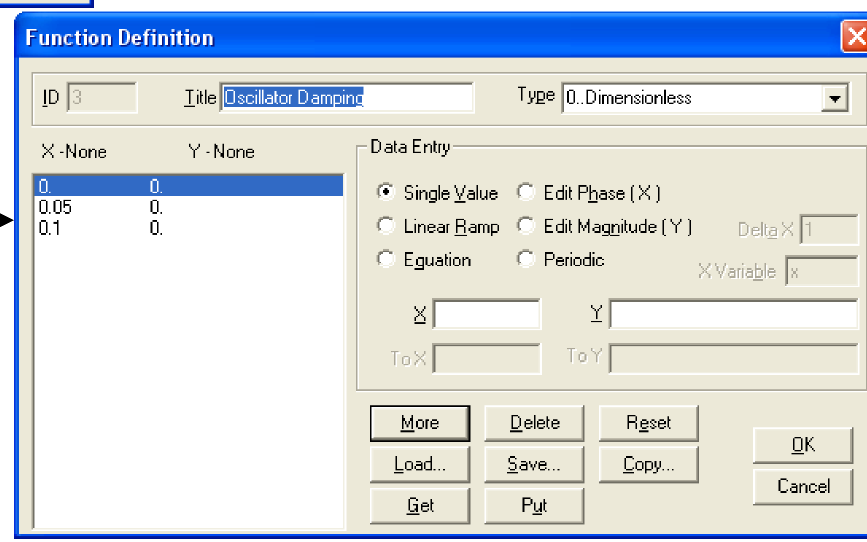
Response Spectrum Generation

- Define Functions using Model->function command



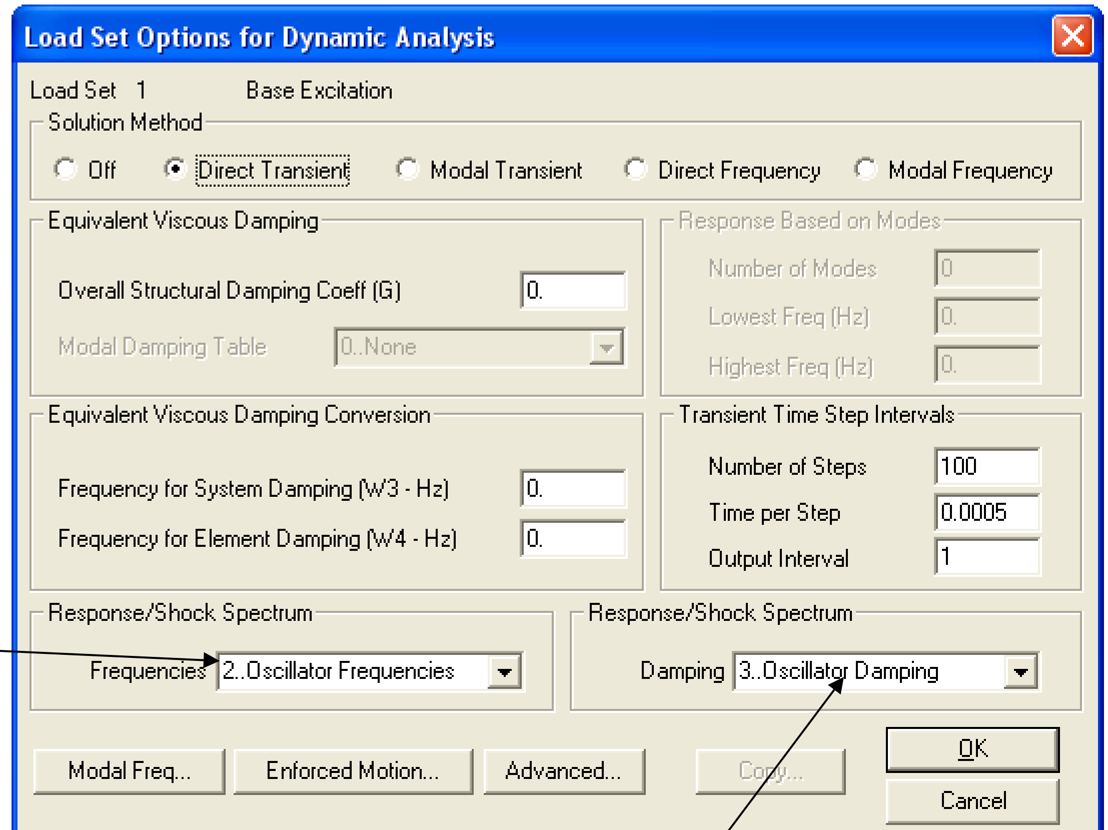
Function to Define Oscillation Frequencies

Function to Define Oscillation Dampings



Response Spectrum Generation

- Use the Model->Load->Dynamic Analysis command, the Load Set Options for Dynamic Analysis dialog box will appear:

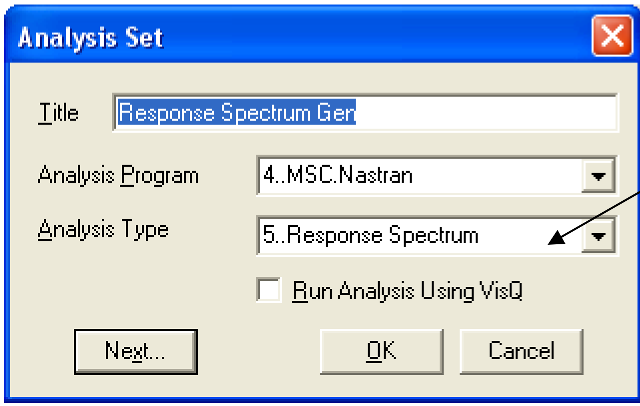


Function to Define Oscillation Frequencies

Function to Define Oscillation Dampings

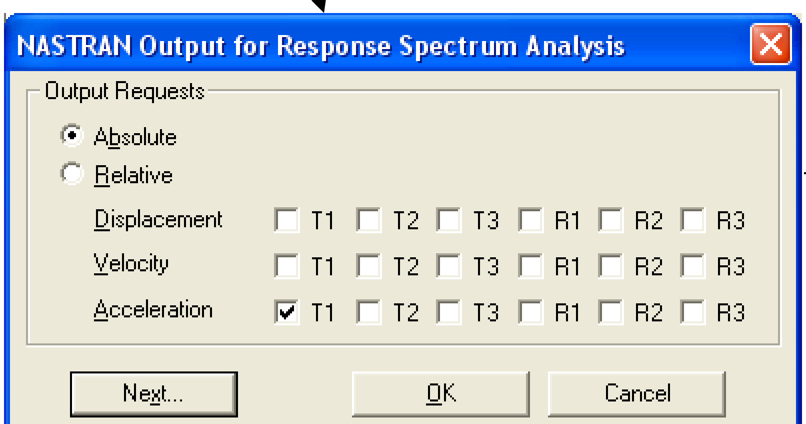
Response Spectrum Generation

- Response Spectrum Specific dialog boxes using the Analysis set manager, Modal->Analysis command.

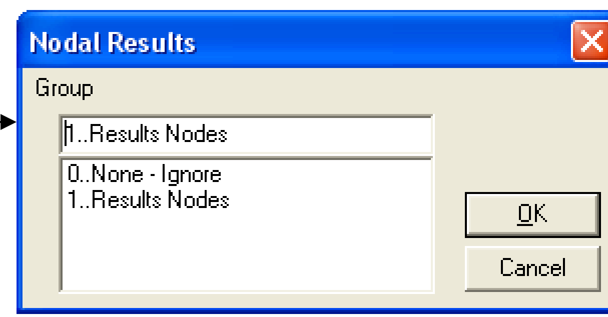


Define Analysis Type as 5..Response Spectrum

Request NASTRAN Output for Response Spectrum Analysis



Choose existing group of nodes to retrieve requested output.



Applying Spectra

Available in Solution 103:

- "Poor Man's Transient." The input spectra are used to determine the peak response of each mode.
- These peak modal responses are combined to obtain the system response (timing of each mode's peak is not known).
- Three methods of combining the modal responses are available (ABS, SRSS, NRL)

Applying Spectra

Procedure:

- A model of the structure to be analyzed is created with the input points identified as ‘SUPORT’ DOFs.
- A “large mass” (usually 10^3 to 10^6 times the structural mass) is attached to the ‘SUPORT’ DOFs.
- System modes are obtained for the modal (including 0.0 Hz modes) with the ‘SUPORT’ DOFs unconstrained.
- This approximates the “cantilevered” modes of the model attached to the “exciting” structure.

Applying Spectra

- The 0.0 Hz modes (Dm) approximate the ‘static’ motion the model experiences when the supporting structure moves statically.
- “Participation Factors” (PF) are calculated using the following expression:

$$\Psi = \Phi^T M D m$$

- PF is used in conjunction with the spectra described as shown in the input section to calculate the peak response for each mode.
- Data recovery quantities (displacements, stresses, forces, etc.) are then calculated for each mode based on its peak motion.
- These quantities are then combined for the modes using the selected method (ABS,SRSS,NRL, NRLO) and the results are printed.

Applying Spectra

- X_r response of a single DOF oscillator due to the base motion is calculated as follows:

$$\ddot{X}_r + g\omega\dot{X}_r + \omega^2 X_r = \ddot{u}_r(t)$$

- The actual transient response at a physical point is

$$u_k(t) = \sum_i \sum_r \Phi_{ik} \Psi_{ir} X_r\,(\omega_i, g_i, t)$$

<u>Absolute Value Rule (ABS) option</u>:

$$\bar{u}_k \cong \sum_i \sum_r \left|\Phi_{ik}\right| \left|\Psi_{ir} \bar{X}_{ri}\,(\omega_i, g_i)\right|$$

where $$X_{ri}\,(\omega_i, g_i) = \max \left| X_{ri}\,(\omega_i, g_i, t)\right|$$

and i represents a mode
and r represents a direction

Applying Spectra

Square Root of the Sum of the Squares Rule (SRSS) option:

$$\bar{u}_k \cong \sqrt{\sum_i (\Phi_{ik}\bar{\xi}_i)^2}$$

Where the average peak modal magnitude, ξ_i is

$$\bar{\xi}_i \cong \sqrt{\sum_r (\Psi_{ir}\bar{X}_r\,(\omega_i, g_i))^2}$$

U.S. Navy Shock Design Modal Summation Convention (NRL) option:

$$\bar{u}_k \cong \left|\Phi_{jk}\bar{\xi}_j\right| + \sqrt{\sum_{i \neq j} (\Phi_{ik}\xi_i)^2}$$

where $\left|\Phi_{jk}\xi_j\right|$ is the peak magnitude

Applying Spectra

- NRLO refers to the NRL method used in version 69 of MSC Nastran. The NRL was updated slightly in version 70 to adhere to NAVSEA-0908-LP-000-3010 specification.

- The ABS rule is the most conservative – it assumes that the modal responses all achieve their peak response at the same time and with the same with the same phase (this rarely happens), therefore it usually over-predicts the response.

Applying Spectra

- Modes that are close in frequency may have their peak response occur at the same time (and with the same phase). The SRSS and NRL methods contain a provision to sum modal responses via ABS method for modes that have closely spaced natural frequencies. "Close" natural frequencies are defined by frequencies that meet the following in equality:

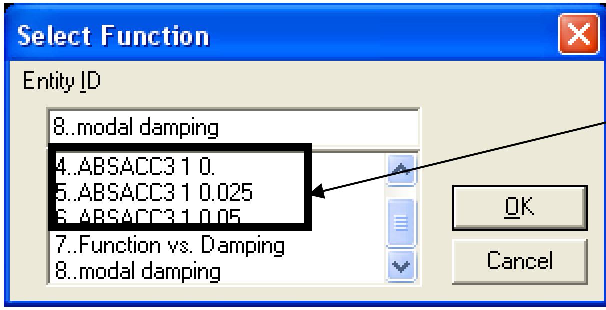
$$f_{i+1} < CLOSE * f_i$$

- The value of CLOSE is defined by PARAM,CLOSE (the default is 1.0).

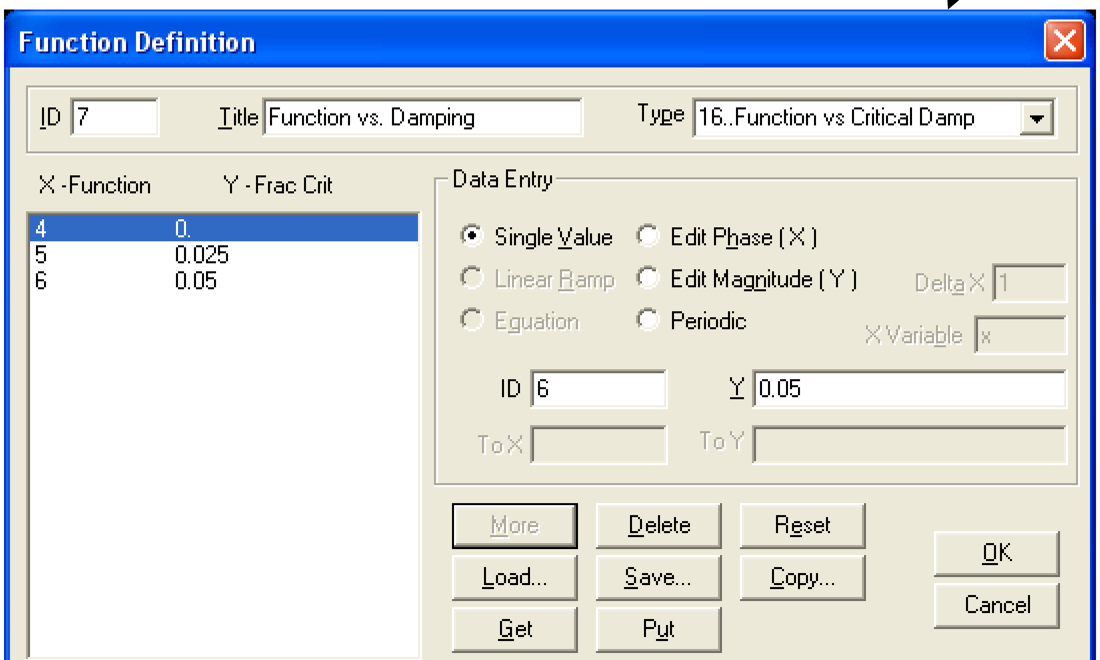
- The modal summation option is set via PARAM,OPTION (ABS is default). Both PARAM,OPTION and PARAM, CLOSE may be set in any subcase allowing for summation by several conventions in a single run.

Response Spectrum Application

- Create a Function vs. Damping function using the “results” of the Shock Spectrum Generation

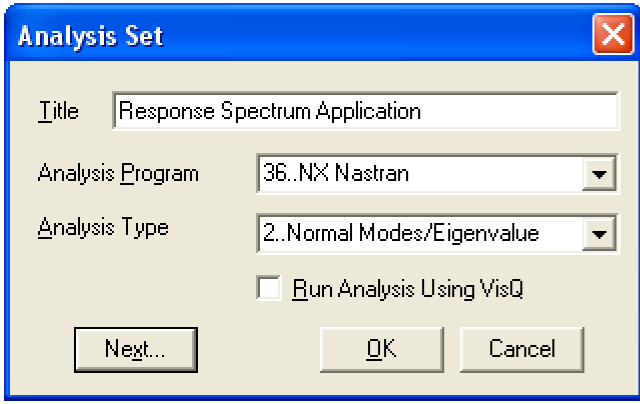


Use these functions to create a Function vs. Damping relationship for the Spectrum Function ID.

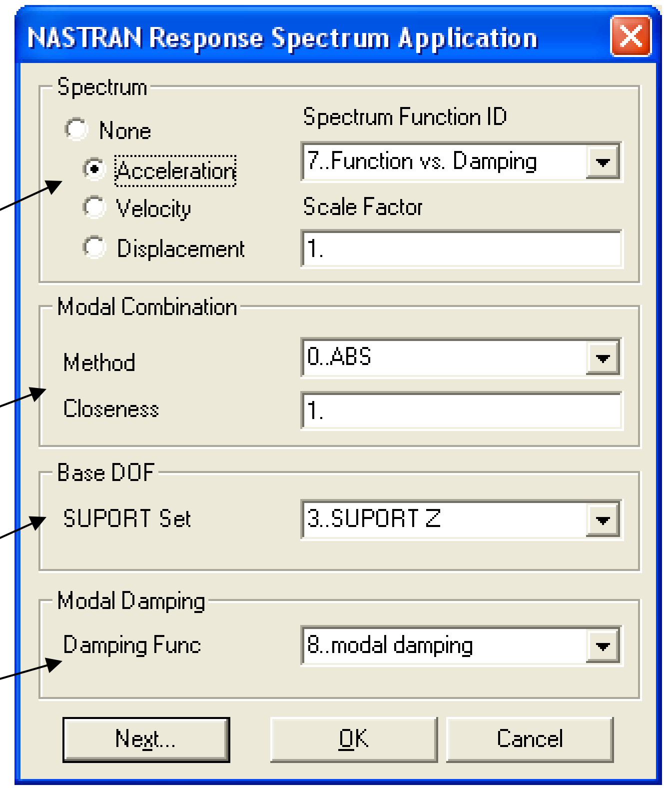


Response Spectrum Application

- Response Spectrum Specific dialog boxes using the Analysis set manager, Modal->Analysis command.



Define Analysis Type as 2..Normal Modes/Eigenvalue



Choose the type of Spectrum and a Spectrum function ID (a scale factor can also be added)

Choose a modal combination method (ABS, SRSS, NRL, NRLO) and a “closeness” factor.

Chose a constraint set to be the SUPORT set and

Choose a damping function for modal damping