Dynamic Analysis Using MSC Nastran

NAS102A Course Notes

June 2013

MSC Software®
Simulating Reality, Delivering Certainty™

NA*V2013*Z*Z*Z*SM-NAS102A-NT

Legal Information

CONTENTS

CONTENTS

CONTENTS

CONTENTS

CONTENTS

CONTENTS

CONTENTS

CONTENTS

CONTENTS

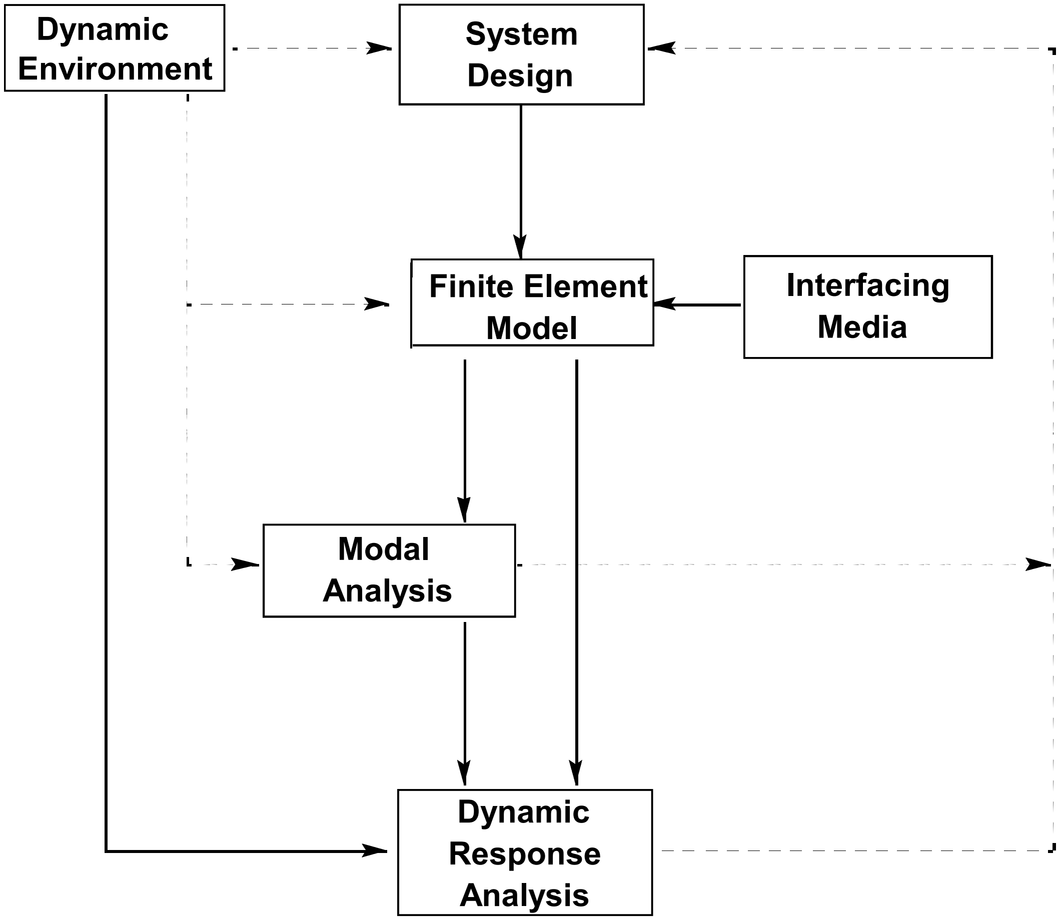
SECTION 1

REVIEW OF FUNDAMENTALS

COURSE OBJECTIVES

- **In this class (NAS102A), you will learn the following:**
  - Fundamentals of Dynamics
  - Normal Modes
  - Direct and Modal Methods
  - Frequency Response
  - Transient Response
  - Damping
  - Enforced Motion
  - Response Spectrum
  - Random Analysis
  - Residual Vectors
- **This class builds the foundation for the Advanced Dynamic Seminar (NAS102B)**

OVERVIEW OF DYNAMIC ANALYSIS PROCESS

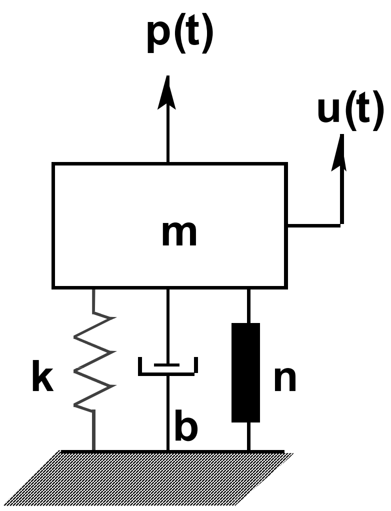


SINGLE DOF SYSTEM

- **Dynamic equation of motion:**

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

  - m = mass (inertia)
  - b = damping (energy dissipation)
  - k = stiffness (restoring force)
  - n = nonlinear 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
  - n is a nonlinear function of $u, \dot{u}$



UNITS

- **Fundamental units**
  - Length L (inch, meter)
  - Mass M (slug, kilogram)
  - Time T (second)
  - Length L (meter, millimeter)
  - Force F (Newton)
  - Time T (second)

$$1\frac{Ns^2}{mm} = 1000kg = 1t$$

- **Fundamental and derived units**

– m	M	Mass	$FT^2/L = F/(LT^{-2})$
– b	MT^{-1}	Damping	$FT/L = F/(T/L)$
– k	MT^{-2}	Stiffness	F/L
– p	MLT^{-2}	Force	F
– u	L	Displacement	L
– $\dot{u}$	LT^{-1}	Velocity	LT^{-1}
– $\ddot{u}$	LT^{-2}	Acceleration	LT^{-2}

UNITS

- **Engineering units for common variables**

Variable	Dimensions*	Common English Units	Common Metric Units
Length	L	in	m
Mass	M	$lb\text{-}sec^2/in$	kg
Time	T	sec	sec
Area	L2	in^2	m^2
Volume	L3	in^3	m^3
Velocity	LT^{-1}	in/sec	m/sec
Acceleration	LT^{-2}	in/sec^2	m/sec^2
Rotation	-	rad	rad
Rotational Velocity	T^{-1}	rad/sec	rad/sec
Rotational Acceleration	T^{-2}	rad/sec^2	rad/sec^2
Circular Frequency	T^{-1}	rad/sec	rad/sec
Frequency	T^{-1}	cps; Hz	cps; Hz
Eigenvalue	T^{-2}	rad^2/sec^2	rad^2/sec^2
Phase Angle	-	deg	deg
Force	MLT^{-2}	lb	N
Weight	MLT^{-2}	lb	N
Moment	$ML2T^{-2}$	in-lb	N-m
Mass Density	ML^{-3}	$lb\text{-}sec^2/in^4$	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	$in\text{-}lb\text{-}sec^2$	$kg\text{-}m^2$
Stiffness	MT^{-2}	lb/in	N/m
Viscous Damping Coefficient	MT^{-1}	lb-sec/in	N-sec/m
Stress	$ML^{-1}T^{-2}$	lb/in^2	Pa; N/m^2
Strain	-	-	-

Notes:

- *L denotes length
- M denotes mass
- T denotes time
- \- Denotes dimensionless

UNITS

- **It is very important to use consistent units**
- **Error in units is the number one cause of modeling errors in dynamic analysis**
- **Most common errors are in mass and damping units**
- **MSC Nastran contains no built-in set of units. The analyst must verify consistency.**
- **Examples of Consistent Units:**
  - *N, t, mm, s*
  - *N, kg, m, s*

SINGLE DOF SYSTEM - UNDAMPED FREE VIBRATIONS

- **Dynamic equation**

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

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

- **Solution**

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

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

  – Initial conditions

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

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

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

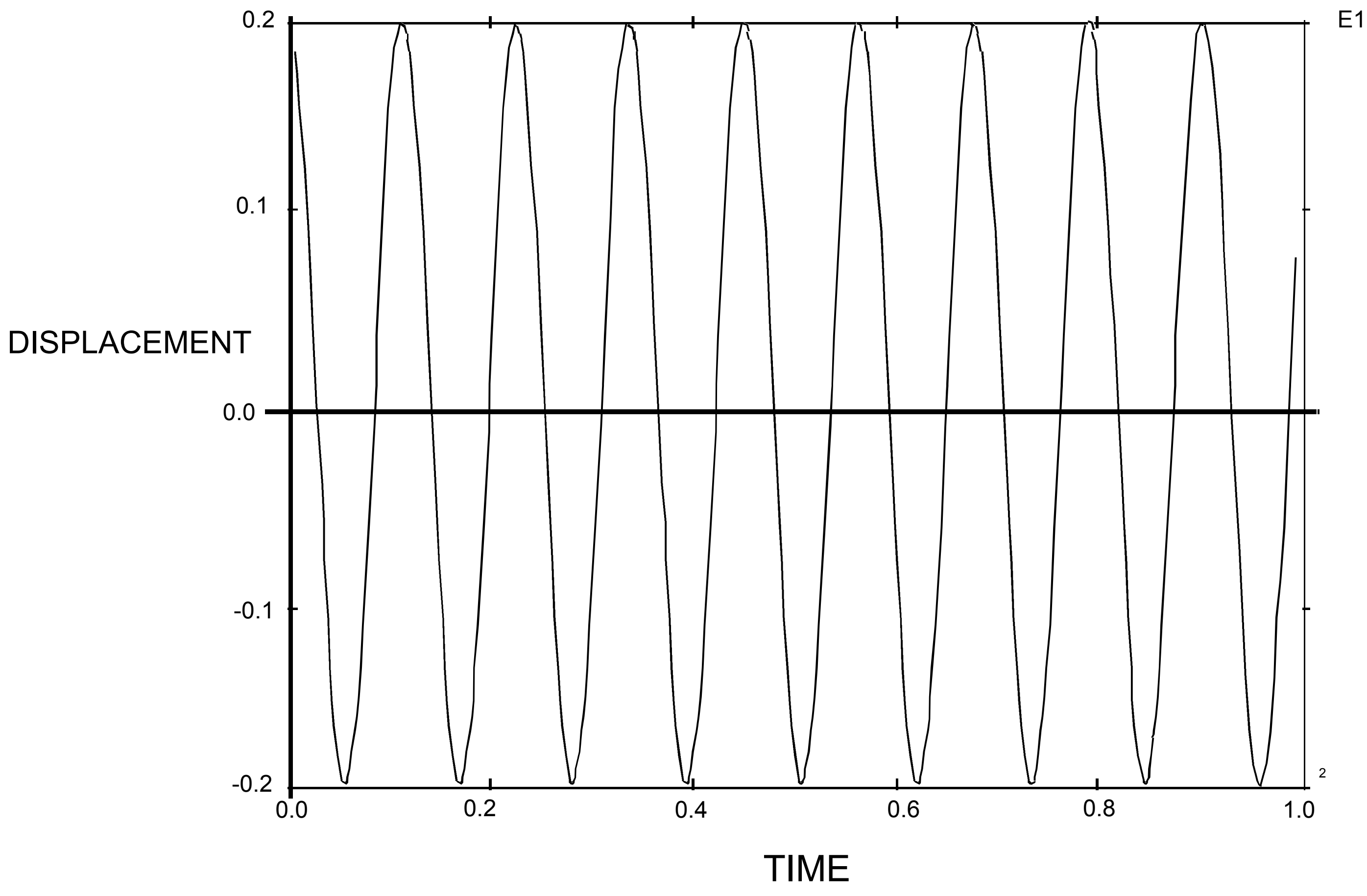
$$\dot{u}(t)=A\omega_n\cos\omega_n t-B\omega_n\sin\omega_n t$$

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

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

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

SINGLE DOF SYSTEM - UNDAMPED FREE VIBRATIONS



SINGLE DOF SYSTEM – DAMPED FREE VIBRATIONS

- **Dynamic Equation**

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

- **Critical Damping**

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

- **Fraction of critical damping**

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

- **The amount of damping determines the form of the solution**
  - Underdamped, $b < b_c$

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

    - Damped natural frequency

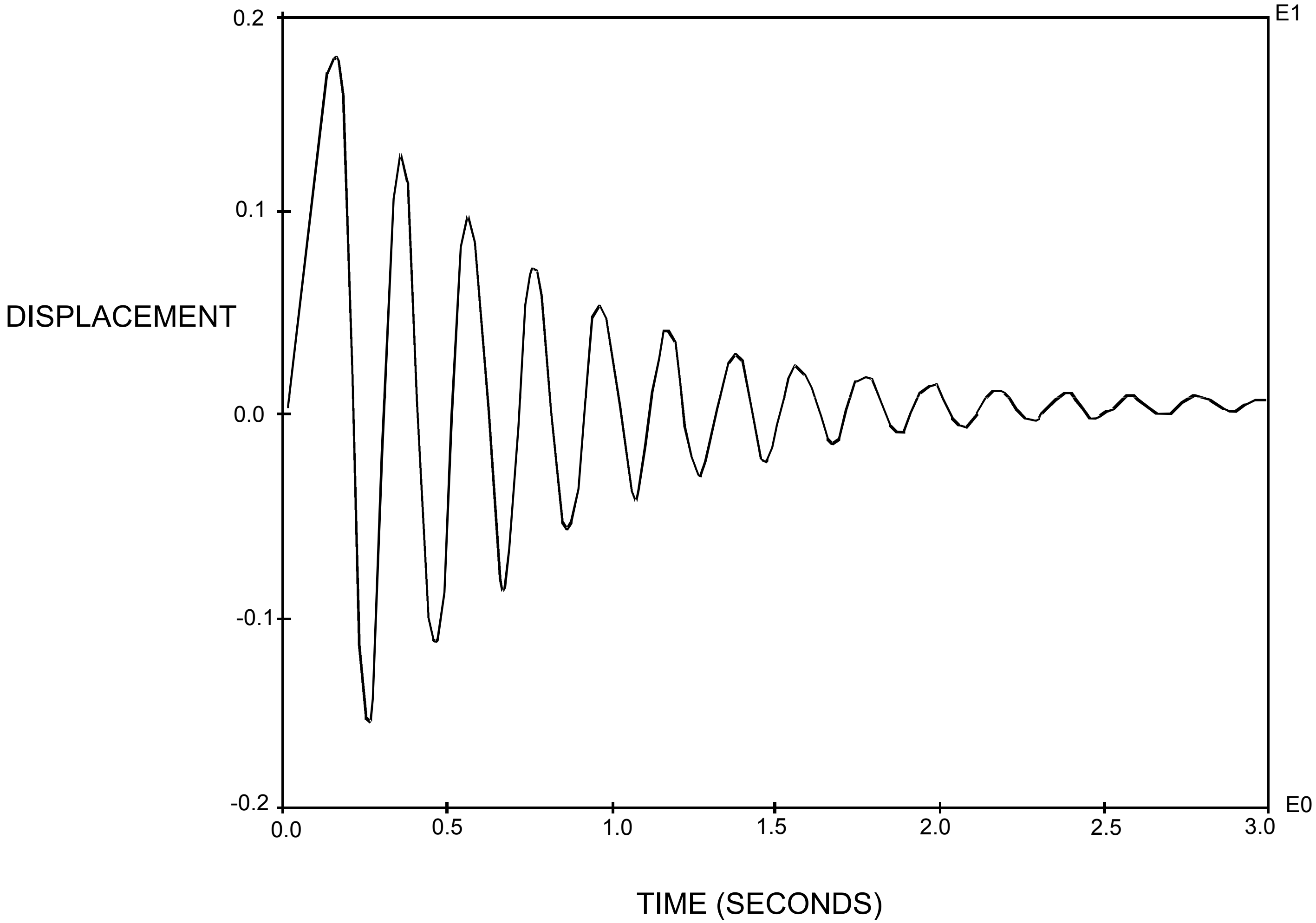
$$\omega_d = \omega_n\sqrt{1-\zeta^2}$$

SINGLE DOF SYSTEM - DAMPED FREE VIBRATIONS

- **Amount of damping determines the form of the solution (continued)**
  - 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 typically have damping in the 0 – 10% range**

DAMPED FREE VIBRATIONS—UNDERDAMPED CASE



CDAMP Element ζ = -0.05

SINGLE DOF SYSTEM - UNDAMPED SINUSOIDAL VIBRATIONS

- **Dynamic equation**

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

  – where ω = forcing frequency

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

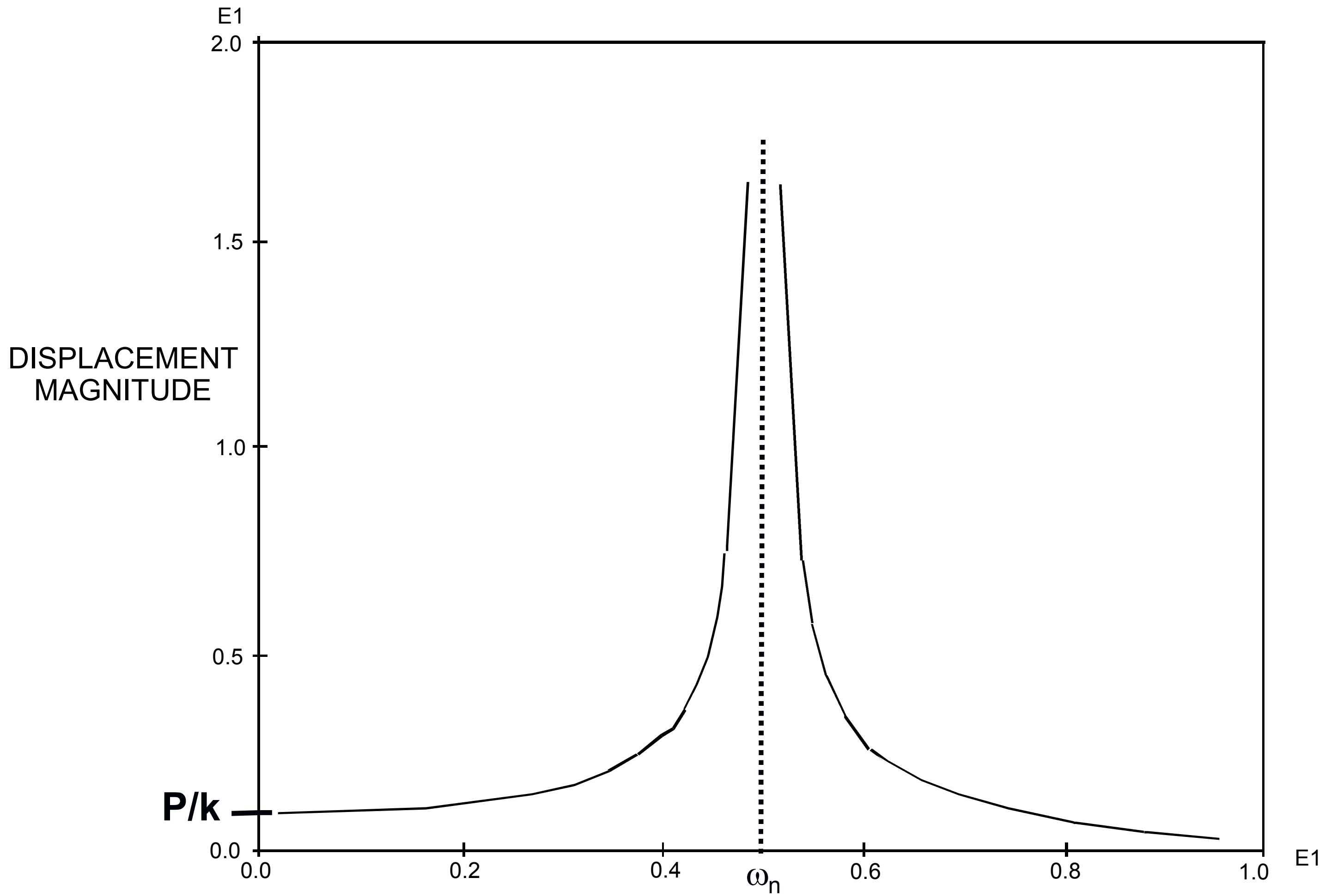
  – where $B = u(t=0)$

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

SINGLE DOF SYSTEM - UNDAMPED SINUSOIDAL VIBRATIONS

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

MAGNIFICATION FACTOR



SD OF System No B

FREQUENCY (HZ)

SINGLE DOF SYSTEM - DAMPED SINUSOIDAL VIBRATIONS

- **Dynamic equation**

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

- **The transient solution decays rapidly and is of no interest**
- **Steady-state solution**

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

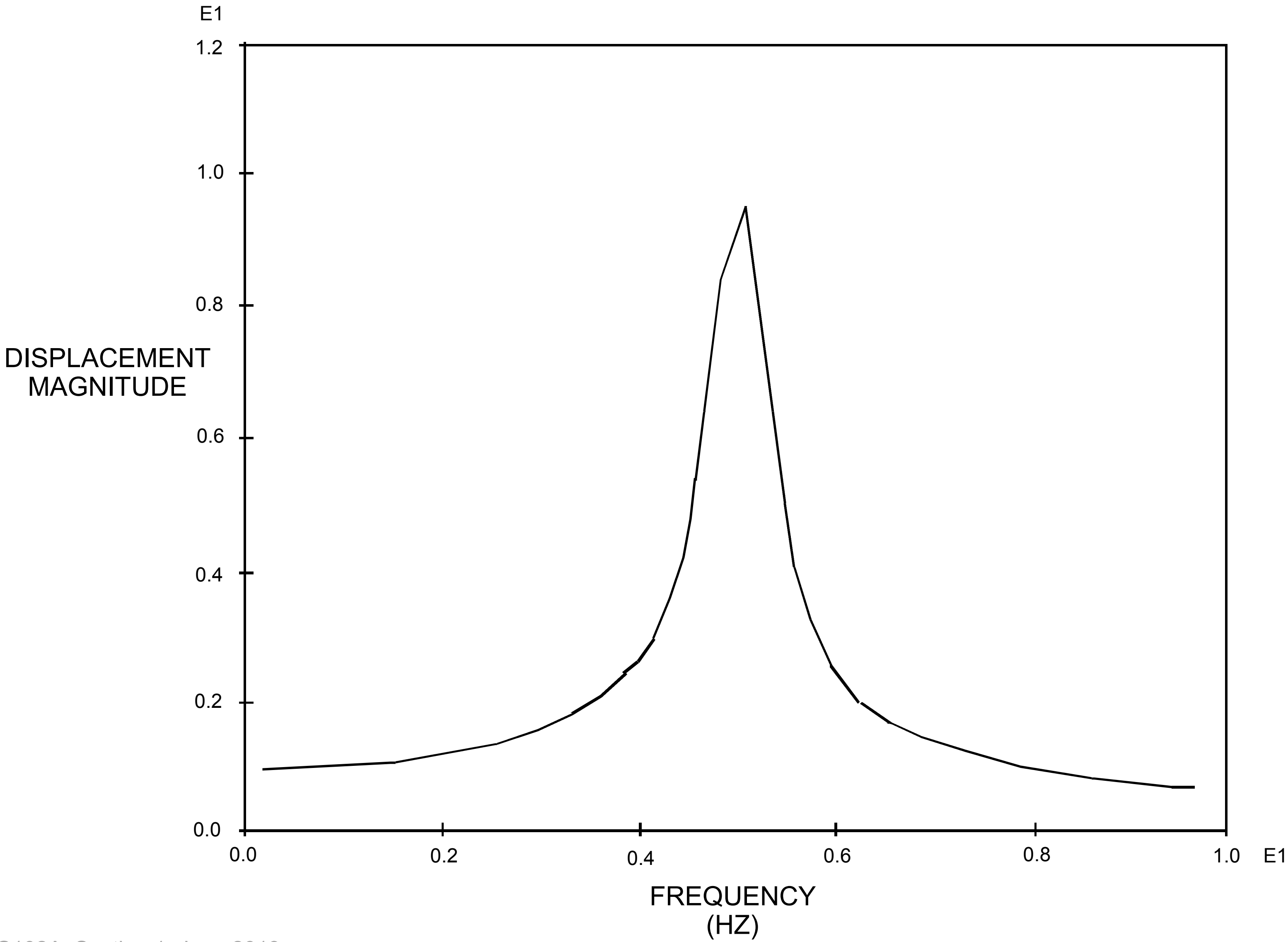
$$\theta=-\tan^{-1}\frac{2\zeta\omega/\omega_n}{1-\omega^2/\omega_n^2}$$

- **θ is phase lead => 180° ≤ θ ≤ 360° – for Nastran output**

SINGLE DOF SYSTEM - DAMPED SINUSOIDAL VIBRATIONS

- **For** $\frac{\omega}{\omega_n} << 1$
  - Magnification factor → 1 (static solution)
  - Phase angle → 360° (response is in phase with the force)
- **For** $\frac{\omega}{\omega_n} >> 1$
  - Magnification factor → 0 (no response)
  - Phase angle → 180° (response has opposite sign of force)
- **For** $\frac{\omega}{\omega_n} \approx 1$
  - Magnification factor $\approx \frac{1}{2\zeta}$
  - Phase angle » 270°

MAGNIFICATION FACTOR



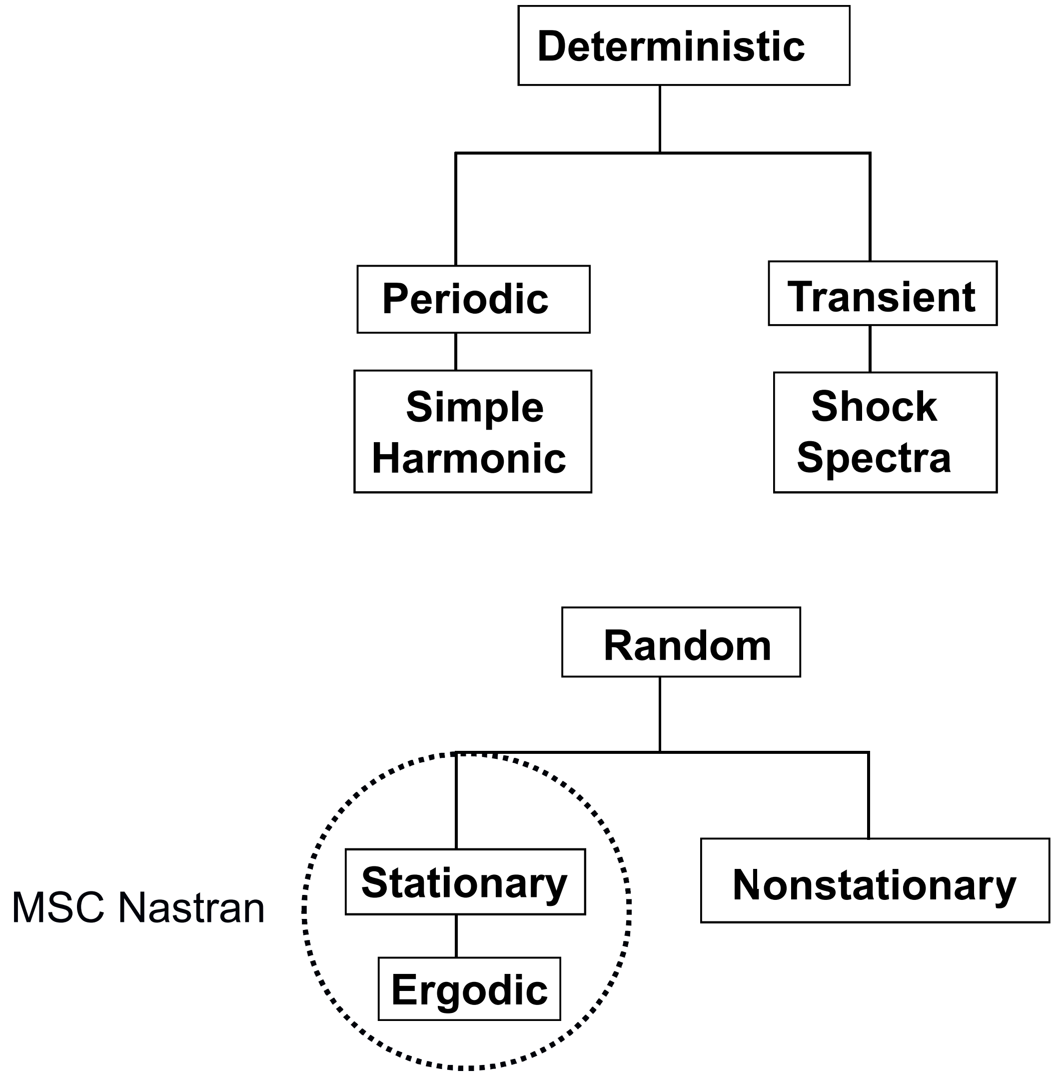
MULTI-DEGREE OF FREEDOM SYSTEM

- **Now the equation of motion becomes**

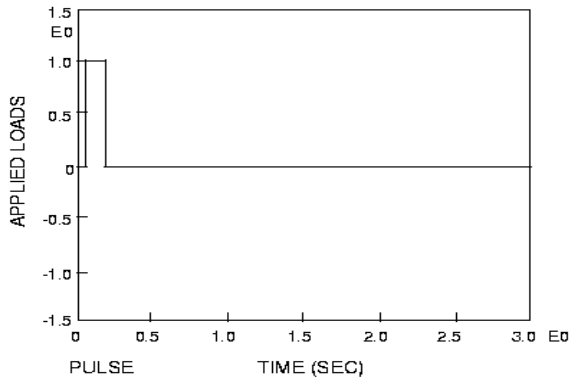
$$[\mathbf{M}]\{\ddot{\mathbf{u}}\}+[\mathbf{B}]\{\dot{\mathbf{u}}\}+[\mathbf{K}]\{\mathbf{u}\}=\{\mathbf{P}\}+\{\mathbf{N}\}$$

  – where
    - {u} = displacement vector
    - {M}= mass matrix
    - {B}= damping matrix
    - {K}= stiffness matrix
    - {P}= forcing function
    - {N}= nonlinear vector of forces

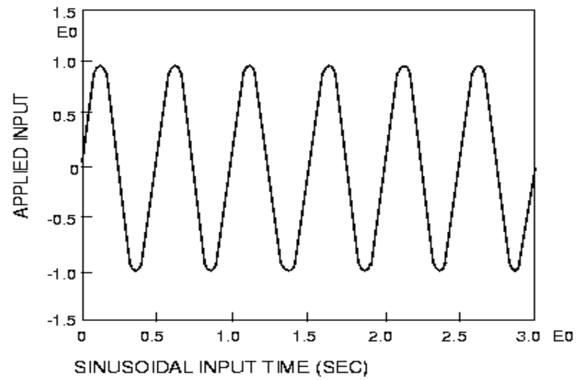
CLASSIFICATION OF DYNAMIC ENVIRONMENTS



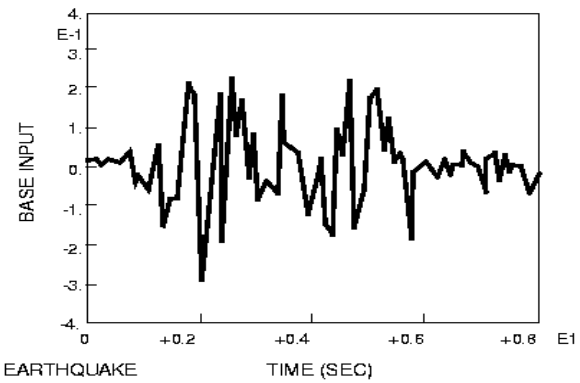
DYNAMIC EXCITATIONS



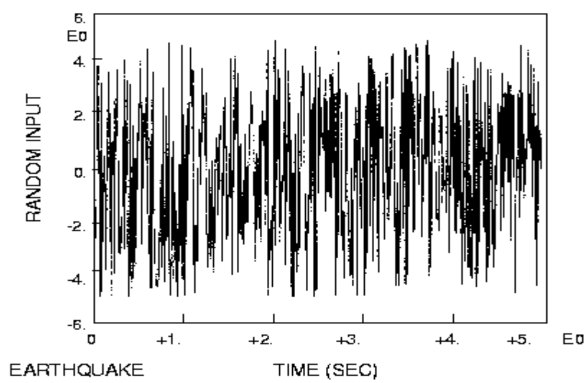
PULSE



SINUSOIDAL



TRANSIENT



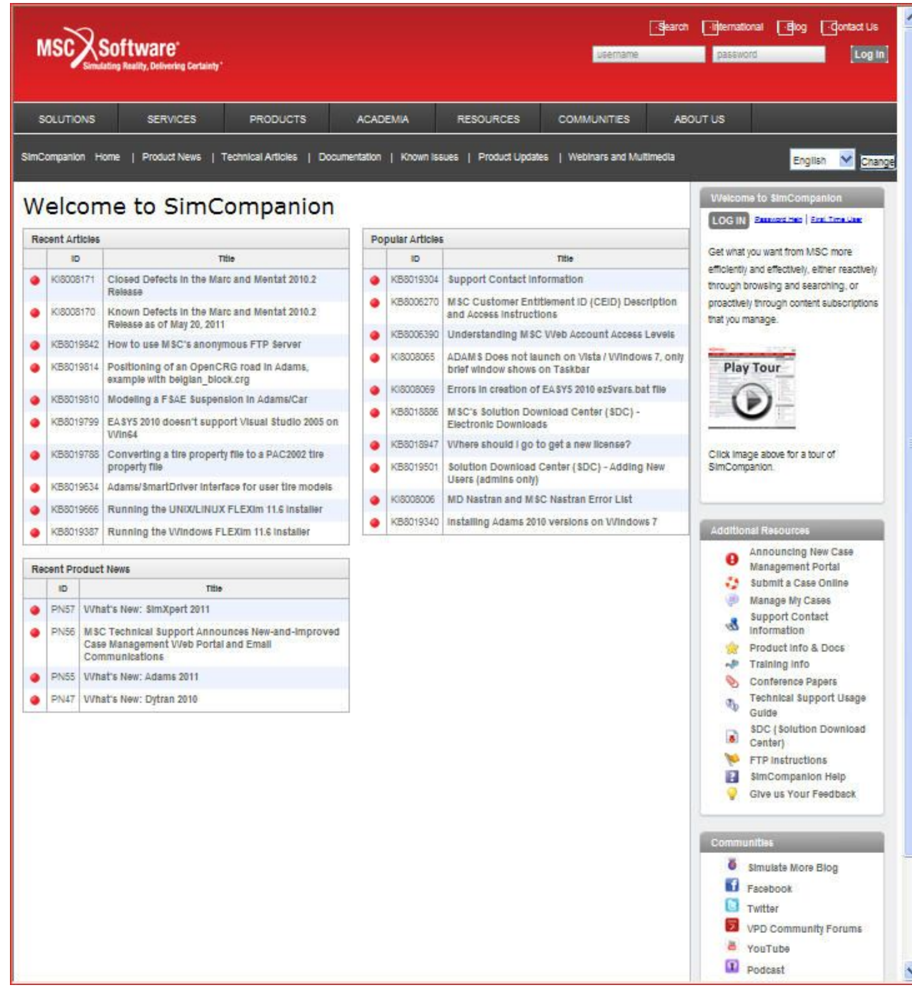
RANDOM

FINITE ELEMENT DYNAMIC MODELING CONSIDERATIONS

- Frequency range
- Grid points/constraints/elements
- Linear versus nonlinear behavior
- Whole system versus superelement models
- Interaction with adjacent media
- Test/measured data integration
- Damping

SIMCOMPANION

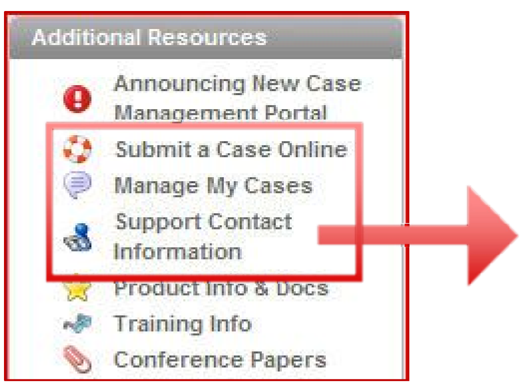
- **One stop for full online support**
- **Find answers to your questions**
- **Search across ALL content**
- **Subscribe to e-mail notification**
- **Single sign-on to ALL content**
- **Access to other support resources**
  - Case Management Portal
  - Discussion Forums
  - Training Information



http://simcompanion.mscsoftware.com

SIMCOMPANION

- **Personalized Support via the following channels**
  - Web
    - Submit a Case Online
    - Manage My Cases
  - Email
    - List of Addresses in **Support Contact Information**
  - Phone
    - List of Phone Numbers in **Support Contact Information**



Web:

http://support.mscsoftware.com/servicerequest

Email:

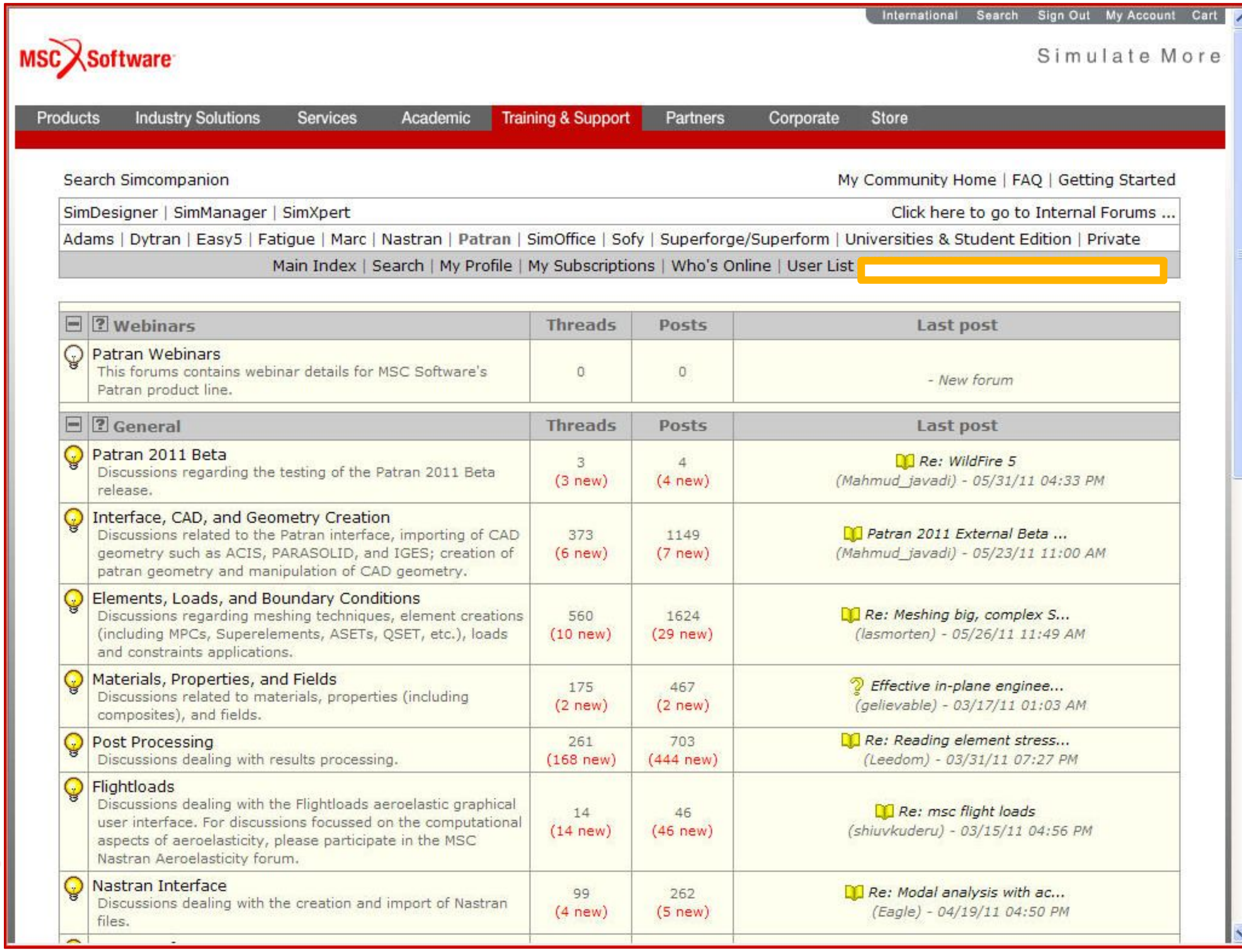
English	
- MD Nastran or MSC Nastran	mscnastran.support@mscsoftware.com
- MD Adams or Adams	mscadams.support@mscsoftware.com
- Patran	mscpatran.support@mscsoftware.com
- Dytran	mscdytran.support@mscsoftware.com
- Easy5	easy5.support@mscsoftware.com
- Fatigue	mscfatigue.support@mscsoftware.com
- Marc	mscmarc.support@mscsoftware.com
- Mvision	mscmvision.support@mscsoftware.com
- Sinda	mscsinda.support@mscsoftware.com
- Sofy	mscsofy.support@mscsoftware.com
- SimDesigner	simdesigner.support@mscsoftware.com
- MSC SimManager	simmanager.support@mscsoftware.com
- MSC SimXpert	simxpert.support@mscsoftware.com
Chinese (Simplified)	support.cn@mscsoftware.com
Chinese (Traditional)	support.tw@mscsoftware.com
Dutch	support.be@mscsoftware.com
French	support.fr@mscsoftware.com
German	support.de@mscsoftware.com
Italian	support.it@mscsoftware.com
Japanese	support.jp@mscsoftware.com
Korean	support.kr@mscsoftware.com
Nordic	support.no@mscsoftware.com
Portuguese	suporte.mscbrasil@mscsoftware.com
Russian	support.ru@mscsoftware.com
Spanish	support.es@mscsoftware.com

Phone:

Belgium and Luxembourg	+31 182 536444
Brazil	0800-891-4346
China - Beijing	+86-10-82607000
China - Shanghai	+86-21-63326655
China - Chengdu	+86-28-86199275/6
China - Shenzhen	+86-755-23811895
Czech Republic	+420 54517 6106
DACH	+49 89 431 987 277
Denmark	+45 61 22 32 00
Finland	0800-9-14709
France	+33 5 34 60 44 80

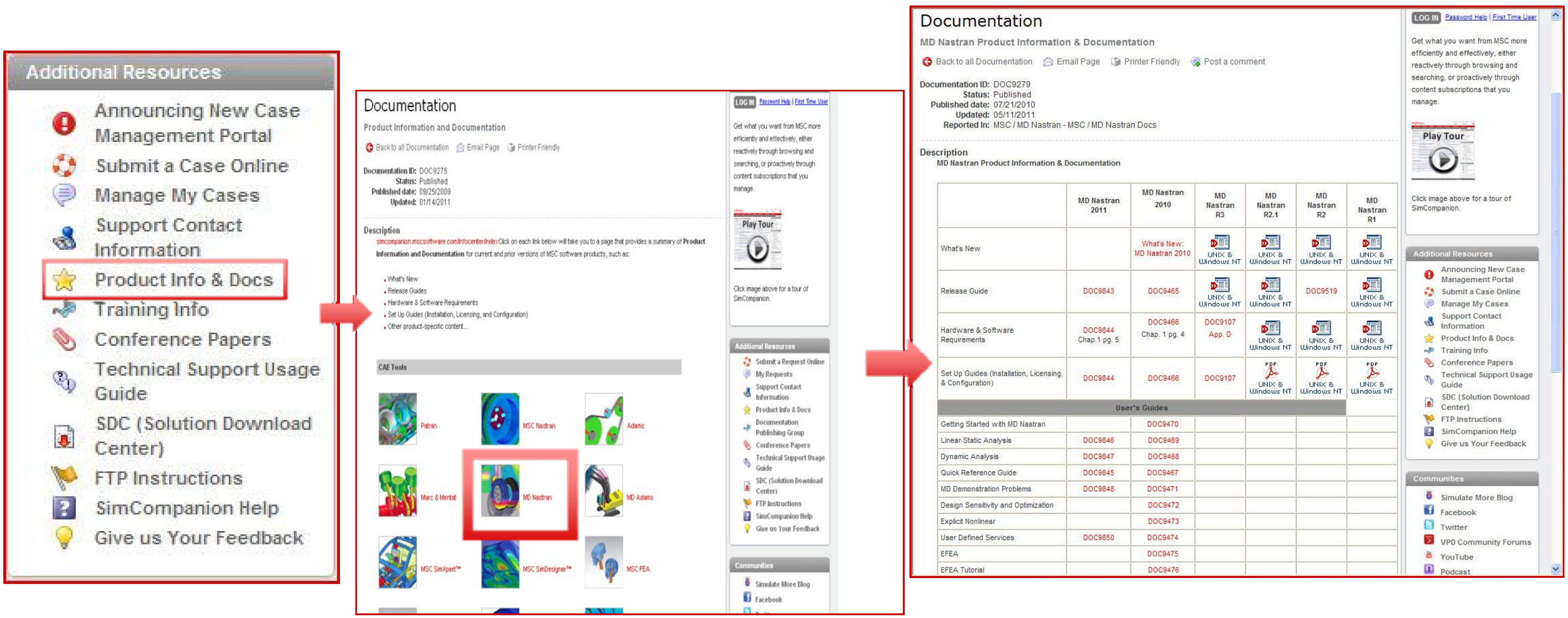
SIMCOMPANION

- **Access to Communities**
  - VPD Community Discussion Forums
  - Subscribe to discussion communities of interest



SIMCOMPANION

- **Product Info and Docs**
  - Access to all Product Documentation



MSC NASTRAN DOCUMENTATION

- **MSC Nastran Documentation**
  - Quick Reference Guide
  - Release Guide
  - Installation and Operation Release Guide
- **User's Guides**
  - Linear Static Analysis
  - Dynamic Analysis
  - Design Sensitivity and Optimization
  - DMAP Programmer's Guide
  - Numerical Methods
  - Aeroelastic Analysis
  - Thermal Analysis
  - Superelement
  - Implicit Nonlinear (SOL 600)
  - Explicit Nonlinear (SOL 700)
  - SOL 400

MSC NASTRAN DOCUMENTATION

- **Other Documentation**
  - Reference Manual
  - Common Questions and Answers
  - Bibliography

http://www.mscsoftware.com/support/prod_support/nastran/biblio/index.cfm

TEXT REFERENCES ON DYNAMIC ANALYSIS

- **W. C. Hurty and M. F. Rubinstein, *Dynamics of Structures*, Prentice-Hall, 1964.**
- **R. W. Clough and J. Penzien, *Dynamics of Structures*, McGraw-Hill, 1975.**
- **S. Timoshenko, D. H. Young, and W. Weaver, Jr., *Vibration Problems in Engineering*, 4th Ed., John Wiley & Sons, 1974.**
- **K. J. Bathe and E. L. Wilson, *Numerical Methods in Finite Element Analysis*, Prentice-Hall, 1976.**
- **J. S. Przemieniecki, *Theory of Matrix Structural Analysis*, McGraw-Hill, 1968.**
- **C. M. Harris and C. E. Crede, *Shock and Vibration Handbook*, 2nd Ed., McGraw-Hill, 1976.**
- **L. Meirovitch, *Analytical Methods in Vibrations*, MacMillan, 1967.**
- **L. Meirovitch, *Elements of Vibration Analysis*, McGraw-Hill, 1975.**
- **M. Paz, *Structural Dynamics Theory and Computation*, Prentice-Hall, 1981.**

TEXT REFERENCES ON DYNAMIC ANALYSIS

- W. T. Thomson, *Theory of Vibrations with Applications*, Prentice-Hall, 1981.
- R. R. Craig, *Structural Dynamics: An Introduction to Computer Methods*, John Wiley & Sons, 1981.
- S. H. Crandall and W. D. Mark, *Random Vibration in Mechanical Systems*, Academic Press, 1963.
- J. S. Bendat and A. G. Piersol, *Random Data Analysis and Measurement Techniques*, 2nd Ed., John Wiley & Sons, 1986.
- R. Gasch, K. Knothe: Strukturdynamik, Band 1+2. Springer, 1987

SECTION 2

DYNAMIC MODELING INPUT

MSC NASTRAN INPUT FILE SETUP

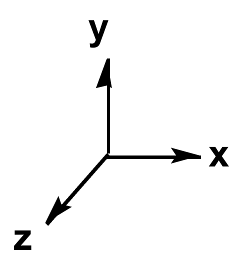
- **FMS and NASTRAN Statements - File allocations and system cell**
  - Executive Control Section
    - Solution type, time allowed, system diagnostics
    - CEND - Required Delimiter
  - Case Control Section
    - Output requests, selects certain Bulk Data items
    - BEGIN BULK - Required Delimiter
  - Bulk Data Section
    - Structural model definition, solution conditions
    - ENDDATA - Required Delimiter

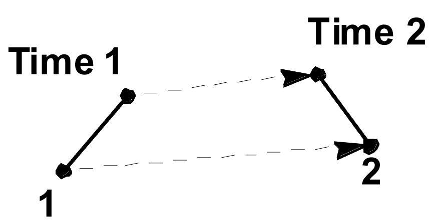
MSC NASTRAN BULK DATA ENTRY FORMAT

- **Fixed field format**
  - GRID^^^^2^^^^^^^3^^^^^^^1.0^^^^^-2.0^^^^3.0^^^^^^^^^^^^^316
- **Free format, (showing same entry as above)**
  - GRID,2,3,1.0,-2.0,3.0,,316
- **Examples in this course will use free format**

FINITE ELEMENT ANALYSIS

- **The real world is not comprised exclusively of 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 structure as a discrete entity**
- **Each GRID may have six DOFs:**
  - translation in the X, Y, and Z directions
  - rotations about the X, Y, and Z axes





- **Bookkeeping is done via the matrices that define the relationships between the DOFs**

COMMONLY USED ELASTIC ELEMENTS

One-Dimensional Geometry		Number of DOFs	
ROD	Pin-ended rod	4	2 Grids x 2 DOFs
BAR	Prismatic beam	12	2 Grids x 6 DOFs
BEAM	Straight beam with warping	14	2 Grids x(6+1)DOFs
BEND	Curved beam, pipe, or elbow	12	2 Grids x 6 DOFs
Two-Dimensional Geometry			
TRIA3 / TRIAR	Triangular plate	18	3 Grids x "6" DOFs
QUAD4/ QUADR	Quadrilateral plate	24	4 Grids x "6" DOFs
SHEAR	4-sided shear panel	8	4 Grids x 2 DOFs
TRIA6	Triangular plate with midside nodes	30	6 Grids x 5 DOFs
QUAD8	Quadrilateral plate with midside nodes	40	8 Grids x 5 DOFs
Three-Dimensional Geometry			
HEXA	Solid with six quadrilateral faces	24-60	8-20 Grids x 3 DOFs
TETRA	Solid with four triangular faces	12-30	4-10 Grids x 3 DOFs
PENTA	Solid with two triangular faces and three quadrilateral faces	18-45	6-15 Grids x 3 DOFs
Zero-Dimensional Geometry			
ELAS	Simple spring connecting two degrees of freedom	2	1-2 Grids x 1DOF
BUSH	Frequency-dependent spring/damper connecting up to six degrees of freedom	6	1-2 Grid x 6 DOFs

COUPLED VERSUS LUMPED MASS

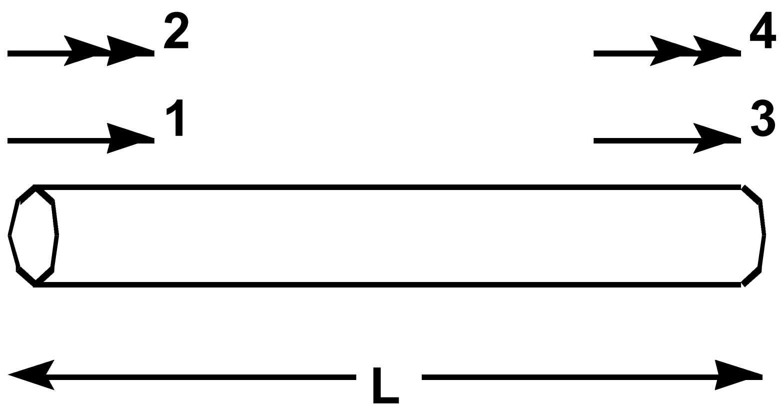
- **Coupled mass is generally more accurate than lumped mass**
- **Lumped mass is preferred for computational speed in dynamic analysis**
- **User-selectable coupled mass matrix for elements**
  - PARAM, COUPMASS, 1 to select coupled mass
  - The default is lumped mass
- **Elements which have either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

COUPLED VERSUS LUMPED MASS

- **Lumped mass contains only diagonal, translational components (no rotational ones)**
- **Coupled mass contains off-diagonal translational components as well as rotations for BAR (though no torsion for BAR, by default), BEAM, and BEND elements.**
  - Setting system cell 398 to 1 generates torsional mass for BAR
    - **NASTRAN SYSTEM(398) = 1 or NASTRAN BARMASS = 1**
  - Setting system cell 398 to 1 and param,coupmass,1 generates consistent axial mass for BAR
- **Neglected inertia may result in massless mechanisms.**

ROD FINITE ELEMENT

- L = Length
- A = Area
- J = Torsional Modulus
- E = Young’s Modulus
- G = Shear Modulus
- ρ = Mass Density
- Iρ = Polar Moment of Inertia



ROD FINITE ELEMENT

- **Stiffness matrix:**

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

- **Classical Consistent mass:**

$$\mathbf{m} = \rho AL \begin{bmatrix} \frac{1}{3} & 0 & \frac{1}{6} & 0 \\ 0 & \frac{I}{3A} & 0 & \frac{I}{6A} \\ \frac{1}{6} & 0 & \frac{1}{3} & 0 \\ 0 & \frac{I}{6A} & 0 & \frac{I}{3A} \end{bmatrix} \quad \text{where} \quad I = \int r^2 dA$$

ROD FINITE ELEMENT

- **Classical and MSC Nastran lumped mass:**

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

- **MSC Nastran coupled mass:**

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

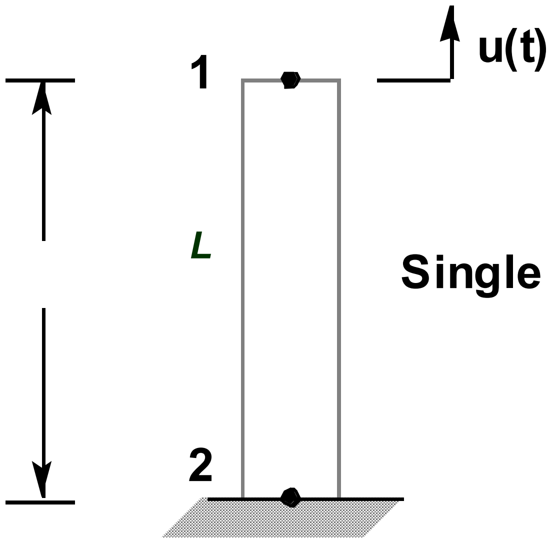
$5/12 = ½(1/2 + 1/3)$

$1/12 = ½(\ 0 + 1/6)$

- **The axial translational terms represent the average of lumped mass and classical consistent mass.**
- **This average is found to be best for ROD and BAR elements.**

JUSTIFICATION FOR MSC NASTRAN COUPLED MASS CONVENTION

- **Consider a fixed-free rod**



Single Element Model

- **Exact quarter-wave natural frequency**

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

JUSTIFICATION FOR MSC NASTRAN COUPLED MASS CONVENTION

- **Different approximations**
  - Lumped mass

$$\omega_{LM} = \sqrt{2}\frac{\sqrt{E/\rho}}{L} = 1.414\frac{\sqrt{E/\rho}}{L} \qquad \textbf{(–10\%)}$$

  - Classical consistent mass

$$\omega_{C} = \sqrt{3}\frac{\sqrt{E/\rho}}{L} = 1.732\frac{\sqrt{E/\rho}}{L} \qquad \textbf{(+10\%)}$$

- **MSC Nastran**
  - Coupled mass

$$\omega_{N} = \sqrt{12/5}\frac{\sqrt{E/\rho}}{L} = 1.549\frac{\sqrt{E/\rho}}{L} \qquad \textbf{(–1.4\%)}$$

MASS UNITS

- **MSC Nastran assumes you are providing a consistent set of units**
- **Weight units may be input instead of mass units if this is more convenient. You must then convert them to mass units using PARAM,WTMASS.**
- **Weight-to-mass conversion:**
  - Mass = (1/G) Weight, (where G = Gravity Acceleration)
  - Mass Density = (1/G) Weight Density
- **PARAM, WTMASS, factor performs conversion with factor = 1/G**
- **The default value for factor is 1.0**
- **Example:**
  - Input RHO = 0.3 for steel weight density.
  - Use PARAM, WTMASS, 0.00259, for G = 386.4 in/sec^2
  - This will multiply the value of RHO (0.3) by .000259 to convert it to the steel mass density

MASS UNITS

- **PARAM,WTMASS is used once per run and multiplies all weight/mass input (including MASSi, CONMi, and nonstructural mass input)**
- **Do not mix input types. Use all mass or all weight inputs.**

MASS INPUT

- **Material density**
  - MATi entries

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

- **Scalar mass**
  - CMASSi, PMASS
- **Grid point mass**
  - CONM1 (6x6 mass matrix) - The user defines half of the terms, symmetry is assumed
  - CONM2 (concentrated mass)

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

MASS INPUT

- **Structural mass (such as CONM2) are always included in the model**
- **Nonstructural mass**
  - Mass input on element property entry which is not associated with geometric properties of element. Input as mass/length for line elements and mass/area for elements with 2-D geometry.
  - Non structural mass can be specified on many property entries (for example, NSM on the PSHELL entry)
  - This type of nonstructural mass is always included in the model
  - Some nonstructural mass are Case Control selectable
  - NSM Case Control selects NSM, NSM1, NSML, NSML1 entries

Bulk Data Entries

NONSTRUCTURAL MASS ENHANCEMENTS

- **Five nonstructural mass Bulk Data entries are available:**
  - NSM, NSM1, NSML, NSML1, and NSMADD
- **Distribute nonstructural mass by element lists or specific property lists associated with property entries**
- **Case Control selectable with NSM Case Control command**
- **NSM callout can be different between superelements—but only one per superelement**

NONSTRUCTURAL MASS ENHANCEMENTS

- **The NSML and NSML1 entries compute a nonstructural mass coefficient value for “area” elements (for example, CQUAD4)**

$$NSM\_value = \frac{Lumped\_non\_structural\_mass\_value}{\Sigma_{elements} AREA}$$

- **The NSML and NSML1 entries compute a nonstructural mass coefficient value for “line” elements (for example, CBAR)**

$$NSM\_value = \frac{Lumped\_non\_structural\_mass\_value}{\Sigma_{elements} LENGTH}$$

- **NSML and NSML1 cannot mix “area” and “line” element on the same entry**
- **NSML and NSML1 are then converted to NSM and NSM1**

NONSTRUCTURAL MASS ENHANCEMENTS

- **NSM and its alternate form NSM1 allows the user to allocate an NSM_value to selected sets of elements.**
- **Example:**
  - Type = PSHELL, PCOMP, PSHEAR, PBAR, PBARL, PBEAM, PBEAML, PBEND, PROD, CONROD, PTUBE, PCONEAX, PRAC2D

	SID	TYPE	ID	VALUE	ID	VALUE	ID	VALUE	
NSM	3	PSHELL	15	.022					

	SID	TYPE	VALUE	ID	ID	ID	ID	ID	
NSM1	3	PSHELL	.022	15					

- **Either one of the above examples assigns a 0.022 nonstructural mass per unit area to PSHELL ID 15. Additive if both are present.**

CONM2 ENTRY

CONM2 **Concentrated Mass Element Connection, Rigid Body Form**

Defines a concentrated mass at a grid point.

Format:

1	2	3	4	5	6	7	8	9	10
CONM2	EID	G	CID	M	X1	X2	X3		
	I11	I21	I22	I31	I32	I33			

Example:

CONM2	2	15	6	49.7					
	16.2		16.2			7.8			

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
G	Grid point identification number. (Integer > 0)
CID	Coordinate system identification number. For CID of -1; see X1, X2, X3 below. (Integer $\geq$ -1; Default = 0)
M	Mass value. (Real)
X1, X2, X3	Offset distances from the grid point to the center of gravity of the mass in the coordinate system defined in field 4, unless CID = -1, in which case X1, X2, X3 are the coordinates, not offsets, of the center of gravity of the mass in the basic coordinate system. (Real)
Iij	Mass moments of inertia measured at the mass center of gravity in the coordinate system defined by field 4. If CID = -1, the basic coordinate system is implied. (Real)

CONM2 ENTRY

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. For a more general means of defining concentrated mass at grid points, see the CONM1 entry description.
3. The continuation is optional.
4. If CID = -1, offsets are internally computed as the difference between the grid point location and X1, X2, X3. The grid point locations may be defined in a nonbasic coordinate system. In this case, the values of Iij must be in a coordinate system that parallels the basic coordinate system.
5. The form of the inertia matrix about its center of gravity is taken as:

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

where

$$\begin{aligned} M &= \int \rho \, dV \\ I11 &= \int \rho (x_2^2 + x_3^2) dV \\ I22 &= \int \rho (x_1^2 + x_3^2) dV \\ I33 &= \int \rho (x_1^2 + x_2^2) dV \\ I21 &= \int \rho x_1 x_2 dV \\ I31 &= \int \rho x_1 x_3 dV \\ I32 &= \int \rho x_2 x_3 dV \end{aligned}$$

CONM2 ENTRY

Remarks:

and x_1, x_2, x_3 are components of distance from the center of gravity in the coordinate system defined in field 4. The negative signs for the off-diagonal terms are supplied automatically. A warning message is issued if the inertia matrix is nonpositive definite, since this may cause fatal errors in dynamic analysis modules.

6. If CID $\geq$ 0, then X1, X2, and X3 are defined by a local Cartesian system, even if CID references a spherical or cylindrical coordinate system. This is similar to the manner in which displacement coordinate systems are defined.
7. See Grid Point and Coordinate System Definition (p. 33) in the *SOL 700 Reference Manual* for a definition of coordinate system terminology.

DEGREES OF FREEDOM SET OPERATIONS

Grid Set (G) DOF = N + M

- N: all degrees of freedom not constrained by multipoint constraint
- M: all degrees of freedom eliminated by multipoint constraints

N = F + S

- F: unconstrained structural degrees of freedom
- S: all degrees of freedom eliminated by single point constraints

F = A + O

- A: the analysis set used in eigensolution (Static Condensation)
- O: Degrees of freedom omitted by structural matrix partitioning

A = L + R

- L: degrees of freedom remaining after the reference degrees of freedom are removed
- R: reference degrees of freedom used to determine free body motion

DEGREES OF FREEDOM SET OPERATIONS

- **Reverse the process on previous slide for the data recovery to G-set**
- **The set definition defined in the previous slide is probably what you will encounter in most situations**
- **See the "Degrees-of-Freedom Set Definition" section of the *Quick Reference Guide* for a complete set definition**

SECTION 3

NORMAL MODE ANALYSIS

REASONS TO COMPUTE NATURAL FREQUENCIES AND NORMAL MODES

- **Assess the dynamic characteristics of the structure. For example, if rotating machinery is going to be installed on a certain structure, to avoid excessive vibrations, it might be necessary to see if the frequency of the rotating mass is close to one of the natural frequencies of the structure.**
- **Assess the possible dynamic amplification of the loads**
- **Use the natural frequencies and normal modes to guide subsequent dynamic analysis (transient response, response spectrum analysis), such as what should the appropriate time step be for integrating the equations of motion in transient analysis?**
- **Use the natural frequencies and mode shapes for subsequent dynamic analysis, that is transient analysis of the structure using modal expansion**
- **Guide the experimental analysis of structures, such as location of accelerometers, etc.**
- **Evaluate the design changes**

THEORETICAL RESULTS

- **Consider:**

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

- **Assume a harmonic solution of the form:**

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

  – Physically, this means that all the coordinates perform synchronous motions. The system configuration does not change its shape during motion, only its amplitude.

- **From Equation 2:**

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

- **Substituting Equations 2 and 3 into Equation 1, we obtain:**

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

- **Which, after dividing by e^{iwt}, simplifies to:**

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

- **This is then an eigenvalue problem**

THEORETICAL RESULTS

- **Therefore, there are two cases:**
  1. If det$([K]-\omega^2[M]) \neq 0$, the only possibility (from Equation 4) is

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

     - This is the so-called trivial solution and is not interesting from a physical point of view.
  2. Otherwise, we need det $([K]-\omega^2[M]) = 0$ in order to have a nontrivial solution for $\{\phi\}$
- **The eigenvalue problem reduces to solve the following:**

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

or

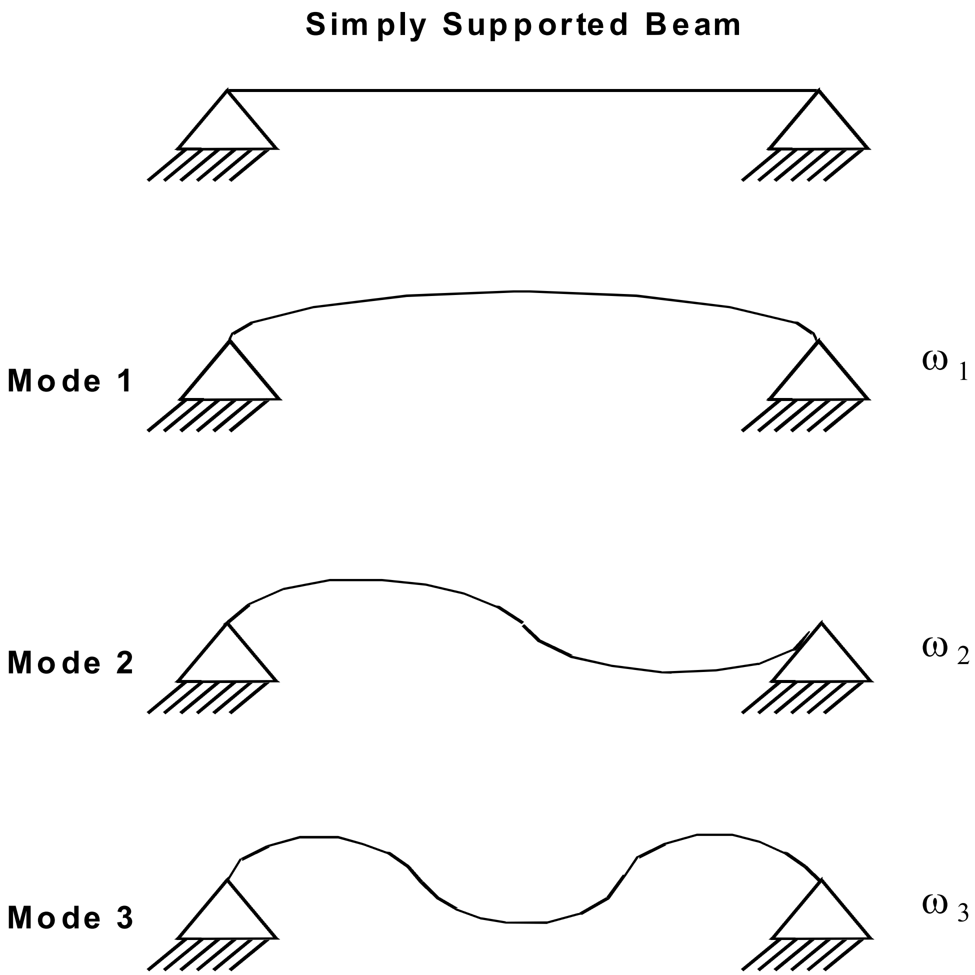
$$\det([K]-\lambda[M]) = 0$$

– where $\lambda = \omega^2$

THEORETICAL RESULTS

- **If the structure has N degrees of freedom with attached mass, then there will be N ω_s that are solutions of the eigenvalue problem. These ω_s (ω_1,ω_2,…,ω_n) are natural frequencies, characteristic frequencies, fundamental frequencies, or resonance frequencies.**
- **The eigenvector $\{\phi\}_j$ associated with the natural frequency $\{\omega\}_j$ is called the normal mode or mode shape. The normal mode corresponds to the deflected shape patterns of the structure.**
- **When a structure is vibrating, its shape at any given time is a linear combination of its normal modes**

THEORETICAL RESULTS



etc.

IMPORTANT FACTS AND RESULTS REGARDING NORMAL MODES AND NATURAL FREQUENCIES

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

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

and

$$\{\phi_i\}^T[K]\{\phi_j\} = 0 \qquad If\ i \neq j$$

also

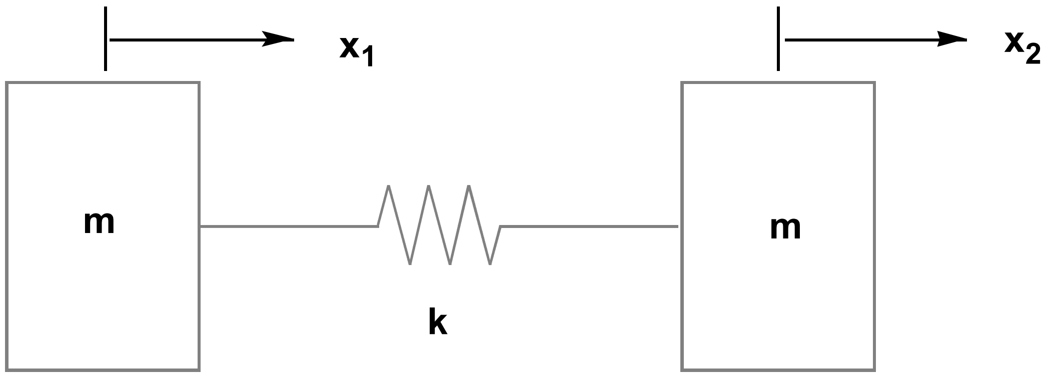
$$\omega_j^2 = \frac{\{\phi_j\}^T[K]\{\phi_j\}}{\{\phi_j\}^T[M]\{\phi_j\}} \qquad \text{Rayleigh Quotient}$$

  – The natural frequencies ($\omega_1, \omega_2, \ldots$) are expressed in radians/seconds. They can also be expressed in hertz (cycles/seconds), using

$$f_j(hertz) = \frac{\omega_j(radian/\sec ond)}{2\pi}$$

IMPORTANT FACTS AND RESULTS REGARDING NORMAL MODES AND NATURAL FREQUENCIES

- **Example: The following unconstrained structure has a rigid body mode**

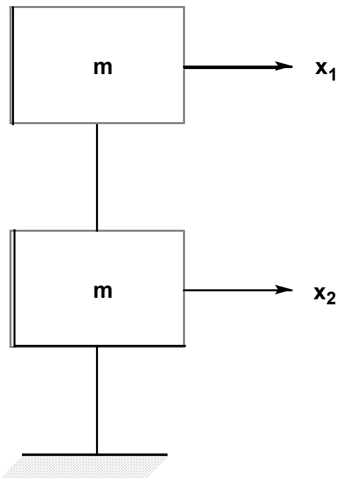


- **If a structure is not totally constrained, that is if it admits a rigid body mode (stress-free mode) or a mechanism, then at least one natural frequency will be zero**

$$\omega_1 = 0 \qquad \{\phi_1\} = \begin{Bmatrix} 1 \\ 1 \end{Bmatrix}$$

IMPORTANT FACTS AND RESULTS REGARDING NORMAL MODES AND NATURAL FREQUENCIES

- **The scaling of normal modes is arbitrary. For example:**



represent the same "mode of vibration"

$$\{\phi_1\} = \begin{Bmatrix} 1 \\ 0.5 \end{Bmatrix}, \quad \{\phi_1\} = \begin{Bmatrix} 300 \\ 150 \end{Bmatrix}, \quad \{\phi_1\} = \begin{Bmatrix} 0.66 \\ 0.33 \end{Bmatrix}$$

IMPORTANT FACTS AND RESULTS REGARDING NORMAL MODES AND NATURAL FREQUENCIES

- **For practical considerations, modes should be normalized by a chosen convention**
- **In MSC Nastran there are three normalization choices (except when using Lanczos):**
  - The unit value of generalized mass (default)

$$\{\phi_i\}^T [M] \{\phi_i\} = 0$$

  - The unit value of the largest A-set component in each mode
  - The unit value of a specific component (not recommended)
- **In the Lanczos method, normalization is to a unit value of generalized mass and to a unit value of the largest component**

ADDITIONAL MODAL PROPERTIES

- **Since strains, internal loads, and stresses develop when a structure deforms, we may recover additional useful modal information utilizing:**
  - Strain-displacement relationships

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

  - Stress-strain relationships

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

  - Static force - displacement relationships

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

  - Element strain energy relationships

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

ADDITIONAL MODAL PROPERTIES

- **Thus, for a given modal displacement we have** $\{u\} = [\phi_i]\xi_i$
  - Modal strain

$$\{\varepsilon_\phi\}_i = \{[K_{\varepsilon u}]\{\phi_i\}\}\xi_i$$

  - Modal stress

$$\{\sigma_\phi\}_i = \{[K_{\sigma\varepsilon}][K_{\varepsilon u}]\{\phi_i\}\}\xi_i$$

  - Modal force

$$\{P_\phi\}_i = \{[K]\{\phi_i\}\}\xi_i$$

  - Modal strain energy

$$Ve_{\phi i} = \left(\frac{1}{2}\right)\{\phi_{e_i}\}^T[K_{ee}]\{\phi_{e_i}\}\xi_i^2$$

METHODS OF COMPUTATION

- **MSC Nastran provides 3 types of methods for eigenvalue extraction:**
  - Tracking method
    - Eigenvalues (or natural frequencies) are determined one at a time using an iterative technique. Two variations of the inverse power method are provided, INV and SINV. This approach is more convenient when few natural frequencies are to be determined. In general, SINV is more reliable than INV.
  - Transformation method
    - The original eigenvalue problem is transformed to the form where:

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

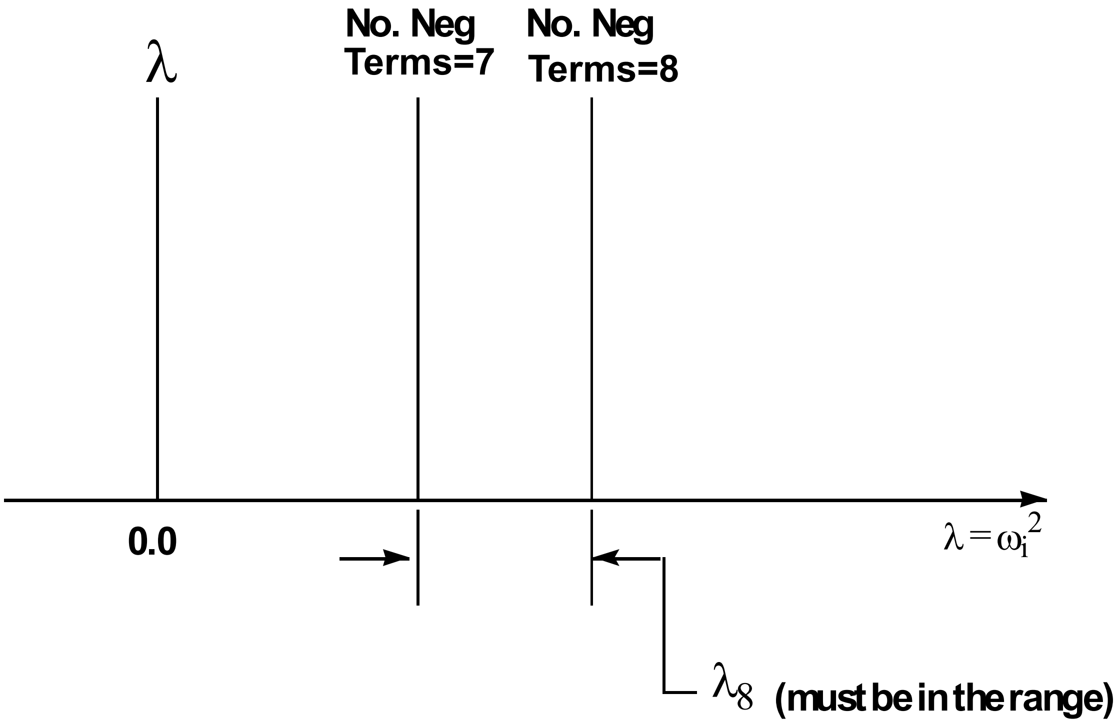
$$[A]\{\phi\}=\lambda\{\phi\}$$

$$[A]=[M]^{-1}[K]$$

    - Then the matrix A is transformed into a tridiagonal matrix using either the Givens technique or the Householder technique. Finally, all the eigenvalues are extracted at once using the QR algorithm. Two variations of the Givens technique and two variations of the Householder technique are provided: GIV, MGIV, HOU, and MHOU. These methods are more efficient when a large proportion of eigenvalues are needed.
  - Lanczos Method (recommended method)
    - This method is a combined tracking-transformation method

STURM SEQUENCE THEORY

- **Choose** λ
- **Factor** $[K - \lambda_i M]$ **into** $[L][D][L^T]$
- **The number of negative terms on the factor diagonal is the number of eigenvalues below:**



LANCZOS METHOD

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

EIGRL - USER INTERFACE FOR LANCZOS METHOD

EIGRL Real Eigenvalue Extraction Data, Lanczos Method

Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method.

Format:

Default = 7, can be increased up to 15

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

Example:

EIGRL	1	0.1	3.2	10					
	NORM=MAX NUMS=2								

Field	Contents
SID	Set identification number. (Unique Integer > 0)
V1, V2	For vibration analysis: frequency range of interest. For buckling analysis: eigenvalue range of interest. See Remark 4. (Real or blank, $-5 \times 10^{16} \le V1 < V2 \le 5. \times 10^{16}$)
ND	Number of roots desired. See Remark 4. (Integer > 0 or blank)
MSGLVL	Diagnostic level. ($0 \le$ Integer ≤ 4; Default = 0)
MAXSET	Number of vectors in block or set. Default is machine dependent. See Remark 14.

EIRGL – USER INTERFACE FOR LANCZOS METHOD

Field	Contents	
SHFSCL	Estimate of the first flexible mode natural frequency. See Remark 10. (Real or blank)	
NORM	Method for normalizing eigenvectors (Character: "MASS" or "MAX")	
	MASS	Normalize to unit value of the generalized mass. Not available for buckling analysis. (Default for normal modes analysis.)
	MAX	Normalize to unit value of the largest displacement in the analysis set. Displacements not in the analysis set may be larger than unity. (Default for buckling analysis.)
ALPH	Specifies a constant for the calculation of frequencies (Fi) at the upper boundary segments for the parallel method based on the following formula. See Remark 13. (Real > 0.0; Default = 1.0): $Fi = (V2 - V1)\dfrac{1 - \mathrm{ALPH}^i}{1. - \mathrm{ALPH}^{\mathrm{NUMS}}}$	
NUMS	Number of frequency segments for the parallel method. (Integer > 0; Default = 1)	
Fi	Frequency at the upper boundary of the i-th segment. See Remark 13. (Real or blank; $V1 < F1 < F2 < \ldots F15 < V2$)	
option_i= value_i	Assigns a value to the fields above except for SID. ALPH, NUMS, and Fi must be specified in this format. V1, V2, ND, MSGLVL, MAXSET, SHFSCL, and NORM may be specified in this format as long as their corresponding field is blank in the parent entry.	

Note: For rest of EIGRL entry, see MSC Nastran 2013 QRG

EIGR - USER INTERFACE FOR ALL METHODS

EIGR — Real Eigenvalue Extraction Data

Defines data needed to perform real eigenvalue analysis.

Format:

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

Example:

EIGR	13	LAN				12			

Field	Contents
SID	Set identification number. (Unique Integer > 0)
METHOD	Method of eigenvalue extraction. (Character)
	Modern Methods:
	LAN Lanczos Method
	AHOU Automatic selection of HOU or MHOU method. See Remark 13.

EIRG - USER INTERFACE FOR ALL METHODS

Field	Contents	
NORM	Method for normalizing eigenvectors. (Character: "MASS," "MAX," or "POINT"; Default = "MASS")	
	MASS	Normalize to unit value of the generalized mass. (Default)
	MAX	Normalize to unit value of the largest component in the analysis set.
	POINT	Normalize to a positive or negative unit value of the component defined in fields 3 and 4. The POINT option is not supported for METHOD=LAN. (Defaults to "MASS" if defined component is zero.)
G	Grid or scalar point identification number. Required only if NORM = "POINT". (Integer > 0)	
C	Component number. Required only if NORM = "POINT" and G is a geometric grid point. ($1 \leq$ Integer ≤ 6)	

Note: For rest of EIGR entry, see MSC Nastran 2013 QRG

SOLUTION CONTROL FOR NORMAL MODES

- **Executive Control Section**
  - SOL 103
- **Case Control Section**
  - METHOD (required - selects EIGRL / EIGR entry)
- **Bulk Data Section**
  - EIGRL / EIGR (Lanczos method)

CASE CONTROL OUTPUT

- **Grid output**
  - DISPLACEMENT (or VECTOR)
  - GPFORCE
  - GPSTRESS
  - SPCFORCE
  - GPKE
- **Element output**
  - ELSTRESS (or STRESS)
  - ESE
  - EKE
  - ELFORCE (or FORCE)
  - STRAIN
- **Special entry**
  - OMODES – selects output for selective modes

EXERCISES

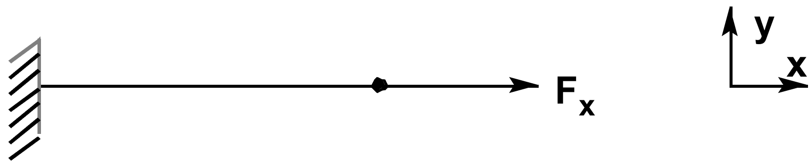
- **Perform Workshop #1, Modal Analysis of a Flat Plate**.

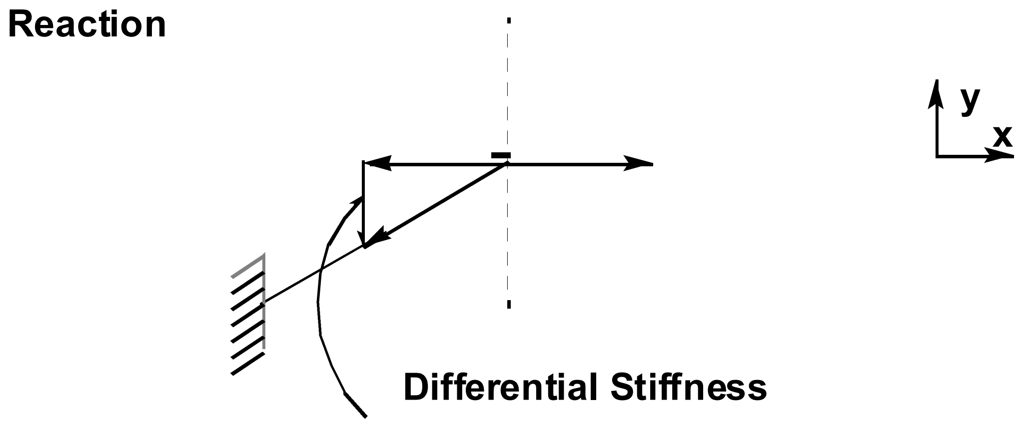
SECTION 4

NORMAL MODES OF PRELOADED STRUCTURES

NORMAL MODES WITH DIFFERENTIAL STIFFNESS

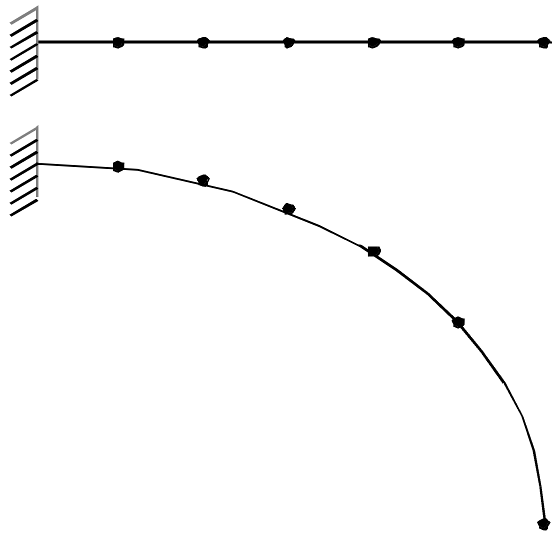
- **Calculate the modes of structures with preloads, large changes in geometry, and/or nonlinear materials**



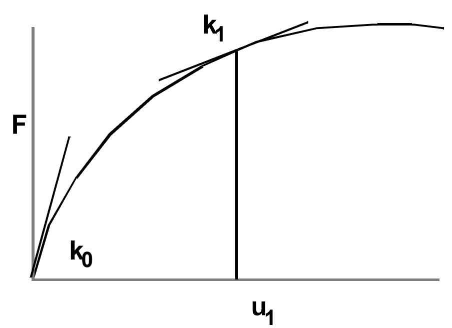


Preloaded Structure

NORMAL MODES WITH DIFFERENTIAL STIFFNESS



Large Geometry Changes



$K_0 = k1$

Nonlinear Material

NORMAL MODES WITH DIFFERENTIAL STIFFNESS

- **Procedure for obtaining frequencies of a preloaded structure**
  - Use SOL 103
  - Material must be linear
  - Two subcases are required
    - The first subcase is a static subcase calling out the preload
    - The second subcase calculates the modes with a method = x callout
      - The second subcase must also contain a statsub = y command, where y = subcase ID of the first subcase.

NORMAL MODES WITH DIFFERENTIAL STIFFNESS

- **Example**

```
$
SUBCASE 1
  load = 1
$
SUBCASE 2
  METHOD = 1
  statsub = 1
$
```

EXERCISES

- **Now perform Workshop #2, Normal Modes with Preload**

SECTION 5

REDUCTION IN 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 math model may be too big to be solved without reduction
  - The math model has more detail than required
  - allows the deletion of selective local modes
  - is more accurate (and probably cheaper) than constructing a separate, smaller dynamic model

REDUCTION METHODS FOR DYNAMICS AVAILABLE IN MSC NASTRAN

- **Guyan reduction (static condensation)**
  - Generally not recommended except for correlation with test data
- **Modal reduction**
- **Component mode synthesis (superelement option)**
  - Component mode synthesis is not discussed in this presentation.

STATIC CONDENSATION - INTERNAL CALCULATION

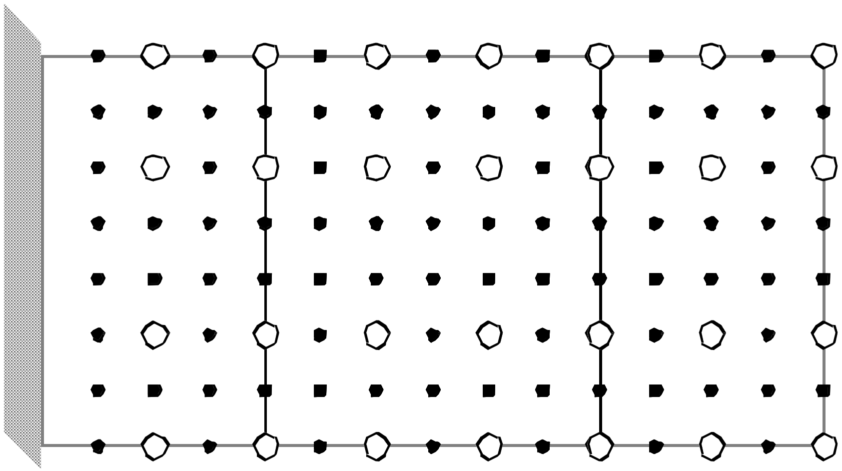
- **Let {u_f} be the set of the unconstrained (free) structural coordinates**
- **Partition**

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

Where,

u_a = analysis set

u_o = omitted set



O-Set → **Degrees of freedom removed during Guyan reduction**

A-Set → **User-selected dynamic degrees of freedom**

STATIC CONDENSATION - INTERNAL CALCULATION

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

$$\begin{pmatrix} K_{oo} & K_{oa} \\ K_{oa}^{T} & K_{aa} \end{pmatrix} \begin{Bmatrix} u_o \\ u_a \end{Bmatrix} = \begin{Bmatrix} P_o \\ P_a \end{Bmatrix} \qquad (1)$$

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

$$\{u_o\} = [G_{oa}]\{u_a\} \qquad (2)$$

  – Where: $[G_{oa}] = -[K_{oo}]^{-1}[K_{oa}]$

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

$$\{u_f\} = \begin{Bmatrix} u_o \\ \cdots\cdots \\ u_a \end{Bmatrix} = \underbrace{\begin{Bmatrix} G_{oa} \\ \cdots\cdots \\ I \end{Bmatrix}}_{\psi} \{u_a\} \qquad (3)$$

- **O-set is dependent upon the A-set via Equation 2. 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.**

STATIC CONDENSATION - INTERNAL CALCULATION

- **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

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

- **Dynamics problems are solved in terms of the reduced coordinates (A-set). O-set components are recovered using Equation 2 from previous slide.**
- **O-set mass, damping, and stiffness is spread to the A-set**
- **The largest cost 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 (that is the matrix bandedness is destroyed)**

STATIC CONDENSATION - INTERNAL CALCULATION

- **SUMMARY**
  - Separate free degrees of freedom (U_f) into the omitted set (U_0) 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

STATIC CONDENSATION – BULK DATA

- **Specify <u>either</u> the A-set or the O-set.**
  - A-set, use ASET entry

1	2	3	4	5	6	7	8	9	10
ASET	ID	C	ID	C	ID	C	ID	C	
ASET	1	123	2	12	4	1	5	1	

and/or

ASET1	C	G	G	G	G	G	G	G	
ASET1	123	1	2	3	4	5			

  - O-set, use OMIT, OMIT1 entry

1	2	3	4	5	6	7	8	9	10
OMIT	ID1	C1	ID2	C2	ID3	C3	ID4	C4	
OMIT	16	2	23	3516					

1	2	3	4	5	6	7	8	9	10
OMIT1	C	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	-ect-					
OMIT1	3	2	1	3	10	9	6	5	
	7	8							

- **Components not specified are placed in the complementary set. If both ASET and OMIT are present, components not specified are placed in the O-set.**

STATIC CONDENSATION – EXECUTIVE/CASE CONTROL

- **Static Condensation can be run in any solution.**
- **There are no special Case Control commands required.**

STATIC CONDENSATION - CONSIDERATIONS

- User effort in selecting A-set points
- Accuracy depends on the user’s skill in selecting A-set points
- Regardless of user’s skill, high accuracy requires a large number of A-set points (cost consideration), 2 to 5 times number of accurate modes wanted
- Stiffness reduction is exact; mass and damping reductions are only approximate
- No loss in accuracy of modes occurs when omitting massless degrees of freedom
- Errors are most pronounced in higher modes
- Local modes may be missed altogether
- Not generally recommended, except when performing test-analysis correlation (see NAS102B)

STATIC CONDENSATION - CONSIDERATIONS

- **The static condensation approximation may miss the local dynamic effects**

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

0

Local Dynamics Effects

Physical Variables

Static Transformation

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

Loads on O-set components

MODAL REDUCTION

- **All MSC Nastran linear dynamic solutions have two versions**
  - Direct - The solution is solved in terms of A-set coordinates
  - 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 ($\{\xi\}$)

Matrix of Mode Shapes ($[\phi_a]$)

- **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 and modal coordinates are handled internally.) (Note: E-set DOFs are not shown here for clarity.)**

$$\left[\phi_a^T\right]\left[M_{aa}\right]\left\{\phi_a\right\}\left\{\ddot{\xi}\right\}+\left[\phi_a^T\right]\left[B_{aa}\right]\left\{\phi_a\right\}\left\{\dot{\xi}\right\}+\left[\phi_a^T\right]\left[K_{aa}\right]\left\{\phi_a\right\}\left\{\xi\right\}=\left[\phi_a^T\right]\left\{P_a\right\}$$

- **If [ϕ] is mass normalized and there are no K2PP, M2PP, B2PP, or TF, then:**

$$\left[I\right]\left\{\ddot{\xi}\right\}+\left[\phi_a^T\right]B_{aa}\left\{\phi_a\right\}\left\{\dot{\xi}\right\}+\left[W^2\right]\left\{\xi\right\}=\left[\phi_a^T\right]\left\{P_a\right\}$$

- **<u>Note</u>: A-set matrices may be reduced matrices from Guyan reduction or GDR. Transformation from modal coordinates to the F-set would require two transformations.**

$$\left\{u_f\right\}=\left[\Psi\right]\left\{u_a\right\}$$
$$\left\{u_a\right\}=\left[\phi_a\right]\left\{\xi\right\}$$
$$\therefore\left\{u_f\right\}=\left[\Psi\right]\left\{\phi_a\right\}\left\{\xi\right\}$$

MODAL REDUCTION – SOLUTION CONTROL

- **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)

EXERCISES

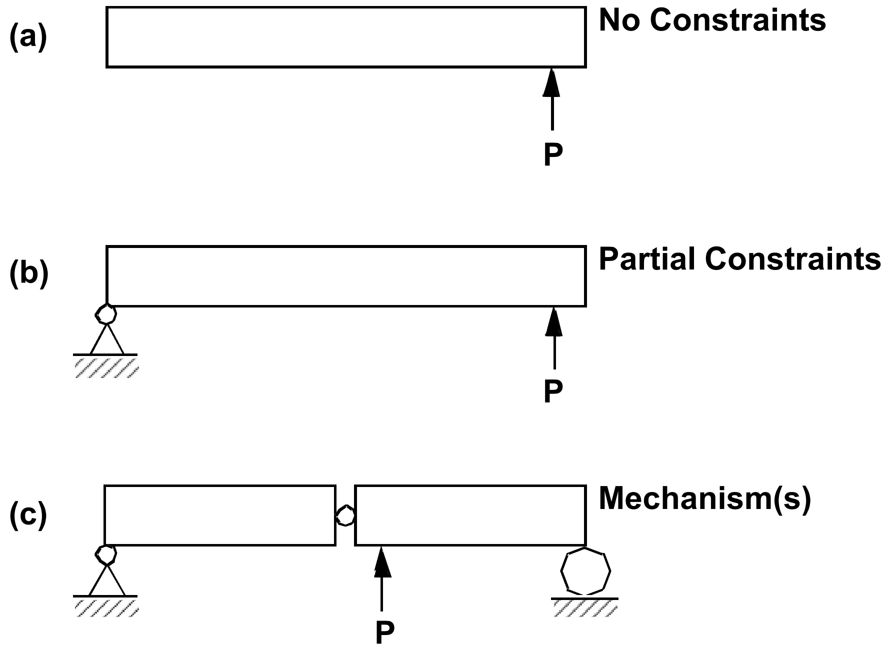
- **Now perform workshop 3, Normal Mode Analysis**

SECTION 6

RIGID BODY MODES

RIGID BODY MODES AND RIGID BODY VECTORS THEORETICAL CONSIDERATIONS

- **A structure has the ability to displace without developing internal loads of stresses if it is not sufficiently grounded**
- **Examples of this are:**



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

RIGID BODY MODES AND RIGID BODY VECTORS THEORETICAL CONSIDERATIONS

- **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, that is:**

$$\{\phi\}_{RIG}^{T}[M]\{\phi_{RIG}\} \succ 0$$

$$\{\phi\}_{RIG}^{T}[K]\{\phi_{RIG}\} = 0$$

- **The SUPORT entry <u>does not</u> 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 MSC Nastran by the following method for methods other than the Lanczos method: (The Lanczos method calculates rigid body modes directly)**
  - Step 1: "a"-set partitioning
  - Step 2: Solve for u_l in terms of u_r

$$\{u_a\} = \begin{Bmatrix} u_l \\ u_r \end{Bmatrix} \begin{matrix} \text{l- set} & \text{(left over set)} \\ \text{r- set} & \text{(SUPORT entry DOF)} \end{matrix}$$

- Note: Pr is not actually applied

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

CALCULATION OF RIGID BODY MODES

$$\{u_l\} = (D_m)\{u_r\}$$

- Where:

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

- This may be used to construct a set of rigid body vectors

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

– <u>Step 3</u>: Mass matrix operations

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

- 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:

$$[\phi_a]^T_{RIG} [K_{aa}] [\phi_a]_{RIG} = K_{rr} = 0^*$$

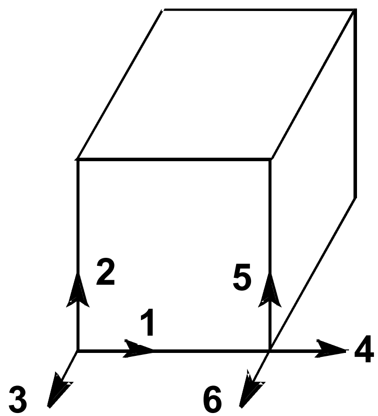
$$[\phi_a]^T_{RIG} [M_{aa}] [\phi_a]_{RIG} = [\not{M}_o]$$

Orthogonalized matrix

- Only if R-set DOFs truly support rigid body modes

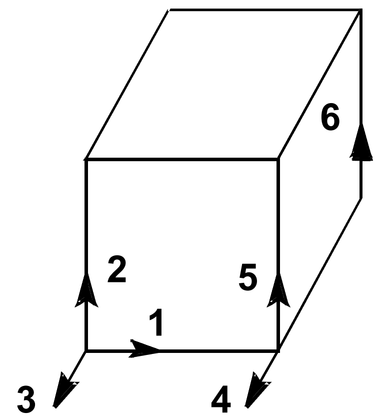
SELECTION OF “SUPORT” DEGREES OF FREEDOM

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



Bad Selection

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



Good Selection

CHECKING OF "SUPORT" DOFS

- **MSC Nastran calculates internal strain-energy (work) for each rigid body vector**

$$[x] = \begin{bmatrix} D^T & I \end{bmatrix} \begin{pmatrix} K_{ll} & K_{lr} \\ K_{rl} & K_{rr} \end{pmatrix} * \begin{Bmatrix} D \\ I_r \end{Bmatrix}$$

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

Rigid Body Vectors for l-set

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.**

- **MSC Nastran also calculates the rigid body error ratio**

$$\varepsilon = \frac{\left\| [x] \right\|}{\left\| K_{rr} \right\|}$$

  - where || || means Euclidian norm of the matrix $\left\| \quad \right\| = \sqrt{\sum_i \sum_j x_{ij}^2}$
  - Note: Only one value of ε is calculated using [X] and [K_{rr}] based on all SUPORT DOFs.

CHECKING OF "SUPORT" DOFS

- **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 chosen by the user. These quantities may be nonzero for any of the following reasons:**
  - Round-off error accumulation
  - The u_r set is over determined leading to redundant supports (high strain energy)
  - 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 AND RIGID BODY VECTORS

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

$$\{u_a\} = \left(\phi_{aRIG} \middle| \phi_{aFLEX}\right) \begin{Bmatrix} \xi_{RIG} \\ \xi_{FLEX} \end{Bmatrix}$$

- Note: MSC Nastran does not check that discarded modes are rigid body modes (i.e. ω = 0).
- When this transformation is applied to the dynamic system and the modes are unit mass normalized, we obtain:

$$\begin{bmatrix} I_{RIG} & 0 \\ 0 & I_{FLEX} \end{bmatrix} \begin{Bmatrix} \ddot{\xi}_{RIG} \\ \ddot{\xi}_{FLEX} \end{Bmatrix} + \left[\phi^T B \phi\right] \begin{Bmatrix} \dot{\xi}_{RIG} \\ \dot{\xi}_{FLEX} \end{Bmatrix} + \begin{bmatrix} 0 & 0 \\ \cancel{0} & \omega^2_{FLEX} \end{bmatrix} \begin{Bmatrix} \xi_{RIG} \\ \xi_{FLEX} \end{Bmatrix} = \begin{Bmatrix} \phi^T_{RIG} P \\ \phi^T_{FLEX} P \end{Bmatrix} + \begin{bmatrix} \phi^T_{RIG} \\ \phi^T_{FLEX} \end{bmatrix} \{N + Q\}$$

RIGID BODY MODES AND RIGID BODY VECTORS

- **As a result of the transformation, the following consequences occur:**
  - Constraint forces are not externally active, that is:

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

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

$$\left[\phi_{RIG}^{T}\right][B] = [0]$$

  - Thus,

$$\begin{bmatrix} \phi_{RIG}^{T} \\ \phi_{FLEX}^{T} \end{bmatrix} [B] \left[\phi_{RIG} \quad \phi_{FLEX}\right] = \begin{bmatrix} 0 & 0 \\ 0 & \phi_{FLEX}^{T}[B]\phi_{FLEX} \end{bmatrix}$$

RIGID BODY MODES AND RIGID BODY VECTORS

– If damping is “proportional,” then:

$$\begin{bmatrix} \phi_{RIG}^{T} \\ \phi_{FLEX}^{T} \end{bmatrix} [B] \begin{bmatrix} \phi_{RIG} & \phi_{FLEX} \end{bmatrix} = \begin{bmatrix} 0 & 0 \\ 0 & \cancel{2\zeta_i \omega_i} \end{bmatrix}$$

– The modal dynamic equations are fully uncoupled

SECTION 7

DAMPING

DAMPING

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

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

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

  - Structural damping force

$$f_s = igku$$

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

    - Where: $i = \sqrt{-1}$

      g = structural damping coefficient

STRUCTURAL DAMPING VERSUS VISCOUS DAMPING

- **Assume sinusoidal response:**

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

  – Then:

  - <u>Viscous damping:</u>

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

$$m\left(-\omega^2 \bar{u} e^{i\omega t}\right) + b\left(i\omega \bar{u} e^{i\omega t}\right) + k\bar{u} e^{i\omega t} = p(t)$$

$$-\omega^2 m \bar{u} e^{i\omega t} + ib\omega \bar{u} e^{i\omega t} + k\bar{u} e^{i\omega t} = p(t)$$

  - *<u>Structural damping:</u>*

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

$$m\left(-\omega^2 \bar{u} e^{i\omega t}\right) + (1 + ig)k\bar{u} e^{i\omega t} = p(t)$$

$$-\omega^2 m \bar{u} e^{i\omega t} + igk\bar{u} e^{i\omega t} + k\bar{u} e^{i\omega t}\, p(t)$$

STRUCTURAL DAMPING VERSUS VISCOUS DAMPING

- **Both equations are identical if:**

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

  - Therefore, if structural damping g is to be modeled using viscous damping b, then the equality holds at only one frequency ω3 (or ω4).

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

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

  - but $b = \frac{gk}{\omega_n} = g\omega_n m$

$$b_c = 2m\omega_n$$

STRUCTURAL DAMPING VERSUS VISCOUS DAMPING

– Then $$\frac{b}{b_c} = \zeta = \frac{g}{2}$$

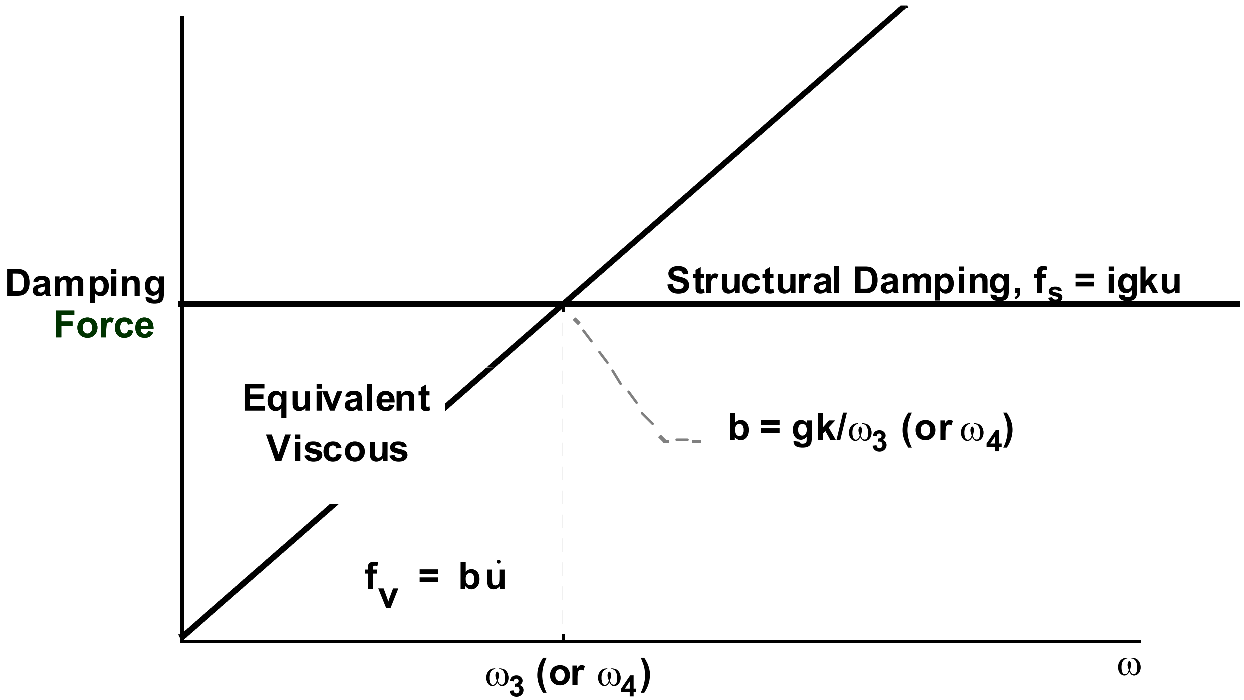
– Where:

ζ = critical damping ratio (percent critical damping)

g = $\frac{1}{Q}$ = structural damping factor

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,** $b/b_{cr} \equiv \zeta$
- **Quality factor Q inversely proportional to energy dissipated per cycle of vibration**
- **At resonance** $\omega \cong \omega_n$
  - ζ = g/2
  - Q = 1/(2ζ)
  - Q = 1/g

STRUCTURAL DAMPING

- **Structural damping**
  - MATi entries

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

- **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 are radians/unit time**
- **If PARAM,G is used, PARAM,W3 must be given a setting greater than zero; otherwise, PARAM,G is ignored in transient response analysis (see Section 8 for more information)**

VISCOUS DAMPING

- **Scalar viscous damping**

CDAMP1	Scalar damper between two DOFs with reference to a property entry.
CDAMP2	Scalar damper between two DOFs without reference to a property entry (PDAMP).
CDAMP3	Scalar damper between two scalar points (SPOINT) with reference to a property entry (PDAMP).
CDAMP4	Scalar damper between two scalar points (SPOINT) without reference to a property entry.
CVISC	Element damper between two grid points; references a property entry (PVISC).
CBUSH	Generalized spring and damper element that may also be frequency dependent.

VISCOUS DAMPING (CDAMP1)

CDAMP1 Scalar Damper Connection

Defines a scalar damper element.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP1	EID	PID	G1	C1	G2	C2			

Example:

CDAMP1	19	6	0		23	2			

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PDAMP property entry. (Integer > 0; Default = EID)
G1, G2	Geometric grid point identification number. (Integer ≥ 0)
C1, C2	Component number. (0 ≤ Integer ≤ 6; 0 or up to six unique integers, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)

VISCOUS DAMPING (CDAMP1)

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. For a discussion of the scalar elements, see Scalar Elements (CELASi, CMASSi, CDAMPi) (p. 172) in the *MSC Nastran Reference Manual*.
5. When CDAMP1 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.

VISCOUS DAMPING (CDAMP2)

CDAMP2 Scalar Damper Property and Connection

Defines a scalar damper element without reference to a material or property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP2	EID	B	G1	C1	G2	C2			

Example:

CDAMP2	16	2.98	32	1					

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
B	Value of the scalar damper. (Real)
G1, G2	Geometric grid point identification number. (Integer $\geq$ 0)
C1, C2	Component number. (0 $\leq$ Integer $\leq$ 6; 0 or up to six unique integers, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)

VISCOUS DAMPING (CDAMP2)

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. For a discussion of the scalar elements, see Scalar Elements (CELASi, CMASSi, CDAMPi) (p. 172) in the *MSC Nastran Reference Manual*.
5. When CDAMP2 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
8. RC network solver does not support CDAMP2 for thermal analysis.

VISCOUS DAMPING (CDAMP3)

CDAMP3 Scalar Damper Connection to Scalar Points Only

Defines a scalar damper element that is connected only to scalar points.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP3	EID	PID	S1	S2					

Example:

CDAMP3	16	978	24	36					

Field	Contents
EID	Unique element identification number. ($0 < \text{Integer} < 100{,}000{,}000$)
PID	Property identification number of a PDAMP entry. ($\text{Integer} > 0$; Default = EID)
S1, S2	Scalar point identification numbers. ($\text{Integer} \geq 0$; $S1 \neq S2$)

VISCOUS DAMPING (CDAMP3)

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar damper element may be defined on a single entry.
4. For a discussion of the scalar elements, see Scalar Elements (CELASi, CMASSi, CDAMPi) (p. 172) in the *MSC Nastran Reference Manual*.
5. When CDAMP3 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. RC network solver does not support CDAMP3 for thermal analysis.

VISCOUS DAMPING (CDAMP4)

CDAMP4 Scalar Damper Property and Connection to Scalar Points Only

Defines a scalar damper element that connected only to scalar points and without reference to a material or property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP4	EID	B	S1	S2					

Example:

CDAMP4	16	-2.6	4	9					

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
B	Scalar damper value. (Real)
S1, S2	Scalar point identification numbers. (Integer $\geq$ 0; $S1 \neq S2$)

VISCOUS DAMPING (CDAMP4)

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar damper element may be defined on a single entry.
4. For a discussion of the scalar elements, see Scalar Elements (CELASi, CMASSi, CDAMPi) (p. 172) in the *MSC Nastran Reference Manual*.
5. If this entry is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. RC network solver does not support CDAMP4 for thermal analysis.

VISCOUS DAMPING (PDAMP)

PDAMP Scalar Damper Property

Specifies the damping value of a scalar damper element using defined CDAMP1 or CDAMP3 entries.

Format:

1	2	3	4	5	6	7	8	9	10
PDAMP	PID1	B1	PID2	B2	PID3	B3	PID4	B4	

Example:

PDAMP	14	2.3	2	6.1					

Field	Contents
PIDi	Property identification number. (Integer > 0)
Bi	Force per unit velocity. (Real)

VISCOUS DAMPING (PDAMP)

Remarks:

1. Damping values are defined directly on the CDAMP2 and CDAMP4 entries, and therefore do not require a PDAMP entry.
2. A structural viscous damper, CVISC, may also be used for geometric grid points.
3. Up to four damping properties may be defined on a single entry.
4. For a discussion of scalar elements, see Scalar Elements (CELASi, CMASSi, CDAMPi) (p. 172) in the *MSC Nastran Reference Manual*.
5. PDAMP is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PDAMP property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PDAMP entries.

VISCOUS DAMPING (CVISC)

CVISC Viscous Damper Connection

Defines a viscous damper element.

Format:

1	2	3	4	5	6	7	8	9	10
CVISC	EID	PID	G1	G2					

Example:

CVISC	21	6327	29	31					

Field	Contents
EID	Element identification number. $(0 < \text{Integer} < 100{,}000{,}000)$
PID	Property identification number of a PVISC entry. (Integer > 0; Default = EID)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; $G1 \neq G2$)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Only one viscous damper element may be defined on a single entry.
3. Grids G1 and G2 must not be coincident. If coincident grids are required, use either the CDAMP or CBUSH entry.

VISCOUS DAMPING (PVISC)

PVISC Viscous Damping Element Property

Defines properties of a one-dimensional viscous damping element (CVISC entry).

Format:

1	2	3	4	5	6	7	8	9	10
PVISC	PID1	CE1	CR1		PID2	CE2	CR2		

Example:

PVISC	3	6.2	3.94						

Field	Contents
PIDi	Property identification number. (Integer > 0)
CE1, CE2	Viscous damping values for extension in units of force per unit velocity. (Real)
CR1, CR2	Viscous damping values for rotation in units of moment per unit velocity. (Real)

VISCOUS DAMPING (PVISC)

Remarks:

1. Viscous properties are material independent; in particular, they are temperature independent.
2. One or two viscous element properties may be defined on a single entry.
3. PVISC is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PVISC property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PVISC entries.

MODAL DAMPING

- **Case Control**
  - SDAMP = n $ selects the modal damping table to be used.
- **Bulk Data**
  - TABDMP1,n,CRIT $ Lists damping values (in "G", "CRIT", or "Q") versus
  - ,x1,y1,x2,y2,..endt $ frequencies.

RAYLEIGH DAMPING

- **Proportional to either the mass or stiffness matrix**
- **Also known as proportional damping**
- **Proportional to mass matrix (param,alpha1,x)**
- **Proportional to stiffness matrix (param,alpha2,y)**
- **Available in transient and frequency response analysis**
- **Scale factors applied to d-set (direct) and h-set (modal)**
- **Added to the viscous damping matrix as follows:**

$$\left[\mathbf{B'}\right] = \left[\mathbf{B}\right] + alpha1 * \left[\mathbf{M}\right] + alpha2 * \left[\mathbf{K}\right]$$

RAYLIEGH DAMPING

- **ALPHA1 and ALPHA2 are Complex parameters**

Real and imaginary

PARAM, ALPHA2, 1.25E-4, 0.

- **Modal Damping Matrix reads**

$$\boldsymbol{\varphi}^{T}\mathbf{B}\boldsymbol{\varphi} = \alpha_1 \boldsymbol{\varphi}^{T}\mathbf{M}\boldsymbol{\varphi} + \alpha_2 \boldsymbol{\varphi}^{T}\mathbf{K}\boldsymbol{\varphi}$$

$$= \alpha_1 \mathbf{I} + \alpha_2 \boldsymbol{\Omega}^2$$

SECTION 8

TRANSIENT RESPONSE 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
  - 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 a 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 MSC Nastran to compute velocity and acceleration output**

DIRECT TRANSIENT RESPONSE

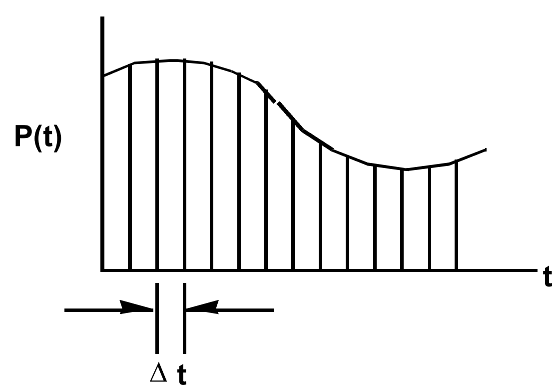
- **Numerical integration (Central Difference type method) (except "smear" force over 3 adjacent time points)**

$$\left[\frac{m}{\Delta t_2}\right]\left(u_{n+1}-2u_n+u_{n-1}\right)+\left[\frac{b}{2\Delta t}\right]\left(u_{n+1}-u_{n-1}\right)+\left[\frac{k}{3}\right]\left(u_{n+1}+u_n+u_{n-1}\right)=\frac{1}{3}\left(P_{n+1}+P_n+P_{n-1}\right)$$

$$\frac{u_{n+1}+u_n+u_{n-1}}{3}$$

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

Time Average "Filters"
In MSC Nastran



DIRECT TRANSIENT RESPONSE

- **Solution**

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

  – Where:

$$[A_1] = \left[M/\Delta t^2 + B/2\Delta t + K/3\right]$$ Dynamic Matrix

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

$$[A_3] = \left[2M/\Delta t^2 - K/3\right]$$

$$[A_4] = \left[-M/\Delta t^2 + B/2\Delta t - K/3\right]$$

Initial conditions from previous

  – Solve by decomposing A1 and applying it to the right-hand side of the above equation.
  – Similar to classical Newmark-Beta method

DIRECT TRANSIENT RESPONSE

- **M, B, and K do not change with time**
- **A_1 needs to be decomposed only once if Dt is unchanged throughout the entire solution. If Dt is changed, A_1 must be redecomposed (which may be a costly operation).**
- **The output time interval may be greater than the solution time interval**
  - In this case, use solution Dt of 0.001 second and output results every fifth time step or with output Dt of 0.005 second.

DAMPING IN DIRECT TRANSIENT RESPONSE

- **Damping matrix B is comprised of several matrices:**

$$B = B^1 + B^2 + \frac{G}{W_3} K^1 + \frac{1}{W_4} \sum G_E K_E$$

  – Where,

    B^1 = damping elements (VISC,DAMP) + B2GG

    B^2 = B2PP direct input matrix + transfer functions

    G = overall structural damping coefficient (PARAM,G)

    W_3 = frequency of interest - rad/sec (PARAM,W3)

    K^1 = global stiffness matrix

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

    W_4 = frequency of interest - rad/sec (PARAM,W4)

    K_E = element stiffness matrix

- **Transient analysis does not permit complex coefficients. Therefore, structural damping is included by means of equivalent viscous damping.**

- **The default values for W3, W4 are 0.0. In this case, they cause associated damping terms to be ignored.**

MODAL TRANSIENT RESPONSE

- **Transform from physical to modal coordinates**

  (1) $$\{u\} = [\phi]\{\xi\}$$

- **Temporarily remove damping. The equation of motion becomes:**

  (2) $$[M]\{\ddot{u}\} + [K]\{u\} = \{P(t)\}$$

- **Substitute Equation 1 into Equation 2 to obtain:**

  (3) $$[M][\phi]\{\ddot{\xi}\} + [K][\phi]\{\xi\} = \{P(t)\}$$

- **Pre-multiply by [ϕ^T] to obtain:**

  (4) $$[\phi^T][M][\phi][\ddot{\xi}] + [\phi^T][K][\phi]\{\xi\} = [\phi^T]\{P(t)\}$$

  – Where $\phi^T M \phi$ = modal mass matrix (diagonal)
  $\phi^T K \phi$ = modal stiffness matrix (diagonal)
  $\phi^T P$ = modal force vector

MODAL TRANSIENT RESPONSE

- **Equation 4 can be written as decoupled SDOF systems:**

(5) $$m_i \ddot{\xi}^i + k_i \xi^i = p_i(t)$$

– Where m_i = i^{th} modal mass
k_i = i^{th} modal stiffness
p_i = i^{th} modal force

DAMPING IN MODAL TRANSIENT RESPONSE

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

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

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

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

  – Where:

$$[A_1] = [\phi^T]\left[\frac{M}{\Delta t^2} + \frac{B}{2\Delta t} + \frac{K}{3}\right][\phi]$$ Dynamic Matrix

$$[A_2] = \frac{1}{3}[\phi^T]\{P_{n+1} + P_n + P_{n-1}\}$$ Applied Forces

$$[A_3] = [\phi^T]\left[\frac{2M}{\Delta t^2} - \frac{K}{3}\right][\phi]$$

$$[A_4] = [\phi^T]\left[-\frac{M}{\Delta t^2} + \frac{B}{2\Delta t} - \frac{K}{3}\right][\phi]$$

} Initial conditions from previous ($[A_3]$ and $[A_4]$)

DAMPING IN MODAL TRANSIENT RESPONSE

- **If modal damping is used, then each mode has damping *b_i***
- **The equations of motion become uncoupled**

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

or

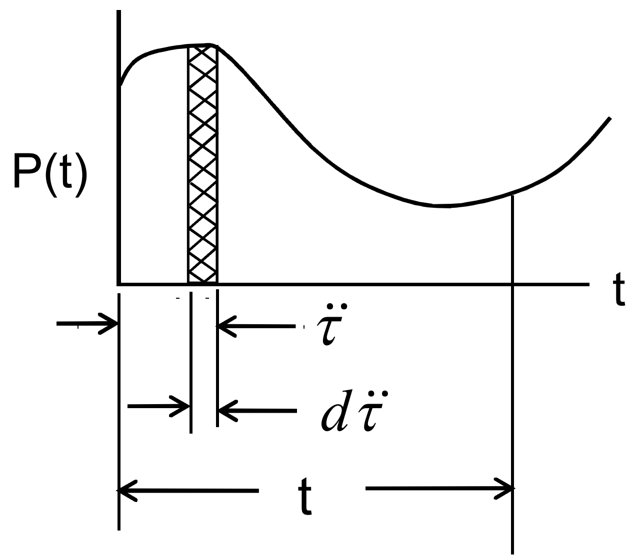
$$\ddot{\xi}_i + 2\zeta_i\omega_i\dot{\xi} + \omega_i^2\xi_i = p_i(t)/m_i$$

– Where:
  - $\zeta_i = b_i/(2m_i\omega_i)$ is the modal damping ration
  - $\omega_i^2 = k_i/m_i$ is the eigenvalue

DAMPING IN MODAL TRANSIENT RESPONSE

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

$$\xi(t) = e^{-bt/2m}\left(\xi_o \cos\omega_d t + \frac{\dot{\xi}_o + (b/2m)\xi_o}{\omega_d}\right) + e^{-bt/2m}\frac{1}{m\omega_d}\int_0^t e^{bt/2m}\, p(\tau)\sin\omega_d(t-\tau)\,d\tau$$



DAMPING IN MODAL TRANSIENT RESPONSE

- It is most efficient to use modal damping ratios since equations are decoupled
- TABDMP1 Bulk Data entry defines the modal damping ratios

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

- Type = G (default), CRIT, or Q

$$\zeta = b/b_{cr} = G/2$$
$$Q = 1/(2\zeta)$$
$$Q = 1/G$$

- Example: for 10% critical damping

$$CRIT = 0.10$$
$$Q = 5.0$$
$$G = 0.2$$

DAMPING IN MODAL TRANSIENT RESPONSE (CONT.)

- **The TABDMP1 Bulk Data entry is selected with the SDAMPING Case Control command**
- **f_i (units: Hz) and g_i define pairs of frequencies and dampings. Straight-line interpolation is used for modal frequencies between consecutive f_is. Linear extrapolation is used at the ends of the table. ENDT ends the table input.**
- **Example: Assume modes at 1.0, 2.5, 3.6, and 5.5 Hz**

Entered		Computed	
f	g	f	g
2.0	0.10	1.0	0.02
3.0	0.18	2.5	0.14
4.0	0.13	3.6	0.15
6.0	0.13	5.5	0.13

Calculated by Extrapolation (row 1.0 / 0.02)

Calculated by Interpolation (rows 2.5 to 5.5)

- **May add nonmodal damping (PARAM, G; VISC; DAMP; GE on MATi entry)**
  - Computational cost due to coupled B causing direct integration to be used
- **Recommended practice: Use only modal damping (TABDMP1) in modal transient response analysis. If discrete damping is desired, use direct transient response analysis.**

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 Dt in modal transient response as in direct. However, the constant is still recommended.**
- **The output time interval may be greater than the solution time interval**

MODE TRUNCATION

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

SELECTIVE MODES DELETION

- **By default all modes calculated are included in the response analysis**
- **Parameters available for excluding modes at low or high end:**
  - Param,lfreq,value – modes below frequency “value” are not included
    - Default value = 0.0
  - Param,hfreq,value – modes above frequency “value” are not included
    - Default value = 1.E+30
  - Param,lmodes,number – only the lowest “number” modes are included
    - Default number = 0 → the retained modes are determined by parameter LFREQ and HFREG
- **Similar set of parameters are available for fluid modes**

DELETING HIGH/LOW MODES

- **Command to delete modes below and/or above certain frequencies**
- **FLSFEL is a Case Control command**

$$\mathbf{FLSFSEL}\left(\mathbf{LFREQ}=\left\{\frac{\mathbf{0.0}}{\mathbf{f}_{\mathbf{s}_1}}\right\}\right),\left(\mathbf{HFREQ}=\left\{\frac{\mathbf{1.}E+\mathbf{30}}{\mathbf{f}_{\mathbf{s}_2}}\right\}\right),\left(\mathbf{LMODES}=\left\{\frac{\mathbf{0.0}}{\mathbf{ms}}\right\}\right)$$

- Where:
  - $\mathbf{f}_{\mathbf{s}_1}$ = lower frequency range for structure (real number)
  - $\mathbf{f}_{\mathbf{s}_2}$ = upper frequency range for structure (real number
  - ms = number of lowest modes to use for structure portion of model
- **Example:**
  - FLSFSEL HFREQ = 4.

SELECTIVE MODES DELETION

- **Previous commands remove high/low modes**
- **MODESELECT allows you to remove selective mode**
  - MODSELECT is a Case Control command

$$\textbf{MODSELECT}\begin{bmatrix}\textbf{STRUCTURE}\\ \textbf{FLUID}\end{bmatrix}=\begin{Bmatrix}\textbf{ALL}\\ \textbf{n}\end{Bmatrix}$$

- **Use to either:**
  - Include a selective set of modes (n > 0)

    or
  - Exclude a selective set of modes (n < 0)
- **Note that the modes that are deleted will not participate in the response**
  - This may lead to incorrect results if the wrong modes are deleted

SELECTIVE MODES DELETION

- **Example:**
  - **Select all 10 modes excluding modes 6 and 7.**

    SET 100 = 1 THRU 10 EXCEPT 6,7

    MODESELECT = 100

    or

    SET 200 = 6,7

    MODESELECT = -200

TRANSIENT EXCITATION

- **Define force as a function of time**
- **Several methods in MSC Nastran:**
  - TLOAD1 – "Brute force"; ordered time, force pairs table input
  - TLOAD2 – Efficient definition for analytical-type loadings
  - LSEQ – Generates dynamic loads from static loads (not recommended)

TLOAD1 ENTRY

- **Defines excitation in the form:** $P(t) = AF(t - \tau)$

1	2	3	4	5	6	7	8	9	10
TLOAD1	SID	EXCITEID	DELAYI/ DELAYR	TYPE	TID/F	US0	VS0		

  – Where:
    - A = spatial load distribution and scale factor (DAREA, static load, thermal load, or LSEQ)
    - t = DELAY entry (Integer) or time delay
    - τ (Real), Default = 0.
    - $F(t - \tau)$ = TABLEDi entry
- **DELAY defines DOFs and time delay**
- **TID – specifies TABLEDi for defining time and force pairs**
- **Selected by DLOAD Case Control command**

  Delay SID P1 C1 T1 P2 C2 T2

  – Pi – grid number

  – Ci – component number

  – Ti – Time delay for designated point pi and component ci (real)

TLOAD1 ENTRY

- **Excitation is defined by TYPE**

TYPE	TYPE of Dynamic Excitation
0, L, LO, LOA or LOAD	Applied load (force or moment) (Default)
1, D, DI, DIS, or DISP	Enforced displacement using large mass or SPC/SPCD data
2, V, VE, VEL or VELO	Enforced velocity using large mass or SPC/SPCD data (not supported for multiple steps in SOL400)
3, A, AC, ACC or ACCE	Enforced acceleration using large mass or SPC/SPCD data
4	FLOW boundary condition on the face of an Eulerian solid element (SOL 700 only).
5	Displacement of SPH elements before activation by a FLOWSPH boundary condition (SOL 700 only).
6	Velocity of SPH elements before activation by a FLOWSPH boundary condition (SOL 700 only).
7	Acceleration of SPH elements before activation by a FLOWSPH boundary condition (SOL 700 only)
12	Velocity of the center of gravity of a rigid body (SOL 700 only)
13	Force or moment on the center of gravity of a rigid body (SOL 700 only).

- **Only loads (first row) will be discussed in this section. For enforced motion, see section 12.**

TLOAD2 ENTRY

- **Defines excitation in the form of**

$$\{P(t)\} = \begin{cases} 0 & , \quad t < (T1 + \tau) \text{ or } t > (T2 + \tau) \\ \{A\}\tilde{t}^{B} e^{C\tilde{t}} \cos(2\pi F\tilde{t} + P) & , \quad (T1 + \tau) \le t \le (T2 + \tau) \end{cases}$$

1	2	3	4	5	6	7	8	9	10
TLOAD2	SID	EXCITEID	DELAYI/ DELAYR	TYPE	T1	T2	F	P	
	C	B	US0	VS0					

  – where

$$\tilde{t} = t - T1 - \tau$$

A	Defined as a spatial load distribution and scale factor (DAREA, static load, thermal load, or LSEQ)
t	Defined on a DELAY entry
TYPE	Defined as in the TLOAD1
T1,T2	Time constants (T2>T1)
F	Frequency (Hz)
P	Phase angle (degrees)
C	Exponential coefficient
B	Growth coefficient

- **Selected by the DLOAD Case Control command**

LOAD SET COMBINATION - DLOAD

- **The applied load P_C is constructed from a combination of component load sets P_K**

$$p_c = S_c \sum_K S_K P_K$$

  – Where:

    S_C = overall scale factor

    S_K = scale factor for k-th load set

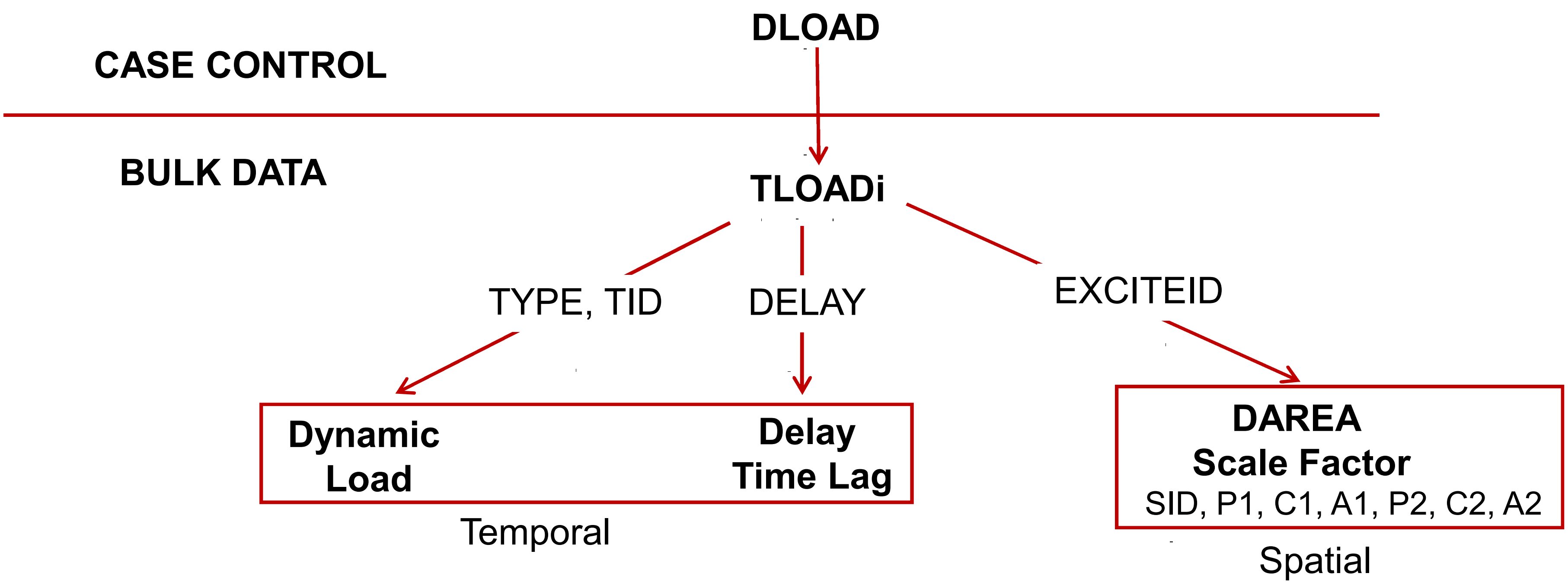
    P_K = SID of TLOAD

1	2	3	4	5	6	7	8	9	10
DLOAD	SID	S_C	S_l	P_1	S_2	P_2	-etc-		

- **TLOAD1s and TLOAD2s must have unique SIDs**
- **Use the DLOAD entry to combine TLOADs**
- **The DLOAD Bulk Data entry is selected by DLOAD Case Control command**

DAREA ENTRY

- **Defines the degree of freedom where the dynamic load is to be applied to the scale factor**
- **Can point to static load directly (for example, FORCE, PLOAD4, etc.) instead of DAREA**
- **Relationship to other input:**



EXAMPLE USING TLOAD1

Type = 0 → Structural load

TLOAD1	SID	EXCITEID	DELAY	TYPE	TID				
TLOAD1	35	29	0.2	0	40				

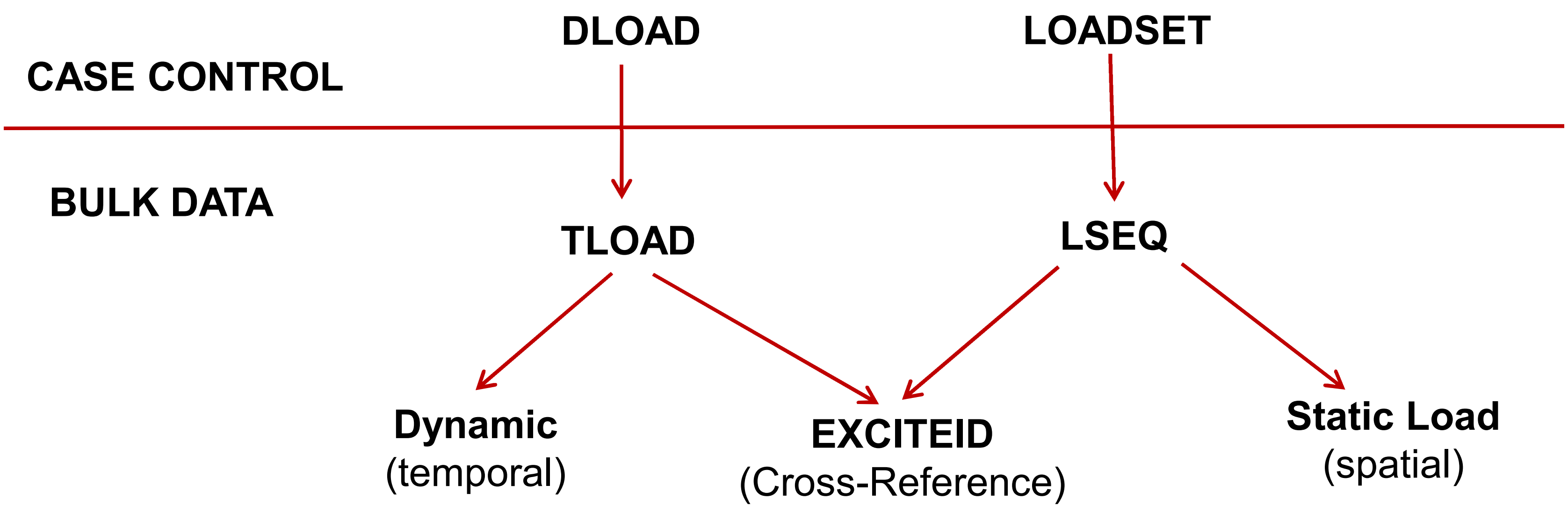
FORCE	SID	G	CID	F	N1	N2	N3		
FORCE	29	30		5.2	1.				

TABLED1	ID	XAXIS	YAXIS						
	X1	Y1	X2	Y2	X3	Y3	X4	Y4	
TABLED1	40								
	0.	0.	0.1	1.5	0.2	2.0	3.0	1.5	
	4.0	1.5	ENDT						

- **Result is the load specified by the TLOAD1, scaled by 5.2, delayed by 0.2 seconds, and applied to grid point 30, component T1**

STATIC LOAD – INDIRECT METHOD

- **Defines static loads that are being applied dynamically**
- **The LSEQ Bulk Data entry is selected by the LOADSET Case Control command**
- **Contains a EXCITEID entry to identify the loadset for use with the TLOAD entries**
- **Relationship to other input**



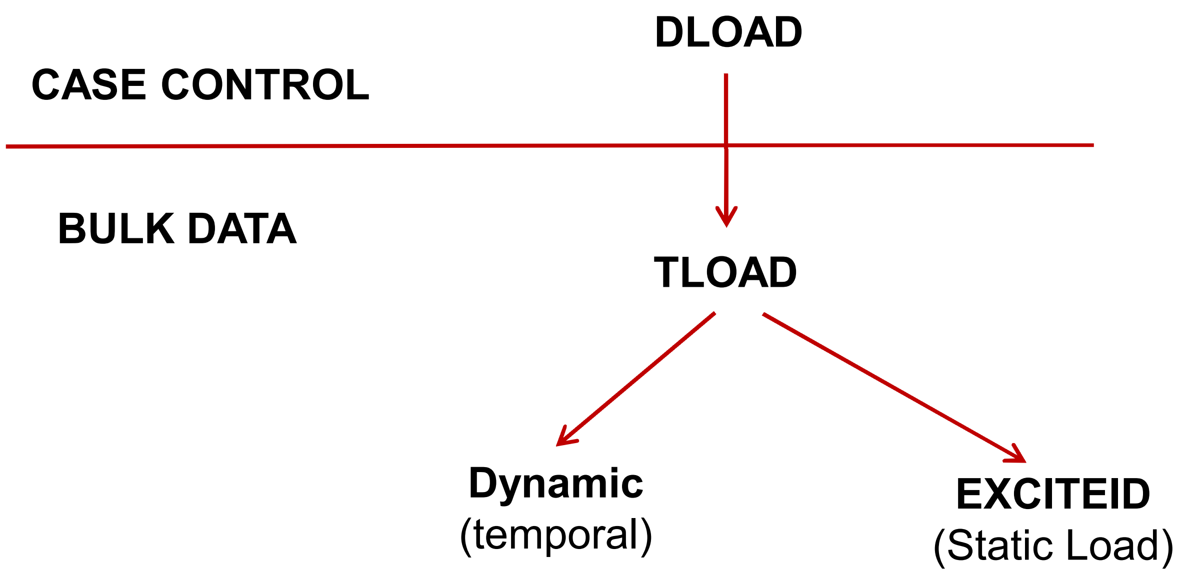
STATIC LOAD – INDIRECT METHOD

- LOADSET must appear above all subcases.
- Only one LOADSET may be specified per superelement

```
DLOAD      = 25
LOADSET    = 27
---------------------------------------------
        TLOAD1  25        28               5

        LSEQ    27        28       100

        PLOAD4  100  ....

        TABLED1    5    ....
```

STATIC LOAD – DIRECT METHOD

- **Defines static loads that are being applied dynamically.**
- **The EXCITEID references the static load (for example, PLOAD4) directly**



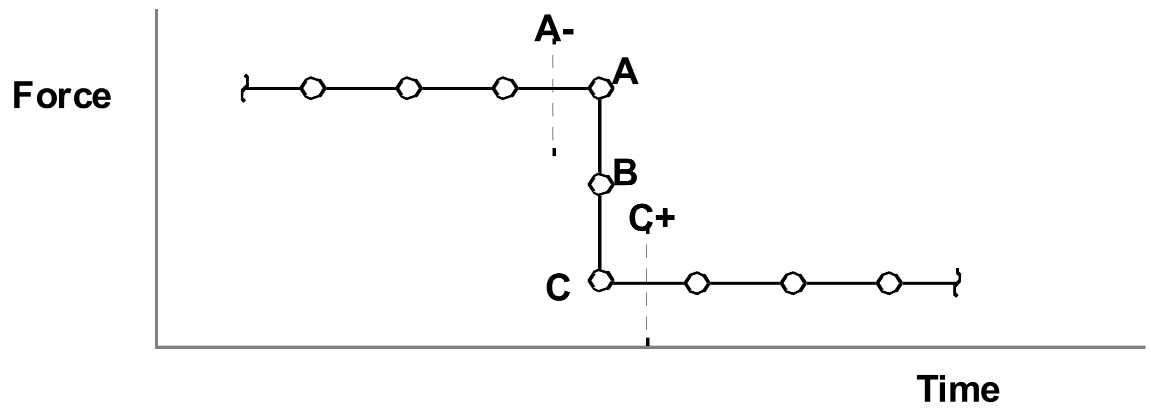
STATIC LOAD – DIRECT METHOD

DLOAD = 25

TLOAD1	**25**	**100**	**5**
PLOAD4	**100**	**…..**	
TABLED1	**5**	**….**	

TRANSIENT EXCITATION CONSIDERATIONS

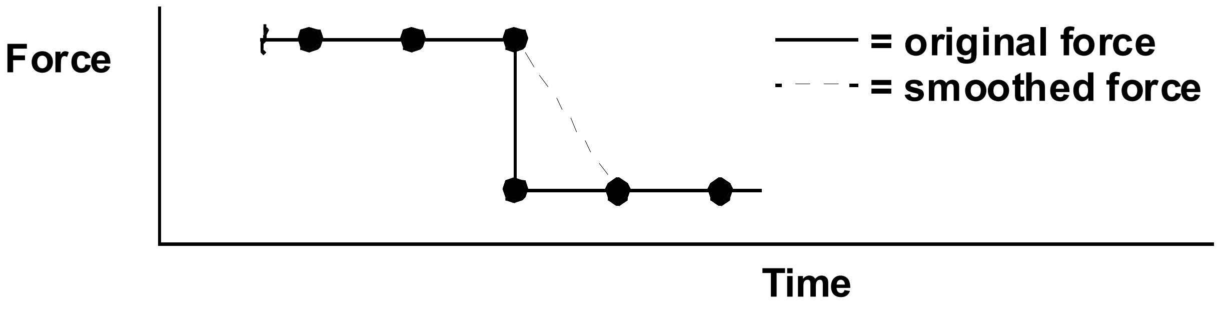
- Remember the 1/3 "smearing" of applied loads. This will smooth the force and decreases apparent frequency content.
- Avoid discontinuous forces. These may cause different results on different computers.



- If NDt causes a solution at ABC, then MSC Nastran should select the average force B
- However, due to numerical roundoff, ND_t on one computer may be at time A- and will give force A. On another computer, ND_t may be at time C+ and will give force C.

TRANSIENT EXCITATION CONSIDERATIONS

- **The integration results will differ depending on whether the force at NDt is A, B, or C.**
- **Smooth a discontinuous force over one Dt**



INITIAL CONDITIONS

- May impose initial displacement and/or velocity in transient response via the TIC Bulk Data entry.
- The IC Case Control command selects the TIC entry.
- Be careful - initial conditions for unspecified DOFs are set to zero.
- Initial conditions may be specified only for A-set DOFs.
- 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 t $\leq$ 0 (constant velocity).

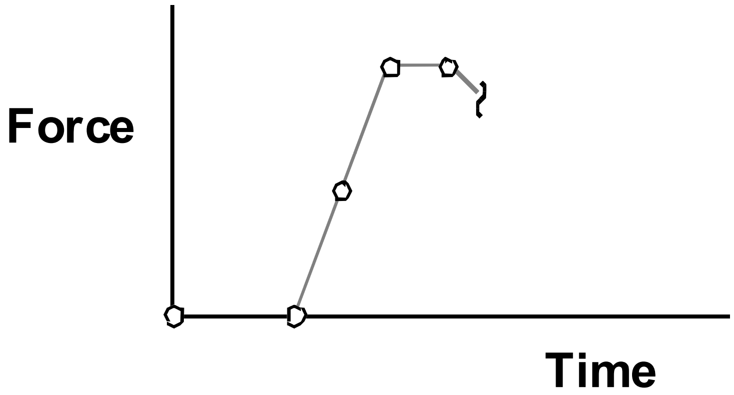
$$\{u_{-1}\} = \{u_0\} - \{\dot{u}_0\}\Delta t$$
$$\{P_{-1}\} = [K]\{u_{-1}\} + [B]\{\dot{u}_0\}$$

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

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

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.**



TRANSIENT INITIAL CONDITION (TIC)

TIC **Transient Analysis Initial Condition**

Defines values for the initial conditions of variables used in structural transient analysis. Both displacement and velocity values may be specified at independent degrees-of-freedom. This entry may not be used for heat transfer analysis.

Format:

IC=SID in Case Control

1	2	3	4	5	6	7	8	9	10
TIC	SID	G	C	U0	V0				

Example:

TIC	100	10	3	0.1	0.5				

Field	Contents
SID	Set identification number. (Integer > 0)
G	Grid, scalar, or extra point or modal coordinate identification number. (Integer > 0). See Remark 4.
C	Component numbers. (Any one of the integers 1 through 6 for grid points, integer zero or blank for scalar or extra points and -1 for modal coordinates.)
U0	Initial displacement. (Real)
V0	Initial velocity. (Real)

TRANSIENT INITIAL CONDITION (TIC)

Remarks:

1. Transient analysis initial condition sets must be selected with the IC Case Control command. Note the use of IC in the Case Control command versus TIC on the Bulk Data entry. For heat transfer, the IC Case Control command selects TEMP or TEMPD entries for initial conditions and not the TIC entry.
2. If no TIC set is selected in the Case Control Section, all initial conditions are assumed to be zero.
3. Initial conditions for coordinates not specified on TIC entries will be assumed to be zero.
4. In direct transient analysis (SOL 109 and 129) as well as in modal transient analysis (SOL 112) wherein the TIC Bulk Data entry is selected by an IC or IC(PHYSICAL) Case Control command, G may reference only grid, scalar or extra points. In modal transient analysis (SOL 112) wherein the TIC Bulk Data entry is selected by an IC(MODAL) Case Control command, G may reference only modal coordinates or extra points.
5. The initial conditions for the independent degrees-of-freedom specified by this Bulk Data entry are distinct from, and may be used in conjunction with, the initial conditions for the enforced degrees-of-freedom specified by TLOAD1 and/or TLOAD2 Bulk Data entries.

TSTEP ENTRY

- **Select integration time step for direct and modal transient response**
  - Integration errors increase with increasing natural frequency
  - Recommended Dt is to use at least eight solution time steps per period (cycle) of response
  - Recommended: $\Delta T \leq 1./(10 * f\,\max)$
- **The TSTEP Bulk Data entry controls solution and output Dt, and is selected by the TSTEP Case Control command**
- **The cost of integration is directly proportional to the number of time steps when Dt is constant**
- **Use an adequate length of time to properly capture long-period (low frequency) response**

TSTEP ENTRY

- **User may change Dt during a run. $\ddot{u}$ is assumed constant for t $\leq$ NDt$_1$. $\ddot{u}$ calculates new initial conditions for the integration based on {u$_n$} and the calculated velocity and acceleration at the transition. The assumption of uniform acceleration assures a smooth transition when Dt is changed.**

$$\{\dot{u}_0\} = \frac{1}{\Delta t_1}\{u_N - u_{N-1}\}$$

$$\{\ddot{u}_0\} = \frac{1}{\Delta t_1^2}\{u_N - 2u_{N-1} + u_{N-2}\}$$

- **The initial conditions for the new integration are:**

$$\{u_0\} = \{u_N\}$$

$$\{u_{-1}\} = \{u_N\} - \Delta t_2\{\dot{u}_0\} - \frac{1}{2}\Delta t_2^2\{\ddot{u}_0\}$$

$$\{P_0\} = \{P_N\}$$

$$\{P_{-1}\} = [K]\{u_{-1}\} + [B]\{\dot{u}_{-1}\} + [M]\{\ddot{u}_{-1}\}$$

$$= [K]\{u_{-1}\} + [B]\{\dot{u}_0 - \Delta t_2\ddot{u}_0\} + [M]\{\ddot{u}_0\}$$

Uniform acceleration

- **Note: New matrices A$_1$ - A$_4$ are assembled, and the new A$_1$ must be decomposed**

TSTEP ENTRY - TRANSIENT TIME STEP

TSTEP **Transient Time Step**

Defines time step intervals at which a solution will be generated and output in transient analysis.

Format: TSTEP = SID in CASE Control

1	2	3	4	5	6	7	8	9	10
TSTEP	SID	N1	DT1	NO1					
		N2	DT2	NO2					
		-etc.-							

Example:

TSTEP	2	10	.001	5					
		9	0.01	1					

Field	Contents
SID	Set identification number. (Integer > 0)
Ni	Number of time steps of value DTi. (Integer ≥ 1)
DTi	Time increment. (Real > 0.0)
NOi	Skip factor for output. Every NOi-th step will be saved for output. (Integer > 0; Default = 1)

TSTEP ENTRY - TRANSIENT TIME STEP

Remarks:

1. TSTEP entries must be selected with the Case Control command TSTEP = SID.
2. Note that the entry permits changes in the size of the time step during the course of the solution. Thus, in the example shown, there are 10 time steps of value .001 followed by 9 time steps of value .01. Also, the user has requested that output be recorded for t = 0.0, .005, .01, .02, .03, etc.
3. See Guidelines and Tools for Effective Dynamic Analysis (p. 543) in the for a discussion of considerations leading to the selection of time steps.
4. In modal frequency response analysis (SOLs 111 and 146), this entry is required only when TLOADi is requested; i.e., when Fourier methods are selected.
5. The maximum and minimum displacement at each time step and the SIL numbers of these variables can be printed by altering DIAGON(30) before the transient module TRD1 and by altering DIAGOFF(30) after the module. This is useful for runs that terminate due to overflow or excessive run times.
6. For heat transfer analysis in SOL 159, use the TSTEPNL entry.

DYNAMIC DATA RECOVERY

- **Three options for recovering displacements and stresses in modal solutions:**
  - mode displacement method
    - The mode displacement method computes physical displacements directly from modal displacements and then computes element stresses from the physical displacements. The number of operations is proportional to number of time steps (T).
    - The mode displacement method can be selected via PARAM, DDRMM, -1.
  - matrix method
    - The matrix method computes displacements per mode and element stresses per mode and then computes physical element stresses as the summation of modal element stresses. Costly operations are proportional to the number of modes (H).

$$\frac{\text{Cost of matrix method}}{\text{Cost of mode displacement method}} = \frac{H}{T} \quad \begin{matrix}\text{Number of Modes} \\ \text{Number of Time Steps}\end{matrix}$$

    - Since H is usually << T, the matrix method is cheaper.
    - The matrix method is the default and is the recommended method for most cases.
  - mode acceleration method
    - The mode acceleration method automatically accounts for the quasi-static response of all high frequency modes (See Appendix C). (Requires mode displacement method)

MODAL TRANSIENT VERSUS DIRECT TRANSIENT

	MODAL	DIRECT
Small Model		X
Large Model	X	
Few Time Steps		X
Many Time Steps	X	
High Frequency Excitation		X
Nonlinearities		X
Initial conditions	X	X
Modal Damping	X	

SOLUTION CONTROL FOR TRANSIENT ANALYSIS

- **Executive Control Section**
- **SOL (for required input see below)**

Method	Structured Solution Sequences
Direct	109
Modal	112

- **Case Control Section**
  - DLOAD (both - required)
  - LOADSET (both - optional)
  - METHOD (modal - required)
  - SDAMPING (modal - optional)
  - IC (Physical) (both - optional)
  - IC (Modal) (modal – optional)
  - TSTEP (both - required)

SOLUTION CONTROL FOR TRANSIENT ANALYSIS

- **Bulk Data Section**

– ASET,OMIT	(both - optional)
– EIGRL or EIGR	(modal - required)
– TSTEP	(both - required)
– TIC	(both - optional)
– TLOADi	(both - required)
– LSEQ	(both - optional)
– DAREA	(both – optional)
– TABLEDi	(both - optional)
– DELAY	(both - optional)
– DLOAD	(both - optional)
– TABDMP1	(modal - optional)

CASE CONTROL OUTPUT

- **Grid output**
  - ACCELERATION
  - DISPLACEMENT (or VECTOR)
  - GPSTRESS
  - NLLOAD (nonlinear load output)
  - OLOAD (output applied load)
  - SACCELERATION
  - SDISPLACEMENT
  - SVELOCITY
  - SVECTOR (A-set eigenvector)
  - SPCFORCES
  - VELOCITY
  - MPCFORCE

(SACCELERATION, SDISPLACEMENT, SVELOCITY: solution set output-A-set in direct solutions, modal variables in modal solutions)

CASE CONTROL OUTPUT

- **Element output**
  - ELSTRESS (or STRESS)
  - ELFORCE (or FORCE)
  - STRAIN
- **Special**
  - OTIME (controls solution output times)

EXERCISES

- **Now Perform the following workshops**
  - Workshop 4, Direct Transient Response
  - Workshop 5, Modal Transient Analysis

SECTION 9

FREQUENCY RESPONSE ANALYSIS

INTRODUCTION TO FREQUENCY RESPONSE ANALYSIS

- **Compute the response to oscillatory excitation**
- **Excitation 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 the forcing) or as real and imaginary components**
- **Two categories of analysis**
  - direct
  - modal

DIRECT FREQUENCY RESPONSE

- **Dynamic equation of motion:**

$$(1) \quad \left[-\omega^2 M + i\omega B + K\right]\{u(\omega)\} = \{P(\omega)\}$$

- **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)

- **Contrast this with transient response analysis**

$$B_{TRANS} = B^1 + B^2 + \frac{G}{W^3}K^1 + \frac{1}{W_4}\sum G_E k_E$$

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

MODAL FREQUENCY RESPONSE

- Convert to modal coordinates and solve as decoupled SDOF systems

$$\xi_i = \frac{P_i}{-m_i\omega^2 + ib_i\omega + k_i}$$

- 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 frequency**
- **Several methods in MSC Nastran:**
  - RLOAD1 – defines frequency-dependent load in real and imaginary forms
  - RLOAD2 – defines frequency-dependent load in magnitude and phase forms
  - LSEQ – generates dynamic loads from static loads
- **DLOAD Bulk Data entries are used to combine frequency-dependent forces**
- **RLOADi entries are selected by DLOAD Case Control commands**

RLOAD1 ENTRY

RLOAD1 Frequency Response Dynamic Excitation, Form 1

Defines a frequency-dependent dynamic load of the form

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

for use in frequency response problems.

Format:

1	2	3	4	5	6	7	8	9	10
RLOAD1	SID	EXCITEID	DELAYI/ DELAYR	DPHASEI/ DPHASER	TC/RC	TD/RD	TYPE		

Example:

RLOAD1	5	3	2.0	10	1				

Field	Contents
SID	Set identification number. See Remarks 1. and 3. (Integer > 0)
EXCITEID	Identification number of a static or thermal load set or a DAREA or FBALOAD (in FRF Based Assembly or FBA process) or SPCD entry set that defines $\{A\}$. See Remarks 4. and 5. (Integer > 0)
DELAYI	Identification number of DELAY or FBADLAY (in FRF Based Assembly or FBA process) Bulk Data entry that defines time delay τ. See Remark 2. (Integer > 0 or blank)

RLOAD1 ENTRY

Field	Contents
DELAYR	Value of time delay τ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 2. (Real or blank)
DPHASEI	Identification number DPHASE or FBAPHAS (in FRF Based Asseembly or FBA process) Bulk Data entry that defines phase angle θ. (See Remark 2. (Integer > 0 or blank)
DPHASER	Value of phase angle θ (in degrees) that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 2. (Real or blank)
TC	Set identification number of the TABLEDi entry that gives $C(f)$. See Remark 2. (Integer > 0 or blank)
RC	Value of C to be used for all frequencies. See Remark 2.. (Real or blank)
TD	Set identification number of the TABLEDi entry that gives $D(f)$. See Remark 2. (Integer > 0 or blank)
RD	Value of D to be used for all frequencies. See Remark 2.. (Real or blank)
TYPE	Defines the type of the dynamic excitation. See Remarks 4. and 5. (Integer, character or blank; Default = 0)

Note: For rest of RLOAD1 entry, see MSC Nastran 2013 QRG

RLOAD2 ENTRY

RLOAD2 Frequency Response Dynamic Excitation, Form 2

Defines a frequency-dependent dynamic excitation of the form.

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

for use in frequency response problems.

Format:

1	2	3	4	5	6	7	8	9	10
RLOAD2	SID	EXCITEID	DELAYI/ DELAYR	DPHASEI/ DPHASER	TB/RB	TP/RP	TYPE		

Example:

RLOAD2	5	3	15	5.0	7				

Field	Contents
SID	Set identification number. See Remarks 1. and 3. (Integer > 0)
EXCITEID	Identification number of a static or thermal load set or a DAREA or FBALOAD (in FRF Based Assembly or FBA process) or SPCD entry set that defines $\{A\}$. See Remarks 4. and 5. (Integer > 0)
DELAYI	Identification number of DELAY or FBADLAY (in FRF Based Assembly or FBA process) Bulk Data entry that defines time delay τ. See Remark 2. (Integer > 0 or blank)

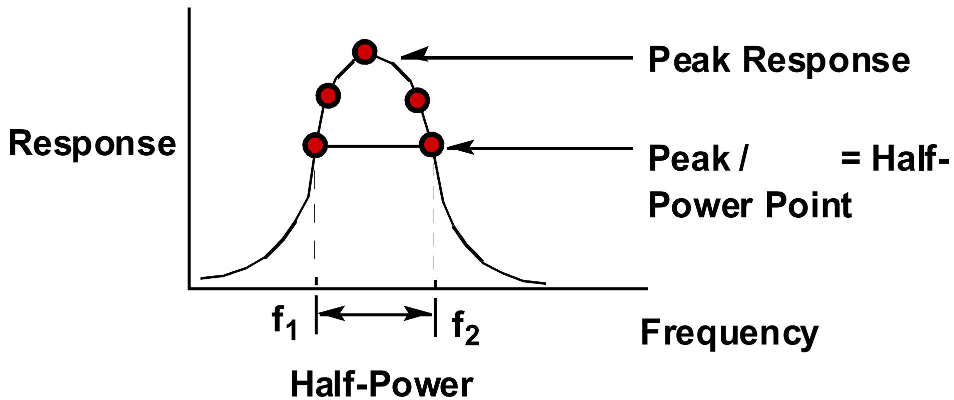
RLOAD2 ENTRY

Field	Contents
DELAYR	Value of time delay τ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 2. (Real or blank)
DPHASEI	Identification number DPHASE or FBAPHAS (in FRF Based Assembly or FBA process) Bulk Data entry that defines phase angle θ. (See Remark 2. (Integer > 0 or blank)
DPHASER	Value of phase angle θ (in degrees) that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 2. (Real or blank)
TB	Set identification number of the TABLEDi entry that gives $B(f)$. (Integer > 0)
RB	Value of B to be used for all frequencies. (Real, non-zero)
TP	Set identification number of the TABLEDi entry that gives $\phi(f)$ in degrees. (Integer $\geq$ 0)
RP	Value of ϕ to be used for all frequencies. See Remark 2. (Real or blank)
TYPE	Defines the type of the dynamic excitation. See Remarks 4. and 5. (Integer, character or blank; Defaults = 0)

Note: For rest of RLOAD2 entry, see MSC Nastran 2013 QRG

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 around resonant frequencies and larger Df in regions far away from resonant frequencies.

FREQI ENTRIES

- **Applicable for Direct and Modal Method**
  - Select frequency step size
  - The FREQ entry defines discrete excitation frequencies
  - The FREQ1 entry defines f_{START}, frequency increment, and the number of increments
  - The FREQ2 entry defines f_{START}, f_{END}, and the number of logarithmic intervals
- **Applicable for Modal Method**
  - The FREQ3 entry defines F1, F2, and the number of frequencies in between using either a linear or log interpolation. Biased towards end points or center.
  - The FREQ4 entry specifies a frequency at each resonant frequency and the number of equally spaced excitation frequencies within the spread
  - The FREQ5 entry specifies a frequency range and fractions of the natural frequencies within that range
  - The FREQ3, FREQ4, and FREQ5 entries are available only for the modal method

FREQI ENTRIES

- **FREQi Bulk Data entries are selected by the FREQUENCY Case Control commands**
- **All FREQi Bulk Data entries with the same set ID are used. Therefore, FREQ, FREQ1, FREQ2, FREQ3, FREQ4, and FREQ5 entries may all be used in an analysis.**

FREQ ENTRY

FREQ — Frequency List

Defines a set of frequencies to be used in the solution of frequency response problems.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ	SID	F1	F2	F3	F4	F5	F6	F7	
	F8	F9	F10	-etc.-					

Example:

FREQ	3	2.98	3.05	17.9	21.3	25.6	28.8	31.2	
	29.2	22.4	19.3						

Field	Contents
SID	Set identification number. (Integer > 0)
Fi	Frequency value in units of cycles per unit time. (Real $\geq$ 0.0)

FREQ ENTRY

Remarks:

1. Frequency sets must be selected with the Case Control command FREQUENCY = SID.
2. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

   $$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}| ,$$

   where DFREQ is a user parameter, with a default of 10^{-5}. f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.
3. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

FREQ1 ENTRY

FREQ1 — Frequency List, Alternate Form 1

Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, frequency increment, and the number of increments desired.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ1	SID	F1	DF	NDF					

Example:

FREQ1	6	2.9	0.5	13					

Field	Contents
SID	Set identification number. (Integer > 0)
F1	First frequency in set. (Real ≥ 0.0)
DF	Frequency increment. (Real > 0.0)
NDF	Number of frequency increments. (Integer > 0; Default = 1)

FREQ1 ENTRY

Remarks:

1. FREQ1 entries must be selected with the Case Control command FREQUENCY = SID.
2. The units for F1 and DF are cycles per unit time.
3. The frequencies defined by this entry are given by

   $$f_i = F1 + DF \cdot (i-1)$$

   where i = 1 to (NDF + 1).
4. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

   $$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}| ,$$

   where DFREQ is a user parameter, with a default of 10^{-5}. f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.
5. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

FREQ2 ENTRY

FREQ2 Frequency List, Alternate Form 2

Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, final frequency, and the number of logarithmic increments desired.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ2	SID	F1	F2	NF					

Example:

FREQ2	6	1.0	8.0	6					

Field	Contents
SID	Set identification number. (Integer > 0)
F1	First frequency. (Real > 0.0)
F2	Last frequency. (Real > 0.0, F2 > F1)
NF	Number of logarithmic intervals. (Integer > 0; Default = 1)

FREQ2 ENTRY

Remarks:

1. FREQ2 entries must be selected with the Case Control command FREQUENCY = SID.
2. The units for F1 and F2 are cycles per unit time.
3. The frequencies defined by this entry are given by

   $$f_i = F1 \cdot e^{(i-1)d}$$

   where $d = \frac{1}{NF} ln \frac{F2}{F1}$ and $i = 1, 2, \ldots, (NF+1)$

   In the example above, the list of frequencies will be 1.0, 1.4142, 2.0, 2.8284, 4.0, 5.6569 and 8.0 cycles per unit time.
4. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

   $$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}| \quad ,$$

   where DFREQ is a user parameter, with a default of 10^{-5}. f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.
5. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

FREQ3 ENTRY

FREQ3 **Frequency List, Alternate 3**

Defines a set of excitation frequencies for modal frequency-response solutions by specifying number of excitation frequencies between two modal frequencies.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ3	SID	F1	F2	TYPE	NEF	CLUSTER			

Example:

FREQ3	6	20.0	2000.0	LINEAR	10	2.0			

Field	Contents
SID	Set identification number. (Integer > 0)
F1	Lower bound of modal frequency range in cycles per unit time. (Real ≥ 0.0 for TYPE = LINEAR and Real = 0.0 for TYPE = LOG)
F2	Upper bound of modal frequency range in cycles per unit time. (Real > 0.0; F2 ≥ F1; Default = F1)
TYPE	LINEAR or LOG. Specifies linear or logarithmic interpolation between frequencies. (Character; Default = "LINEAR")

FREQ3 ENTRY

Field	Contents
NEF	Number of excitation frequencies within each subrange including the end points. The first subrange is between F1 and the first modal frequency within the bounds. The second subrange is between first and second modal frequencies between the bounds. The last subrange is between the last modal frequency within the bounds and F2. (Integer > 1; Default = 10)
CLUSTER	Specifies clustering of the excitation frequency near the end points of the range. See Remark 6. (Real > 0.0; Default = 1.0)

Remarks:

1. FREQ3 applies only to modal frequency-response solutions (SOLs 111, 146, and 200) and is ignored in direct frequency response solutions.
2. FREQ3 entries must be selected with the Case Control command FREQUENCY = SID.
3. In the example above, there will be 10 frequencies in the interval between each set of modes within the bounds 20 and 2000, plus 10 frequencies between 20 and the lowest mode in the range, plus 10 frequencies between the highest mode in the range and 2000.
4. Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.

Note: For rest of FREQ3 entry, see MSC Nastran 2013 QRG

FREQ4 ENTRY

FREQ4 **Frequency List, Alternate Form 4**

Defines a set of frequencies used in the solution of modal frequency-response problems by specifying the amount of "spread" around each natural frequency and the number of equally spaced excitation frequencies within the spread.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ4	SID	F1	F2	FSPD	NFM				

Example:

FREQ4	6	20.0	2000.0	0.30	21				

Field	Contents
SID	Set identification number. (Integer > 0)
F1	Lower bound of frequency range in cycles per unit time. (Real ≥ 0.0; Default = 0.0)
F2	Upper bound of frequency range in cycles per unit time. (Real > 0.0; F2 > F1; Default = 1.0E20)
FSPD	Frequency spread, +/- the fractional amount specified for each mode which occurs in the frequency range F1 to F2. (1.0 > Real > 0.0; Default = 0.10)
NFM	Number of evenly spaced frequencies per "spread" mode. (Integer > 0; Default = 3; If NFM is even, NFM + 1 will be used.)

FREQ4 ENTRY

Remarks:

1. FREQ4 applies only to modal frequency-response solutions (SOLs 111, 146, and 200 and is ignored in direct frequency-response solutions.
2. FREQ4 entries must be selected with the Case Control command FREQUENCY = SID.
3. There will be NFM excitation frequencies between $(1 - FSPD) \cdot f_N$ and $(1 + FSPD) \cdot f_N$, for each natural frequency in the range F1 to F2.
4. In the example above there will be 21 equally spaced frequencies across a frequency band of $0.7 \cdot f_N$ to $1.3 \cdot f_N$ for each natural frequency that occurs between 20 and 2000. See Figure 8-98 for the definition of frequency spread.

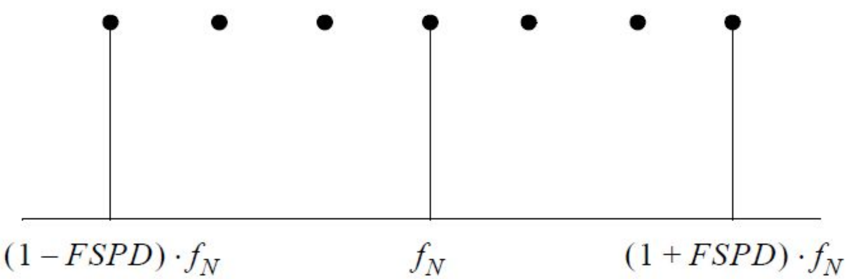


Figure 8-98 Frequency Spread Definition

Excitation frequencies may be based on natural frequencies that are not within the range (F1 and F2) as long as the calculated excitation frequencies are within the range. Similarly, an excitation frequency calculated based on natural frequencies within the range (F1 through F2) may be excluded if it falls outside the range.

FREQ4 ENTRY

Remarks:

5. The frequency spread can be used also to define the half-power bandwidth. The half-power bandwidth is given by $2 \cdot \xi \cdot f_N$, where ξ is the damping ratio. Therefore, if FSPD is specified equal to the damping ratio for the mode, NFM specifies the number of excitation frequency within the half-power bandwidth. See Figure 8-99 for the definition of half-power bandwidth.

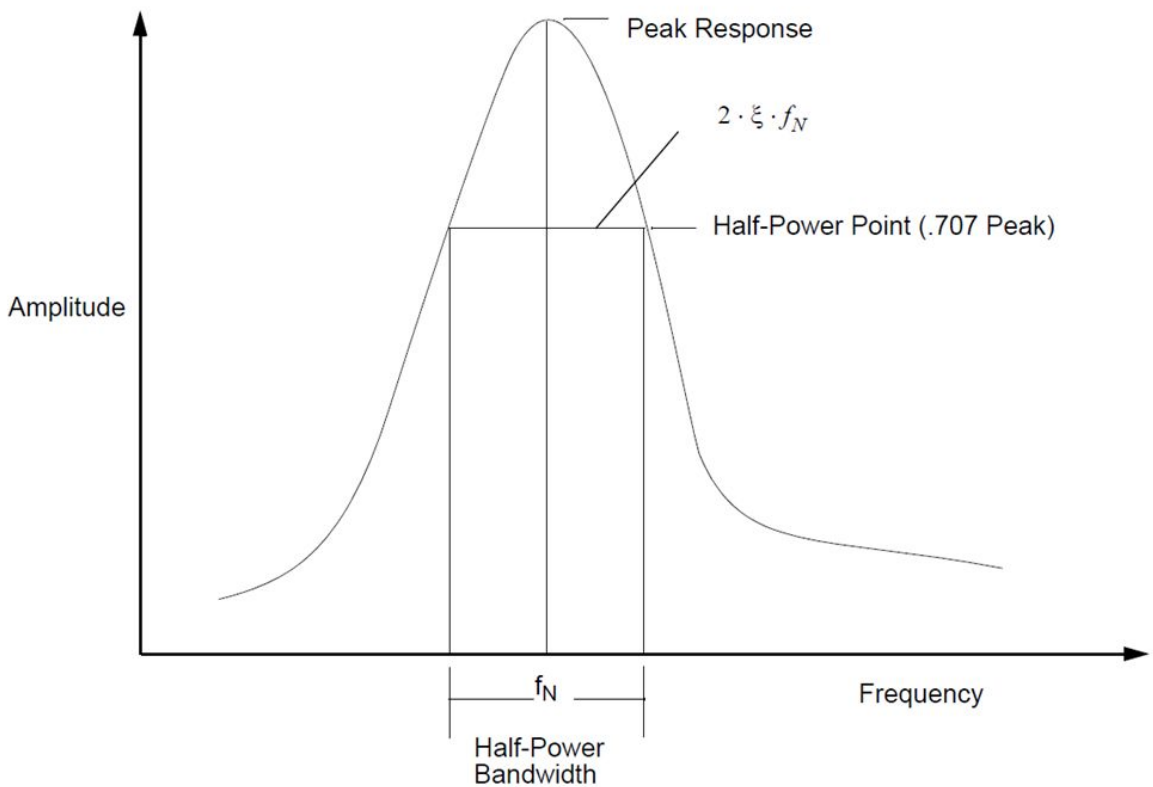


Figure 8-99 Half-Power Bandwidth Definition

FREQ4 ENTRY

Remarks:

6. Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.
7. All FREQi entries with the same set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

   $$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|$$

   where DFREQ is a user parameter with a default of 10^{-5}. The values f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.
8. In design optimization (SOL 200), the excitation frequencies are derived from the natural frequencies computed at each design cycle.
9. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

FREQ5 ENTRY

FREQ5 Frequency List, Alternate Form 5

Defines a set of frequencies used in the solution of modal frequency-response problems by specification of a frequency range and fractions of the natural frequencies within that range.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ5	SID	F1	F2	FR1	FR2	FR3	FR4	FR5	
	FR6	FR7	-etc.-						

Example:

FREQ5	6	20.0	2000.0	1.0	0.6	0.8	0.9	0.95	
	1.05	1.1	1.2						

Field	Contents
SID	Set identification number. (Integer > 0)
F1	Lower bound of frequency range in cycles per unit time. (Real ≥ 0.0; Default = 0.0)
F2	Upper bound of frequency range in cycles per unit time. (Real > 0.0; F2 > F1; Default = 1.0E20)
FRi	Fractions of the natural frequencies in the range F1 to F2. (Real > 0.0)

FREQ5 ENTRY

Remarks:

1. FREQ5 applies only to modal frequency-response solutions (SOLs 111, 146, and 200) and is ignored in direct frequency response solutions.
2. FREQ5 entries must be selected with the Case Control command FREQUENCY = SID.
3. The frequencies defined by this entry are given by

   $$f_i = FRi \cdot f_{N_i}$$

   where f_{N_i} are the natural frequencies in the range F1 through F2.
4. In the example above, the list of frequencies will be 0.6, 0.8, 0.9, 0.95, 1.0, 1.05, 1.1, and 1.2 times each natural frequency between 20 and 2000. If this computation results in excitation frequencies less then F1 and greater than F2, those computed excitation frequencies are ignored.

   Excitation frequencies may be based on natural frequencies that are not within the range (F1 and F2) as long as the calculated excitation frequencies are within the range. Similarly, an excitation frequency calculated based on natural frequencies within the range (F1 through F2) may be excluded if it falls outside the range.
5. Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.

FREQ5 ENTRY

Remarks:

6. All FREQi entries with the same set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

   $$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|$$

   where DFREQ is a user parameter with a default of 10^{-5}. The values f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.
7. In design optimization (SOL 200), the excitation frequencies are derived from the natural frequencies computed at each design cycle.
8. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

DYNAMIC DATA RECOVERY

- **The matrix method and the mode displacement method are used to recover data from modal frequency analysis.**

$$\frac{\text{Cost of matrix method}}{\text{Cost of mode displacement method}} = \frac{H}{F}$$

  - Where:
    - H = number of modes
    - F = number of excitation frequencies
- **The matrix method is the default and is cheaper for H < F and is the recommended method for most cases.**
  - PARAM, DDRMM, 0 (default)
- **The mode displacement method may be selected by**
  - PARAM, DDRMM,-1.

MODAL VERSUS DIRECT FREQUENCY RESPONSE

	Modal	Direct*
Small Model		X
Large Model	X	
Few Excitation Frequencies		X
Many Excitation Frequencies	X	
Modal Damping	X	

SORT1 VERSUS SORT2 OUTPUT

- SORT1 is the listing of the output for each excitation frequency
- SORT2 is the listing of the output for each requested grid point or element
- Primarily useful for frequency response analysis

	Transient Response		Frequency Response	
	Direct	Modal	Direct	Modal
Default	2	2	1	1
Deformed Plot Requests	1	1	1	1
XY Plot Requests	2	2	2	2

- If SORT1 and SORT2 are mixed in a run, all output will default to SORT1 for frequency response and SORT2 for transient response

SOLUTION CONTROL FOR FREQUENCY RESPONSE

- **Executive Control Section**
  - SOL (for required input see below)

Method	Structured Soultion Sequences
Direct	108
Modal	111

- **Case Control Section**
  - DLOAD (both - required)
  - LOADSET (both - optional)
  - METHOD (modal - required)
  - SDAMPING (modal - optional)
  - FREQUENCY (both - required)

SOLUTION CONTROL FOR FREQUENCY RESPONSE

- **Bulk Data Section**
  - ASET, OMIT (both - optional)
  - EIGRL or EIGR (modal - required)
  - FREQ (both - required)
  - RLOADi (both – required)
  - LSEQ (both - optional)
  - DAREA (both - optional)
  - DELAY (both - optional)
  - DPHASE (both - optional)
  - TABDMP1 (modal - optional)
  - DLOAD (both - optional)
  - TLOADi (both - optional)

CASE CONTROL OUTPUT

- **Grid Point**
  - ACCELERATION
  - DISPLACEMENT (or VECTOR)
  - OLOAD
  - SACCELERATION
  - SDISPLACEMENT
  - SVELOCITY
  - SVECTOR
  - SPCFORCES
  - VELOCITY
  - MPCFORCE

CASE CONTROL OUTPUT

- **Element Output**
  - ELSTRESS (or STRESS)
  - ELFORCE (or FORCE)
  - STRAIN
  - ESE
  - EKE
  - EDE
- **Special**
  - OFREQUENCY (control solution output frequencies)

FREQUENCY-DEPENDENT SPRINGS AND DAMPERS

- **Specifies stiffness as a function of forcing frequency**
- **Specifies damping as a function of forcing frequency**
- **Different impedance in different direction**
- **CBUSH**
  - Defines generalized spring, damper connectivity
- **PBUSH**
  - Defines nominal spring and damper values
- **PBUSHT**
  - Defines frequency-dependent spring and damper values

CBUSH ENTRY

CBUSH **Generalized Spring-and-Damper Connection**

Defines a generalized spring-and-damper structural element that may be nonlinear or frequency dependent.

Format:

1	2	3	4	5	6	7	8	9	10
CBUSH	EID	PID	GA	GB	GO/X1	X2	X3	CID	
	S	OCID	S1	S2	S3				

Example 1: Noncoincident grid points.

CBUSH	39	6	1	100	75				

Example 2: GB not specified.

CBUSH	39	6	1					0	

Example 3: Coincident grid points ($GA \neq GB$).

CBUSH	39	6	1					6	

CBUSH ENTRY

Example 4: Noncoincident grid points with fields 6 through 9 blank and a spring-damper offset.

CBUSH	39	6	1	600					
	0.25	10	0.	10.	10.				

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PBUSH entry. (Integer > 0; Default = EID)
GA, GB	Grid point identification number of connection points. See Remark 6. (GA > 0, GB ≥ 0 or blank)
Xi	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA. (Real)
GO	Alternate method to supply vector $\vec{v}$ using grid point GO. Direction of $\vec{v}$ is from GA to GO. $\vec{v}$ is then transferred to End A. See Remark 3. (Integer > 0)
CID	Element coordinate system identification. A 0 value means the basic coordinate system will be used. If CID is blank, then the element coordinate system is determined from GO or Xi. See Figure 8-22 and Remark 3. (Integer ≥ 0 or blank)
S	Location of spring damper. See Figure 8-22. (0.0 ≤ Real ≤ 1.0; Default = 0.5)
OCID	Coordinate system identification of spring-damper offset. See Remark 9. (Integer ≥ -1; Default = -1, which means the offset point lies on the line between GA and GB according to Figure 8-22)
S1, S2, S3	Components of spring-damper offset in the OCID coordinate system if OCID ≥ 0. See Figure 8-23 and Remark 9. (Real)

CBUSH ENTRY

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Figure 8-22 shows the bush element geometry.
3. CID ≥ 0 overrides GO and Xi. Then the element x-axis is along T1, the element y-axis is along T2, and the element z-axis is along T3 of the CID coordinate system. If the CID refers to a cylindrical coordinate system or a spherical coordinate system, then grid GA is used to locate the system. If for cylindrical or spherical coordinate, GA falls on the z-axis used to define them, it is recommended that another CID be selected to define the element x-axis.
4. For noncoincident grids ($GA \neq GB$), when GO or (X1, X2, X3) is given and no CID is specified, the line AB is the element x-axis and the orientation vector $\vec{v}$ lies in the x-y plane (similar to the CBEAM element).
5. For noncoincident grids ($GA \neq GB$), if neither GO or (X1, X2, X3) is specified and no CID is specified, then the line AB is the element x-axis. This option is valid only when K1 (or B1) or K4 (or B4) or both on the PBUSH entry are specified (but K2, K3, K5, K6 or B2, B3, B5, B6 are not specified). If K2, K3, K5, or K6 (or B2, B3, B5, or B6) are specified, a fatal message will be issued.
6. If the distance between GA and GB is less than .0001, or if GB is blank, then CID must be specified. GB blank implies that B is a grounded terminal, a gounded terminal is a point with a displacement that is constrained to zero.

CBUSH ENTRY

Remarks:

7. If PID references a PBUSHT entry, then the CBUSH element may only be defined in the residual structure and cannot be attached to any omitted degrees-of-freedom.
8. Element impedance output is computed in the CID coordinate system. The impedances in this system are uncoupled.
9. If OCID = -1 or blank (default) then S is used and S1, S2, S3 are ignored. If OCID ≥ 0, then S is ignored and S1, S2, S3 are used.

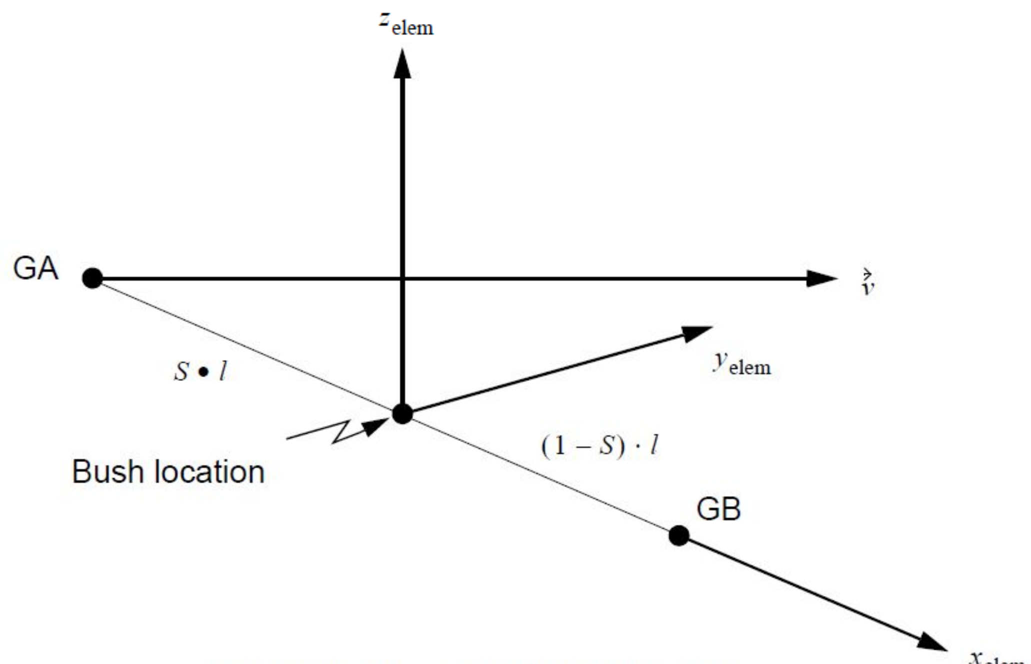


Figure 8-22 CBUSH Element

CBUSH ENTRY

Remarks:

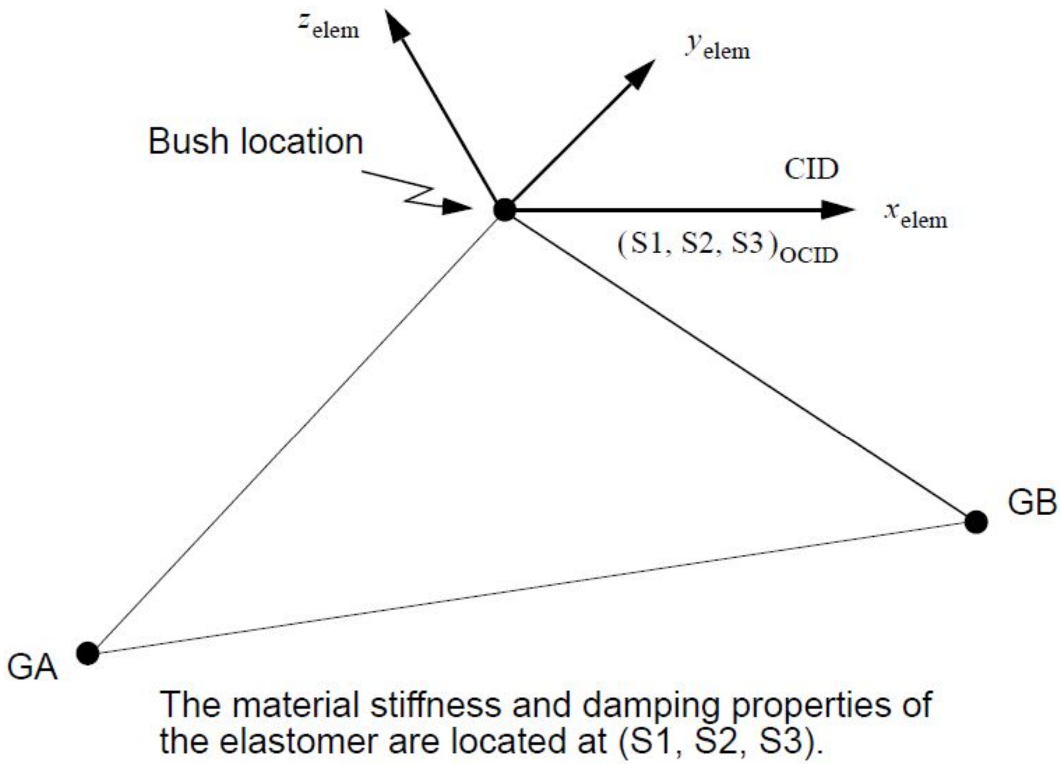


Figure 8-23 Definition of Offset S1, S2, S3

10. When CID ≥ 0, the element x-axis is set as in Remark 3. This means that the element force is always computed as $Ke \cdot (UB - UA)$; if $UA > UB$, a compressive force will result. This is unlike the GO or Xi options, where relative positive elongation in tension and relative negative elongation is compression.

CBUSH ENTRY

Remarks:

11. The CBUSH element is designed to satisfy rigid body equilibrium requirements. For noncoincident grids, internal rigid links connect the bush location to the grid locations. This results in coupling between translational and rotational degrees-of-freedom at the grids even when no rotational springs or dampers are specified on the PBUSH.
12. For SOL 600, if G0, X1, X2, X3, CID or OCID are entered, a Severe Warning will be issued and Marc will not run. SOL 600 translates the spring and damping terms in global coordinates of GA and GB and will use K1 to K6 and B1 to B6 whether or not GA and GB are coincident or not. S, S1, S2, S3 are ignored. CID is ignored unless it is zero, in which case K1 and B1 are along the axis of GA to GB and K2-K6 and B2-B6 are ignored if entered.

PBUSH ENTRY

PBUSH Generalized Spring-and-Damper Property

Defines the nominal property values for a generalized spring-and-damper structural element.

Format:

1	2	3	4	5	6	7	8	9	10
PBUSH	PID	"K"	K1	K2	K3	K4	K5	K6	
		"B"	B1	B2	B3	B4	B5	B6	
		"GE"	GE1	GE2	GE3	GE4	GE5	GE6	
		"RCV"	SA	ST	EA	ET			
		"M"	M						

Example 1:

Stiffness and structural damping are specified.

PBUSH	35	K	4.35	2.4				3.1	
		GE	.06						
		RCV	7.3	3.3					

PBUSH ENTRY

Example 2:

Damping force per unit velocity are specified.

PBUSH	35	B	2.3						

Field	Contents
PID	Property identification number. (Integer > 0)
"K"	Flag indicating that the next 1 to 6 fields are stiffness values in the element coordinate system. (Character)
Ki	Nominal stiffness values in directions 1 through 6. See Remarks 2. and 3. (Real; Default = 0.0)
"B"	Flag indicating that the next 1 to 6 fields are force-per-velocity damping. (Character)
Bi	Nominal damping coefficients in direction 1 through 6 in units of force per unit velocity. See Remarks 2., 3., and 9. (Real; Default = 0.0)
"GE"	Flag indicating that the next fields, 1 through 6 are structural damping constants. See Remark 7. (Character)
GEi	Nominal structural damping constant in directions 1 through 6. See Remarks 2. and 3. (Real; Default = 0.0)

PBUSH ENTRY

Field	Contents
"RCV"	Flag indicating that the next 1 to 4 fields are stress or strain coefficients. (Character)
SA	Stress recovery coefficient in the translational component numbers 1 through 3. (Real; Default = 1.0)
ST	Stress recovery coefficient in the rotational component numbers 4 through 6. (Real; Default = 1.0)
EA	Strain recovery coefficient in the translational component numbers 1 through 3. (Real; Default = 1.0)
ET	Strain recovery coefficient in the rotational component numbers 4 through 6. (Real; Default = 1.0)
"M"	Flag indicating that the following entries are mass and inertia properties for the CBUSH element.
M	Lumped mass of the CBUSH. (Real≥0.0; Default=0.0)

Note: for the rest of PBUSH entry, see MSC Nastran 2013 QRG

PBUSHT ENTRY

PBUSHT **Frequency Dependent or Nonlinear Force Deflection Spring and Damper Property**

Defines the frequency dependent properties or the stress dependent properties for a generalized spring and damper structural element.

Format:

1	2	3	4	5	6	7	8	9	10
PBUSHT	PID	"K"	TKID1	TKID2	TKID3	TKID4	TKID5	TKID6	
		"B"	TBID1	TBID2	TBID3	TBID4	TBID5	TBID6	
		"GE"	TGEID1	TGEID2	TGEID3	TGEID4	TGEID5	TGEID6	
		"KN"	TKNID1	TKIND2	TKNID3	TKIND4	TKIND5	TKIND6	
			FDC	FUSE	DIR	OPTION	LOWER	UPPER	
			FSRS	LRGR					

Example:

PBUSHT	35	K	72						
		B	18						

PBUSHT ENTRY

Field	Contents
PID	Property identification number that matches the identification number on a PBUSH entry. (Integer > 0)
"K"	Flag indicating that the next 1 to 6 fields are stiffness frequency table identification numbers. (Character)
TKIDi	Identification number of a TABLEDi entry that defines the stiffness vs. frequency relationship in directions 1 through 6. (Integer ≥ 0; Default = 0)
"B"	Flag indicating that the next 1 to 6 fields are force per velocity frequency table identification numbers. (Character)
TBIDi	Identification number of a TABLEDi entry that defines the force per unit velocity damping vs. frequency relationship in directions 1 through 6. (Integer ≥ 0; Default = 0)
"GE"	Flag indicating that the next field is a structural damping frequency table identification number. (Character)
TGEIDi	Identification number of a TABLEDi entry that defines the non-dimensional structural damping vs. frequency relationship. (Integer ≥ 0; Default = 0)
"KN"	Flag indicating that the next 1 to 6 fields are nonlinear force-deflection table identification numbers. (Character)
TKNIDi	Identification number of a TABLEDi entry that defines the force vs. deflection relationship in directions 1 through 6. (Integer ≥ 0; Default = 0)

Note: for the rest of PBUSHT entry, see MSC Nastran 2013 QRG

FREQUENCY DEPENDENT IMPEDANCE SAMPLE

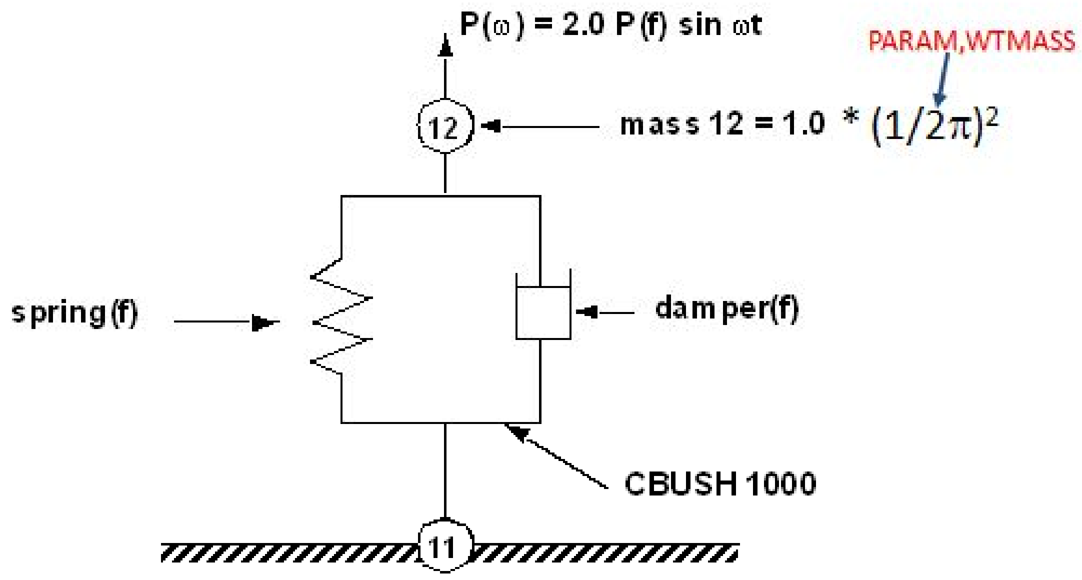


Table of Impedance

Forcing Frequency (Hz)	K(f)	B(f)	P(f)
0.9	0.81	0.286479	0.81
1	1	0.31831	1
1.1	1.21	0.350141	1.21
f_i	f_i^2	f_i/π	f_i^2

SAMPLE USING CBUSH ELEMENT

```
$
$                cbush1.dat
$
TIME 10
SOL 108
CEND
TITLE = VERIFICATION PROBLEM, FREQ. DEP. IMPEDANCE              BUSHVER
SUBTITLE = SINGLE DOF, CRITICAL DAMPING, 3 EXCITATION FREQUENCIES
ECHO = BOTH
SPC = 1002
DLOAD = 1
DISP = ALL
FREQ = 10
ELFO = ALL
BEGIN BULK
$ CONVENTIONAL INPUT FOR MOUNT
GRDSET,          ,                ,                ,                ,                ,                23456 $ PS
$ TIE DOWN EVERYTHING BUT THE 1 DOF
GRID,            11,              ,                0.,              0.,              0.0              $ GROUND
=,               12,              =,               =,               =,               ,                $ ISOLATED DOF
SPC1,            1002             123456           11               $ GROUND
CONM2,           12,              12,              ,                1.0              $ THE ISOLATED MASS
$
$                EID              PID              GA               GB               GO/X1            X2               X3               CID
$
CBUSH            1000             2000             11               12                                                                  0
$
PBUSH            2000             K                1.0
                                  B                0.0
$
PBUSHT           2000             K                2001
                                  B                2002
$
TABLED1, 2001 $ STIFFNESS TABLE
,                0.9              0.81,            1.0,             1.0,             1.1,             1.21             ENDT
TABLED1 2002 $ DAMPING TABLE
,                0.9              .2864789,  1.0,                   .318309,   1.1   ,                .3501409  ENDT
$CONVENTIONAL INPUT FOR FREQUENCY RESPONSE
PARAM,           WTMASS,          .0253303 $ 1/(2*PI)**2. GIVES FN=1.0
DAREA,           1,               12,              1,               2.  $CAUSES UNIT DEFLECTION
FREQ,            10,              0.9,             1.0,             1.1  $  BRACKET THE NATURAL FREQUENCY
RLOAD1,          1,               1,               ,                ,                3
TABLED1,3   $  TABLE FOR FORCE VS. FREQUENCY
,                0.9,             0.81,            1.,              1.,              1.1,             1.21,ENDT  $ P = K
ENDDATA
```

DISPLACEMENT OUTPUT FOR CBUSH ELEMENT

```
  FREQUENCY =   9.000000E-01
                                      C O M P L E X   D I S P L A C E M E N T   V E C T O R
                                                       (REAL/IMAGINARY)

      POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
0          11      G      0.0            0.0            0.0            0.0            0.0            0.0
                          0.0            0.0            0.0            0.0            0.0            0.0
0          12      G     -6.682744E-08   0.0            0.0            0.0            0.0            0.0
                         -1.000000E+00   0.0            0.0            0.0            0.0            0.0
1    VERIFICATION PROBLEM, FREQ. DEP. IMPEDANCE      BUSHVER              SEPTEMBER   2, 2011  MSC NASTRAN   2/ 2/11   PAGE    10
      SINGLE DOF, CRITICAL DAMPING, 3 EXCITATION FREQUENCIES

      FREQUENCY =   1.000000E+00
FREQUENCY =   1.000000E+00
                                      C O M P L E X   D I S P L A C E M E N T   V E C T O R
                                                       (REAL/IMAGINARY)

      POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
0          11      G      0.0            0.0            0.0            0.0            0.0            0.0
                          0.0            0.0            0.0            0.0            0.0            0.0
0          12      G     -1.046835E-07   0.0            0.0            0.0            0.0            0.0
                         -9.999999E-01   0.0            0.0            0.0            0.0            0.0
1    VERIFICATION PROBLEM, FREQ. DEP. IMPEDANCE      BUSHVER              SEPTEMBER   2, 2011  MSC NASTRAN   2/ 2/11   PAGE    11
      SINGLE DOF, CRITICAL DAMPING, 3 EXCITATION FREQUENCIES
0
      FREQUENCY =   1.100000E+00
                                      C O M P L E X   D I S P L A C E M E N T   V E C T O R
                                                       (REAL/IMAGINARY)

      POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
0          11      G      0.0            0.0            0.0            0.0            0.0            0.0
                          0.0            0.0            0.0            0.0            0.0            0.0
0          12      G     -6.855670E-08   0.0            0.0            0.0            0.0            0.0
                         -9.999998E-01   0.0            0.0            0.0            0.0            0.0
1    VERIFICATION PROBLEM, FREQ. DEP. IMPEDANCE      BUSHVER              SEPTEMBER   2, 2011  MSC NASTRAN   2/ 2/11   PAGE    12
      SINGLE DOF, CRITICAL DAMPING, 3 EXCITATION FREQUENCIES
```

FORCE OUTPUT FOR CBUSH ELEMENT

```
  FREQUENCY =   9.000000E-01
                          C O M P L E X   F O R C E S   I N   B U S H   E L E M E N T S   ( C B U S H )
                                                              (REAL/IMAGINARY)

                   ELEMENT-ID        FORCE-X       FORCE-Y       FORCE-Z       MOMENT-X      MOMENT-Y      MOMENT-Z
0                        1000      1.620000E+00  0.0           0.0           0.0           0.0           0.0
                                  -8.100001E-01  0.0           0.0           0.0           0.0           0.0
1    VERIFICATION PROBLEM, FREQ. DEP. IMPEDANCE      BUSHVER              SEPTEMBER   2, 2011  MSC NASTRAN   2/ 2/11   PAGE    13
      SINGLE DOF, CRITICAL DAMPING, 3 EXCITATION FREQUENCIES
0
      FREQUENCY =   1.000000E+00
                          C O M P L E X   F O R C E S   I N   B U S H   E L E M E N T S   ( C B U S H )
                                                              (REAL/IMAGINARY)

                   ELEMENT-ID        FORCE-X       FORCE-Y       FORCE-Z       MOMENT-X      MOMENT-Y      MOMENT-Z
0                        1000      2.000000E+00  0.0           0.0           0.0           0.0           0.0
                                  -1.000000E+00  0.0           0.0           0.0           0.0           0.0
1    VERIFICATION PROBLEM, FREQ. DEP. IMPEDANCE      BUSHVER              SEPTEMBER   2, 2011  MSC NASTRAN   2/ 2/11   PAGE    14
      SINGLE DOF, CRITICAL DAMPING, 3 EXCITATION FREQUENCIES
0
      FREQUENCY =   1.100000E+00
                          C O M P L E X   F O R C E S   I N   B U S H   E L E M E N T S   ( C B U S H )
                                                              (REAL/IMAGINARY)

                   ELEMENT-ID        FORCE-X       FORCE-Y       FORCE-Z       MOMENT-X      MOMENT-Y      MOMENT-Z
0                        1000      2.419999E+00  0.0           0.0           0.0           0.0           0.0
                                  -1.210000E+00  0.0           0.0           0.0           0.0           0.0
1    VERIFICATION PROBLEM, FREQ. DEP. IMPEDANCE      BUSHVER              SEPTEMBER   2, 2011  MSC NASTRAN   2/ 2/11   PAGE    15
      SINGLE DOF, CRITICAL DAMPING, 3 EXCITATION FREQUENCIES
```

EXERCISES

- **Perform the following workshops:**
  - Workshop #6, Direct Frequency Response
  - Workshop #7, Modal Frequency Response

SECTION 10

DYNAMIC EQUATIONS OF MOTION

DYNAMIC MATRIX ASSEMBLY

- **MSC 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**

DIRECT METHODS

- **The general dynamic equation used in the direct methods is:**

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

  - 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:**

$$\left[K_{dd}\right] = (1+ig)\left[K_{dd}^1\right] + \left[K_{dd}^2\right] + i\left[K_{dd}^4\right]$$
$$\left[B_{dd}\right] = \left[B_{dd}^1\right] + \left[B_{dd}^2\right]$$
$$\left[M_{dd}\right] = \left[M_{dd}^1\right] + \left[M_{dd}^2\right]$$

- **For transient response, the dynamic matrices are:**

$$\left[K_{dd}\right] = \left[K_{dd}^1\right] + \left[K_{dd}^2\right]$$
$$\left[B_{dd}\right] = \left[B_{dd}^1\right] + \left[B_{dd}^2\right] + \frac{g}{\omega_3}\left[K_{dd}^1\right] + \frac{1}{\omega_4}\left[K_{dd}^4\right]$$
$$\left[M_{dd}\right] = \left[M_{dd}^1\right] + \left[M_{dd}^2\right]$$

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 [Ke] of an individual structural element by an element damping factor ge and combining the 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).

DYNAMIC MATRIX DEFINITIONS

- g, w_3, w_4 are the constants specified by the user
- The structural matrices $[K_{aa}]$, $[K^4{}_{aa}]$,$[M_{aa}]$, and $[B_{aa}]$ are expanded by the addition of zeroes in the rows and columns corresponding to extra points to form $[M^1{}_{dd}]$, $[B^1{}_{dd}]$, $[K^1{}_{dd}]$, and $[K^4{}_{dd}]$
- Only the $[K^2{}_{dd}]$, $[B^2{}_{dd}]$, and $[M^2{}_{dd}]$ matrices can reference extra points
- The direct input matrices $[K^2{}_{pp}]$, $[B^2{}_{pp}]$, and $[M^2{}_{pp}]$ are processed through the multipoint and single-point constraint elimination and any reduction procedure
- Note: The extra points are unaffected by any constraint or reduction procedures. The constraint and reduction procedures can only eliminate grid or scalar point DOFs but not extra points.
- The matrices $[K_{dd}]$, $[B_{dd}]$, and $[M_{dd}]$ are examined to identify rows and columns which are null in all three matrices. For transient and frequency response, $[K_{dd}]$ is augmented by placing unity in each null row and column. In complex eigenvalue analysis, null rows and columns are discarded from $[K_{dd}]$, $[B_{dd}]$, and $[M_{dd}]$.

MODAL METHODS

- **The general dynamic equation used in the modal methods is:**

$$\left[M_{hh}p^2 + B_{hh}p + K_{hh}\right]\{u_n\} = \{p_h\}$$

  – Where:
    - p = a derivative operator
    - u_h = the union of the modal coordinates x_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 u_h to u_d 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:**

$$[K_{hh}] = [k_i] + [\phi_{dh}]^T \left(ig\left[K_{dd}^1\right] + \left[K_{dd}^2\right] + i\left[K_{dd}^4\right]\right)[\phi_{dh}]$$
$$[B_{hh}] = [b_i] + [\phi_{dh}]^T \left(\left[B_{dd}^1\right] + \left[B_{dd}^2\right]\right)[\phi_{dh}]$$
$$[M_{hh}] = [m_i] + [\phi_{dh}]^T \left[M_{dd}^2\right][\phi_{dh}]$$

  – Where:
    - $[m_i]$ = a diagonal matrix with terms $m_{ii} = [\phi_{ai}]^T[M_{aa}][\phi_{ai}]$
    - $[b_i]$ = a diagonal matrix with terms $b_{ii} = \omega_i g(\omega_i) m_{ii}$
  – where
    - ω_i = the radian frequency of the *i*-th normal mode
    - $g(\omega_i)$ = a damping factor obtained from interpolation of a user-supplied table (TABDMP1)
    - $[k_i]$ = a diagonal matrix with terms $k_{ii} = \omega^2_i m_{ii}$

- **If parameter KDAMP = -1 then**

$$m_{ii} = m_{ii}$$
$$b_{ii} = 0$$
$$k_{ii} = \left(1 + ig(\omega_i)\right)k_{ii}$$

  – Note: default for parameter KDAMP = 1

MODAL METHODS

- $g(\omega_i)$ is a damping factor obtained from the interpolation of a user-supplied table (TABDMP1)
- $[m_i]$, $[b_i]$, 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:

$$[K_{hh}] = [k_i] + [\phi_{dh}]^T [K_{dd}^2][\phi_{dh}]$$

$$[B_{hh}] = [b_i] + [\phi_{dh}]^T \left([B_{dd}^1] + [B_{dd}^2] + \frac{g}{\omega_3}[K_{dd}^1] + \frac{1}{\omega_4}[K_{dd}^1] \right) [\phi_{dh}]$$

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

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

SECTION 11

RESIDUAL VECTOR METHODS

CONCEPT OF MODAL APPROACH

- **Assume response can be represented as a linear combination of calculated modes**

$$\{ U \} = [\phi] \{ \xi \}$$

- **Number of possible modes = number of degrees of freedom with mass on them**
- **Same results as direct if all modes are retained**
  - Not practical
  - Defeats purpose of modal approach

CONCEPT OF MODAL APPROACH

$$\{U\} = [\phi]\{\xi\} = [\phi]_r\{\xi\}_r + [\phi]_n\{\xi\}_n$$

– where
  - $[\ \phi\ ]_r$ = modes that are retained
  - $[\ \phi\ ]_n$ = modes not retained

- **$[\ \phi\ ]_r$ is usually a small subset of $[\ \phi\ ]$**
- **Quality of modal solution depends on how well a linear combination of modes$[\ \phi\ ]_{r,}$ those are retained, can represent the actual solution due to the applied loads**

COMPENSATING FOR THE MISSING MODAL CONTENTS

- **Two methods are available to compensate for the missing modal contents**
  - Residual Vector
    - Recommended method and is turned on by default for most modal solutions
  - Mode Acceleration

RESIDUAL VECTOR

- **Augment modes with static vectors obtained from static loading**
- **The response of the neglected modes tends to be static if these frequencies are high as compared to the excitation frequency**
  - As $(\omega/\omega_n) << 1$, dynamic magnification → 1 (see Section 1)
  - Excellent approximation of missing modes if the above condition is satisfied
- **Improves modal solutions in all cases**
- **Recommended to be included for all response analysis using the modal approach (the default)**
- **Supports superelement analysis**
- **Two eigenvalue tables printed**
  - Original eigenvalue table
  - Second eigenvalue table with the additional eigenvalues appended at the end – one for each additional residual vector

RESIDUAL VECTOR

- **Re-orthogonalization is performed to ensure that linearly dependent vectors are removed**
- **Residual Vectors are simply static displacements from various sources that are added to make sure that a linear combination of the modes can represent the static solution. Residual Vectors come from following sources:**
  - Inertial forces due to rigid body motion
  - Applied loads/Static loads
  - Structural, viscous, and inertial forces due to enforced motion
  - Forces at user specified discrete degrees of freedom (RVDOFi entries)
  - Discrete damping forces due to viscous elements (CDAMPi and CVISC entries)

RESIDUAL VECTOR – RESVEC ENTRY

- **Residual Vectors can be requested with new RESVEC Case Control command**

$$\text{RESVEC}\left[\left(\begin{bmatrix}\text{INRLOD}\\\text{NOINRL}\end{bmatrix},\begin{bmatrix}\text{APPLOD}\\\text{NOAPPL}\end{bmatrix},\begin{bmatrix}\text{ADJLOD}\\\text{NOADJLOD}\end{bmatrix},\begin{bmatrix}\text{RVDOF}\\\text{NORVDO}\end{bmatrix},\begin{bmatrix}\text{DAMPLOD}\\\text{NODAMP}\end{bmatrix},\begin{bmatrix}\text{DYNRSP}\\\text{NODYNRSP}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{SYSTEM/NOSYTEM}\\\text{COMPONENT/NOCOMPONENT}\\\text{BOTH or YES}\\\text{NO}\end{Bmatrix}$$

Describer	Meaning
INRLOD/ NOINRL	Controls calculation of residual vectors based on inertia relief (Default = INRLOD).
APPLOD/ NOAPPL	Controls calculation of residual vectors based on applied loads (Default = APPLOD).
ADJLOD/ NOADJLOD	Controls calculation of residual vectors based on adjoint load vectors (SOL 200 only; Default = ADJLOD).
RVDOF/ NORVDOF	Controls calculation of residual vectors based on RVDOFi entries (Default = RVDOF).
DAMPLOD/ NODAMP	Controls calculation of residual vectors based on viscous damping (Default = DAMPLOD).
DYNRSP/ NODYNRSP	Controls whether the residual vectors will be allowed to respond dynamically in the modal transient or frequency response solution. See Remark 5. (Default = DYNRSP).

RESIDUAL VECTOR – RSEVEC ENTRY

Describer	Meaning
SYSTEM/ NOSYSTEM	Controls calculation of residual vectors for system (a-set) modes. For NOSYSTEM, describers inside the parentheses are ignored. See Remark 2. for default.
COMPONENT/ NOCOMPONENT	Controls calculation of residual vectors for component (superelement or o-set) modes. For NOCOMPONENT, describers inside the parentheses are ignored. See Remark 2. for default.
BOTH or YES	Requests calculation of residual vectors for both system modes and component modes. See Remark 2. for default.
NO	Turns off calculation of residual vectors for both system and component modes, and describers inside the parentheses are ignored. See Remark 2. for default.

RESIDUAL VECTOR PROCESSING

- **Ensure loads are linearly independent from modal inertial forces**
- **Determine base vectors from static response due to loads**
- **Ensure base vectors are linearly independent**
  - Independent from modal vectors
  - Independent from other base vectors
- **Orthogonalize base vectors with respect to the modal vectors to produce residual vectors. These vectors will result in diagonal mass and stiffness matrices.**

SECTION 12

ENFORCED MOTION

ENFORCED MOTION IN DYNAMIC ANALYSIS

- **Used to analyze constrained structures with base input acceleration, displacement, and velocity**
- **Common examples are earthquakes (for transient analysis), swept-sine shaker test simulation (for frequency response analysis), and automobile suspensions**

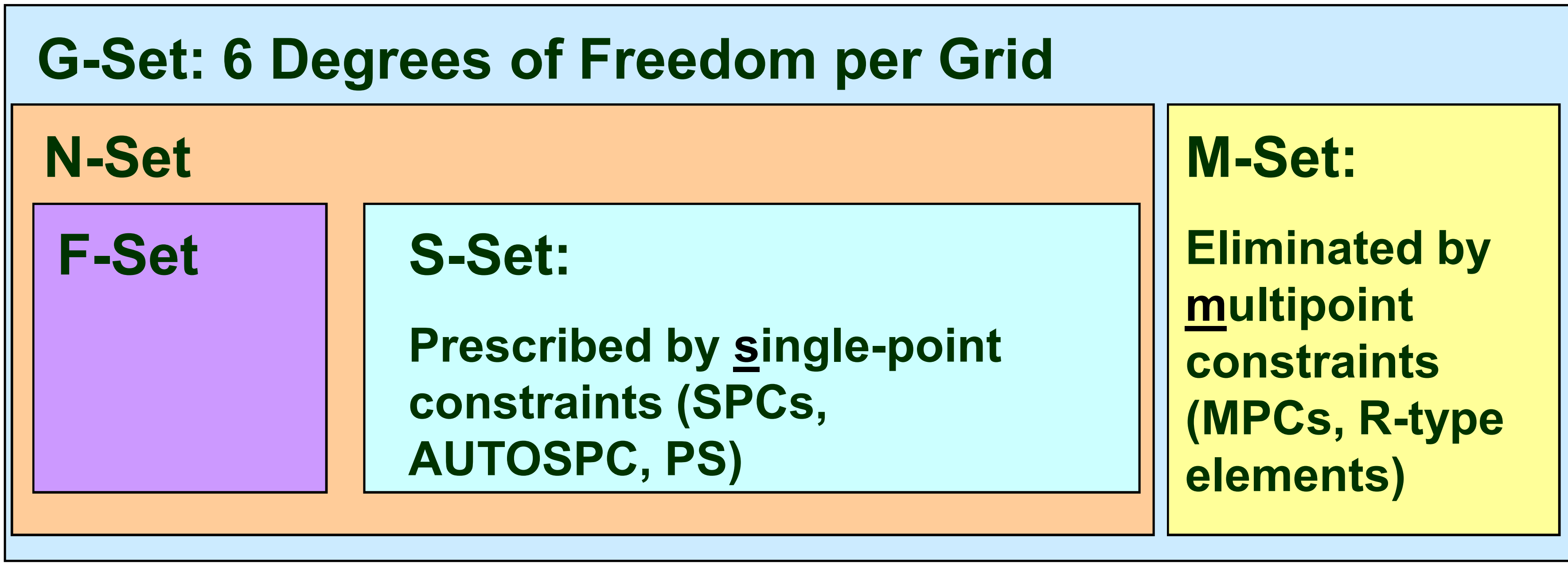
ANALYSIS METHODS

- **Four available methods:**
  1. Direct Specification of enforced displacement, velocity, or acceleration
     - This is the recommended method and the only one that will be discussed in this lecture.
     - This method of enforced motion can be specified by the direct specification of displacements, velocities or accelerations via SPC / SPC1 and SPCD Bulk Data entries
     - Interface is very similar to enforced motion in static analysis

  1. Large Mass
  2. Large Stiffness (Enforced displacement only)
  3. Lagrange Multiplier

DEGREE OF FREEDOM SETS

- **In the direct method, enforced motion is applied to the degrees of freedom in the S-set**



PARAMETERS FOR DIRECT ENFORCED MOTION

- **Two parameters controlling Enforced Motion**
  - ENFMETH
    - Controls the solution method
  - ENFMOTN
    - Controls the output

PARAMETERS

- **ENFMETH (Default = ABS)**
  - selects the method for solving enforced motion in dynamic analysis using SPC/SPCD entries. The total solution of a dynamic enforced motion analysis can be regarded as a combination of a static and dynamic enforced motion
  - It is recommended to use PARAM,ENFMETH,REL in order to get correct results at lower frequencies
  - PARAM,ENFMETH can be set to either REL or ABS
    - ABS – does it on 1 step
    - REL – does it in 2 steps

PARAMETERS

- **ENFMOTN (Default = ABS)**
  - This parameter is designed for use with the SPC/SPCD method of enforced motion specification in SOLs 108, 109, 111, 112, 146, and 200.
  - The default value of ABS implies that the results of the analysis represent absolute motion of the model. If the value is specified as REL, then the results represent motion relative to the enforced motion of the base. In the case of modal dynamic analysis (SOL 111 and SOL 112), this latter scenario is equivalent to employing the large mass approach and excluding the rigid body modes from the analysis.
  - This parameter is for Output only.
  - PARAM,ENFMOTN can be set to either REL or ABS
    - PARAM,ENFMOTN,ABS – absolute motion of the model (default)
    - PARAM,ENFMOTN, REL – relative motion to the enforced motion of the base.

BASIC EQUATIONS FOR USING ABSOLUTE MOTION

- **For the N-set, the equation of motion is as follows:**

(1)
$$\begin{bmatrix} M_{ff} & M_{fs} \\ M_{sf} & M_{ss} \end{bmatrix} \begin{Bmatrix} \ddot{u}_f \\ \ddot{u}_s \end{Bmatrix} + \begin{bmatrix} B_{ff} & B_{fs} \\ B_{sf} & B_{ss} \end{bmatrix} \begin{Bmatrix} \dot{u}_f \\ \dot{u}_s \end{Bmatrix} + \begin{bmatrix} K_{ff} & K_{fs} \\ K_{sf} & K_{ss} \end{bmatrix} \begin{Bmatrix} u_f \\ u_s \end{Bmatrix} = \begin{Bmatrix} p_f \\ p_s + q_s \end{Bmatrix}$$

  - Where
    - P_f and P_s = known applied loads
    - U_s = known prescribed displacements
    - Q_s = known constrain forces

EQUATIONS FOR TRANSIENT RESPONSE

- **Equation 1 can be solved for the F-set displacements:**

(2) $$\mathbf{M}_{ff}\ddot{\mathbf{u}}_f + \mathbf{B}_{ff}\dot{\mathbf{u}}_f + \mathbf{K}_{ff}\mathbf{u}_f = \mathbf{p}_f - \left(\mathbf{M}_{fs}\ddot{\mathbf{u}}_s + \mathbf{B}_{fs}\dot{\mathbf{u}}_s + \mathbf{K}_{fs}\mathbf{u}_s\right)$$

- **Subsequently, the constraint forces are obtained from the second matrix equation:**

(3) $$\mathbf{q}_s = -\mathbf{p}_s + \begin{bmatrix}\mathbf{M}_{sf} & \mathbf{M}_{ss}\end{bmatrix}\begin{Bmatrix}\ddot{\mathbf{u}}_f \\ \ddot{\mathbf{u}}_s\end{Bmatrix} + \begin{bmatrix}\mathbf{B}_{sf} & \mathbf{B}_{ss}\end{bmatrix}\begin{Bmatrix}\dot{\mathbf{u}}_f \\ \dot{\mathbf{u}}_s\end{Bmatrix} + \begin{bmatrix}\mathbf{K}_{sf} & \mathbf{K}_{ss}\end{bmatrix}\begin{Bmatrix}\mathbf{u}_f \\ \mathbf{u}_s\end{Bmatrix}$$

EQUATIONS FOR FREQUENCY RESPONSE

- **In frequency response analysis, the F-set displacements are obtained from:**

(4) $$\left(-\omega^2\mathbf{M}_{ff}+i\omega\mathbf{B}_{ff}+\mathbf{K}_{ff}\right)\mathrm{U}_f=\mathbf{P}_f-\left(-\omega^2\mathbf{M}_{fs}+i\omega\mathbf{B}_{fs}+\mathbf{K}_{fs}\right)\mathrm{U}_s$$

- **The constraint forces read:**

(5) $$\mathbf{Q}_s=-\mathbf{P}_s+\left(-\omega^2\begin{bmatrix}\mathbf{M}_{sf} & \mathbf{M}_{ss}\end{bmatrix}+i\omega\begin{bmatrix}\mathbf{B}_{sf} & \mathbf{B}_{ss}\end{bmatrix}+\begin{bmatrix}\mathbf{K}_{sf} & \mathbf{K}_{ss}\end{bmatrix}\right)\begin{Bmatrix}\mathbf{U}_f\\ \mathbf{U}_s\end{Bmatrix}$$

RELATIVE MOTION APPROACH

(6) $\{u_f\} = \{Y_f\} + \{X\}$

– where
  - $\{ U_f \}$ = absolute motion
  - $\{ Y_f \}$ = relative motion
  - $\{ X \}$ = base motion

$K_{ff}U_f + K_{fs}U_s = P_f$

$K_{ff}Y_f + K_{ff}X + K_{fs}U_s = P_f$

Rel. Motion: $K_{ff}Y_f = P_f$

➔ $K_{ff}X + K_{fs}U_s = 0$

$X = -K_{ff}^{-1} K_{fs}U_s$

- **{ X } - motion of f-set DOFs due to specified enforced motion of the s-set DOFs (computed purely from static considerations)**

$$\{\mathbf{X}\} = -[\mathbf{K}_{\mathbf{ff}}]^{-1}[\mathbf{K}_{\mathbf{fs}}]\{\mathbf{U}_{\mathbf{s}}\} = [\mathbf{Z}]\{\mathbf{U}_{\mathbf{s}}\}$$

RELATIVE MOTION APPROACH

- **Substituting equations 6 and 7 into equation 2, we get:**

$$\mathbf{M_{ff}\ddot{Y}_f + B_{ff}\dot{Y}_f + K_{ff}Y_f = p_f - (M_{ff}Z + M_{fs})\ddot{u} - (B_{ff}Z + B_{fs})\dot{u}_s}$$

- **Solve for $\ddot{Y}_f$, $\dot{Y}_f$, and Y_f**
- **$\ddot{U}_f$, $\dot{U}_f$, and U_f are obtained using equation 6**
  - This is the default output
- **To get $\ddot{Y}_f$, $\dot{Y}_f$, and Y_f set PARAM,ENFMOTN,REL**

USER INTERFACE

- SPC / SPC1 Bulk Data entries are used to identify the degrees of freedom with enforced motion. These entries are activated by an SPC Case Control command.
- SPCD Bulk Data entries are used to define the enforced motion. These entries are referenced by the EXCITEID field of TLOADi or RLOADi Bulk Data entries.
- The TYPE field of the TLOADi or RLOADi Bulk Data entries indicates the type of enforced motion
- For modal method, residual vectors should always be included which is the default

THE TYPE FIELD

- **The type of excitation is defined in field 5 of TLOADi Bulk Data entries or in field 8 of RLOADi Bulk Data entries:**

TYPE	TYPE of Dynamic Excitation
0, L, LO, LOA or LOAD	Applied load (force or moment) (Default)
1, D, DI, DIS or DISP	Enforced displacement using SPC/SPCD data
2, V, VE, VEL or VELO	Enforced velocity using SPC/SPCD data
3, A, AC, ACC or ACCE	Enforced acceleration SPC/SPCD data

- **The character fields may be shortened to as little as a single character**

EXAMPLE: EXECUTIVE AND CASE CONTROL

```
SOL 111
CEND
$
TITLE   =Example for Direct Enforced Motion
SUBTITLE=Modal Frequency Response Analysis
$
SPC     =1
METHOD  =10
FREQUENCY=20

PARAM,ENFMETH,REL
$
SET 1 = 1000,1001
ACCELERATION(SORT2,PRINT,PHAS)=1
$
SUBCASE 1
   LABEL=Unit Acceleration in x-Direction
   DLOAD=100
$
SUBCASE 2
   LABEL=Unit Acceleration in y-Direction
   DLOAD=200
$
```

EXAMPLE: BULK DATA DECK

```
BEGIN BULK
$
PARAM, G, 0.02                       $ 2% Structural Damping
SPC1, 1, 3456, 1000                  $ z-Displ. and Rotations are fixed
SPC1, 1, 12,   1000                  $ x- and y-Accelerations are prescribed
$
$ Modal Reduction
EIGRL, 10,, 150.                     $ Modes up to 150Hz

$
$ Base Motion Excitation
$
RLOAD1, 100, 1001,,, 10,, A          $ Load of Subcase 1:
SPCD,  1001, 1000, 1, 1.             $    Unit x-Acceleration
$
RLOAD1, 200, 1002,,, 10,, A          $ Load of Subcase 2:
SPCD,  1002, 1000, 2, 1.             $    Unit y-Acceleration
$
TABLED1, 10                          $ Constant for all Frequencies
       , 0., 1., 100., 1., ENDT
FREQ1, 20, 1., 1., 49                $ Frequency Range from 1Hz to 50Hz
$
INCLUDE 'tower.bdf'                  $ Structural Model
$
ENDDATA
```

INITIAL CONDITION SPECIFICATION FOR ENFORCED MOTION (SPC/SPCD)

- **Enforced acceleration or velocity in transient analysis (using SPC/SPCD) requires integration to compute corresponding enforced velocities and/or displacements**
- **Above integration involves the use of initial conditions**
- **The US0 (initial displacement) and VS0 (initial velocity) fields on the TLOADi entries in conjunction with the EXCITEID field set the initial conditions**

INITIAL CONDITION SPECIFICATION FOR ENFORCED MOTION (SPC/SPCD)

- **Format of TLOAD1 is shown below:**

TLOAD1	SID	EXCITEID	DELAY	TYPE	TID	US0	VS0		

- **Format of TLOAD2 is shown below:**

TLOAD2	SID	EXCITEID	DELAY	TYPE	T1	T2	F	P	
	C	B	US0	UV0					

  - US0 – initial displacement
  - VS0 – initial velocity
  - US0 and VS0 can be used in conjunction with other initial condition (TIC) at independent DOFs

EXAMPLE SPECIFYING INITIAL DISPLACEMENT

- **A simple 2 dof system is used to illustrate the use of initial displacement.**
  - The following enforced displacement is applied to grid 1, x direction

$$d(t) = \cos(2\pi 5t)$$

    - Note that d(0) = 1.0
  - The following enforced velocity is applied to grid 1, y direction

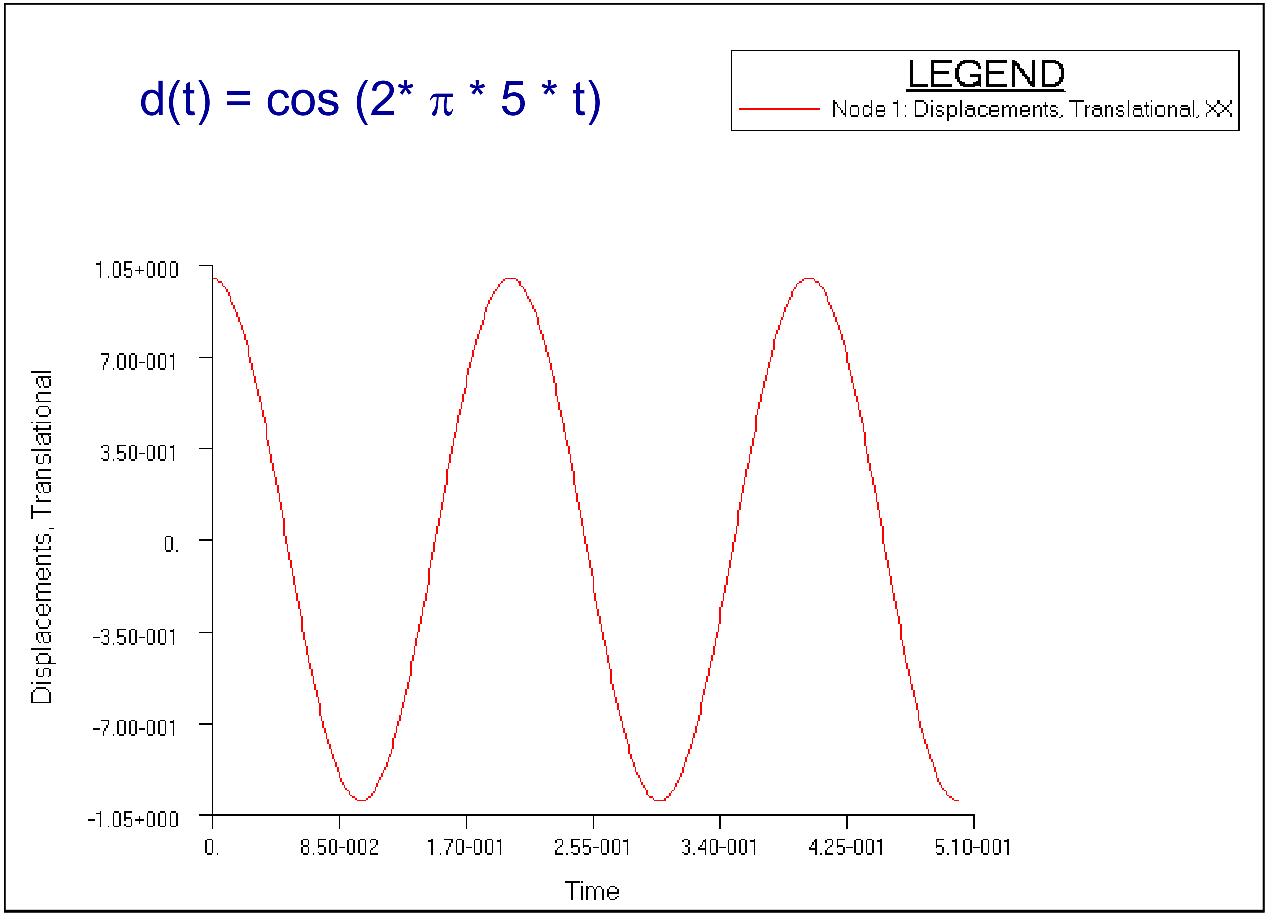
$$v(t) = -10\pi \sin(2\pi 5t)$$

    - Note that v(0) = 0
  - The following enforced acceleration is applied to grid 1, z direction

$$a(t) = -100\pi^2 \cos(2\pi 5t)$$

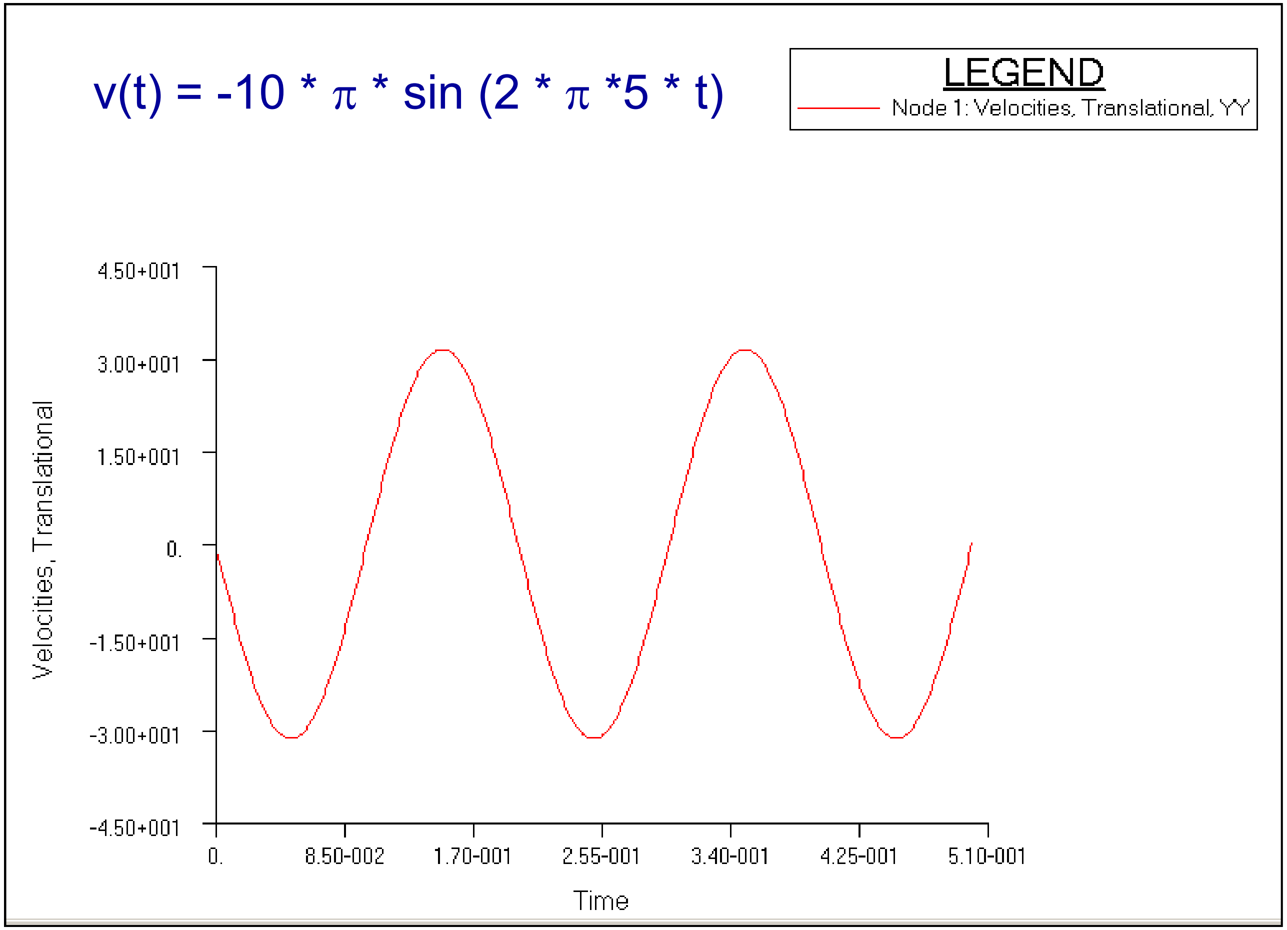
EXAMPLE SPECIFYING INITIAL DISPLACEMENT

- **Applied enforced displacement at grid 1 in x direction**



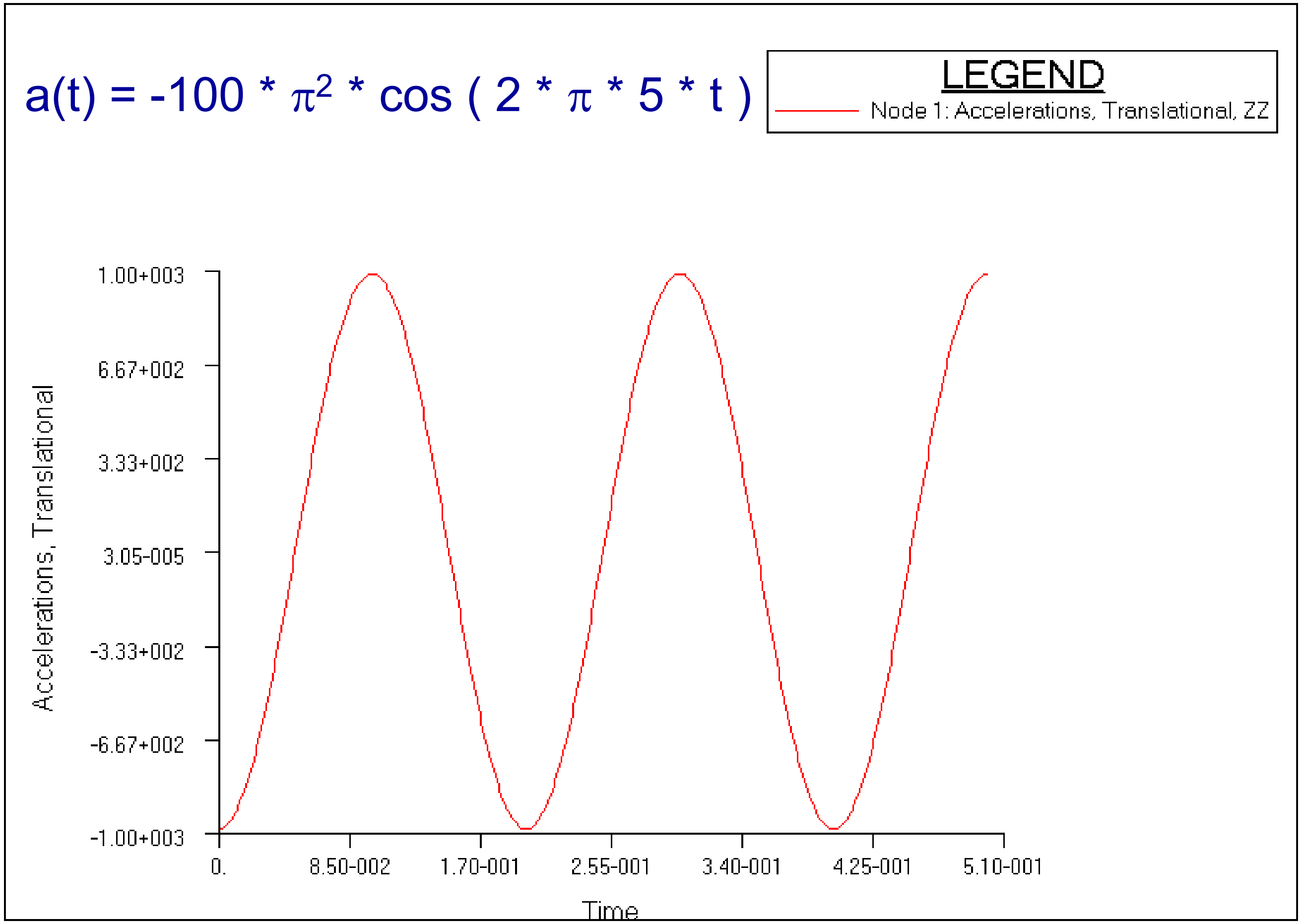
EXAMPLE SPECIFYING INITIAL DISPLACEMENT

- **Applied enforced velocity at grid 1 in y direction**



EXAMPLE SPECIFYING INITIAL DISPLACEMENT

- **Applied enforced acceleration at grid 1 in z direction**



EXAMPLE SPECIFYING INITIAL DISPLACEMENT

- **Without US0, the enforced displacement is off when using enforced velocity and acceleration**
  - This affects the subsequent displacement, velocity, and acceleration output
- **With the appropriate US0 added to the enforced velocity and enforced acceleration input, the correct enforced displacement is applied**
  - The product of US0 and the value specified on the SPCD should equal to the enforced value

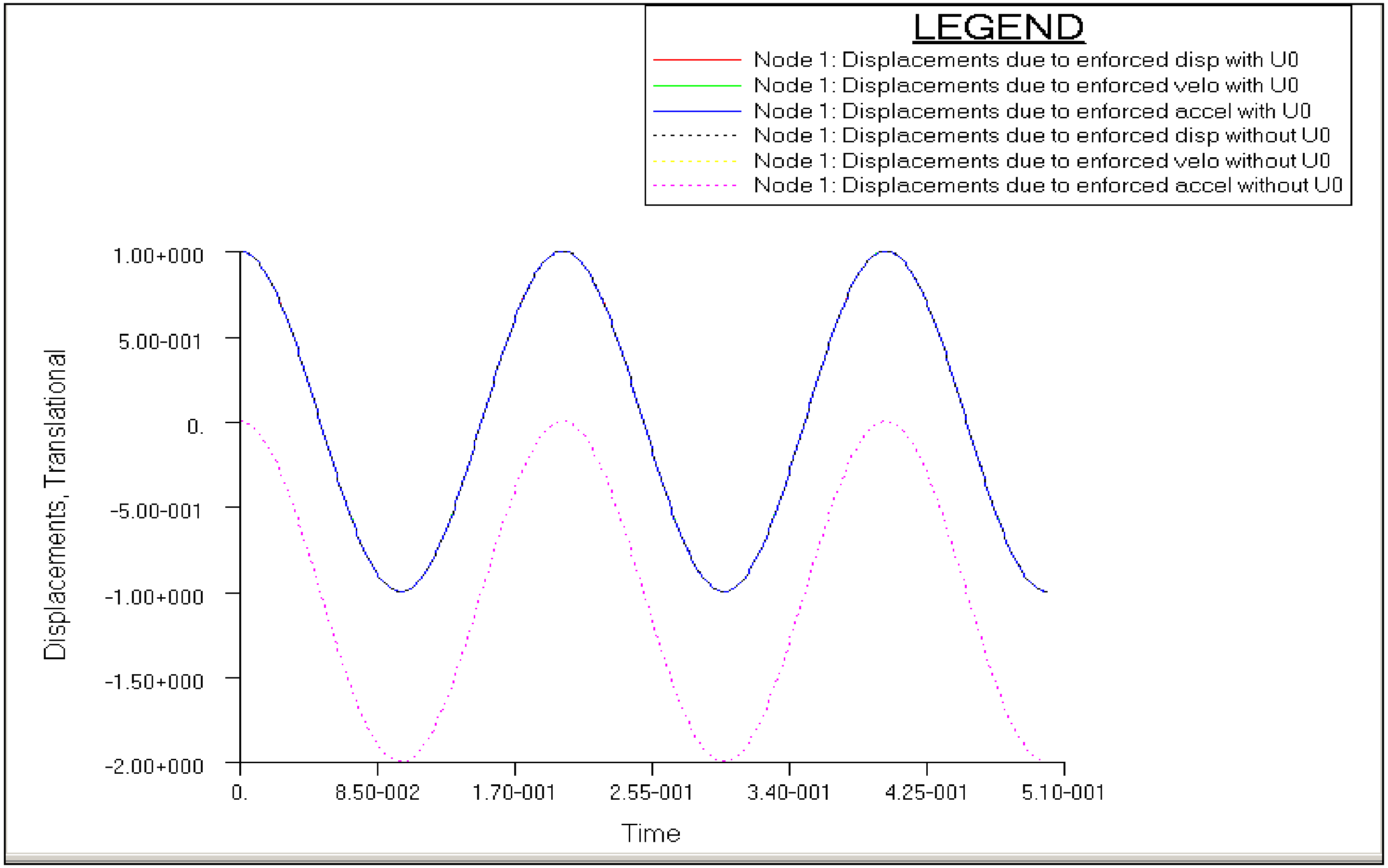
EXAMPLE SPECIFYING INITIAL DISPLACEMENT

```
begin bulk
param,post,0
tstep,40,500,.001,1
grid,1,,,,,,123
grid,2
dload,5000,1.0,1.0,30,1.0,300,1.0,3000
$
$ Enforced displacement
$         at Grid 1 - Component 1
$ Corresponding response
$         is at Grid 2 - Component 1
$
celas2,100,986.9604,1,1,2,1
cmass2,200,1.,2,1
spcd,20,1,1,1.
tload2,30,20,,d,0.,1000.,5.
$
$ Enforced velocity
$         at Grid 1 - Component 2
$ Corresponding response
$         is at Grid 2 - Component 2
```

```
celas2,1000,986.9604,1,2,2,2
cmass2,2000,1.,2,2
spcd,200,1,2,-31.4159
tload2,300,200,,v,0.,1000.,5.,-90.
,,,-3.183-2
$
$ Enforced acceleration
$         at Grid 1 - Component 3
$ Corresponding response
$          is at Grid 3 - Component 3
$
celas2,10000,986.9604,1,3,2,3
cmass2,20000,1.,2,3
spcd,2000,1,3,-986.960
tload2,3000,2000,,a,0.,1000.,5.
,,,-1.013-3
enddata
```

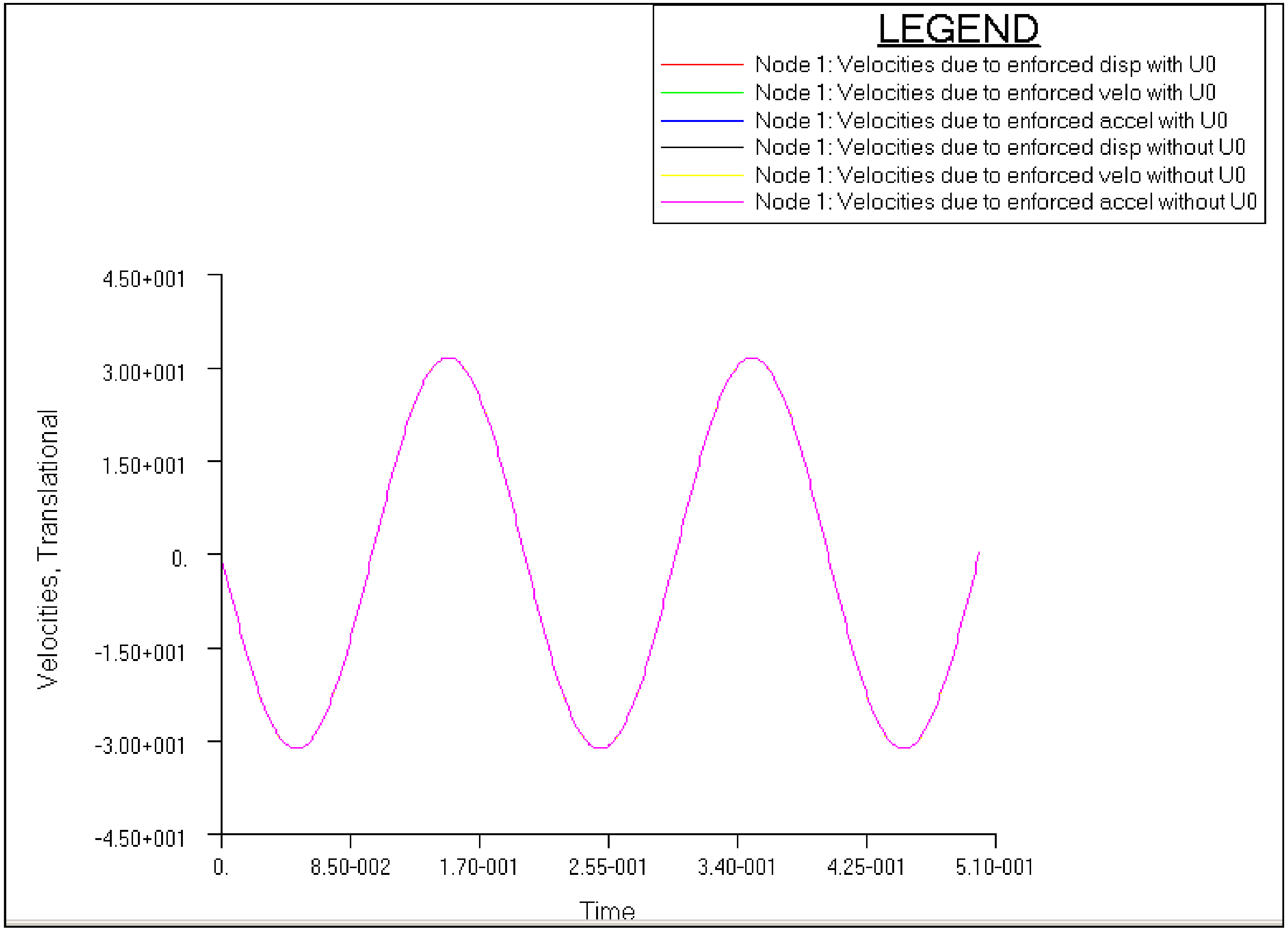
EXAMPLE SPECIFYING INITIAL DISPLACEMENT

- **Displacement output at grid 1**



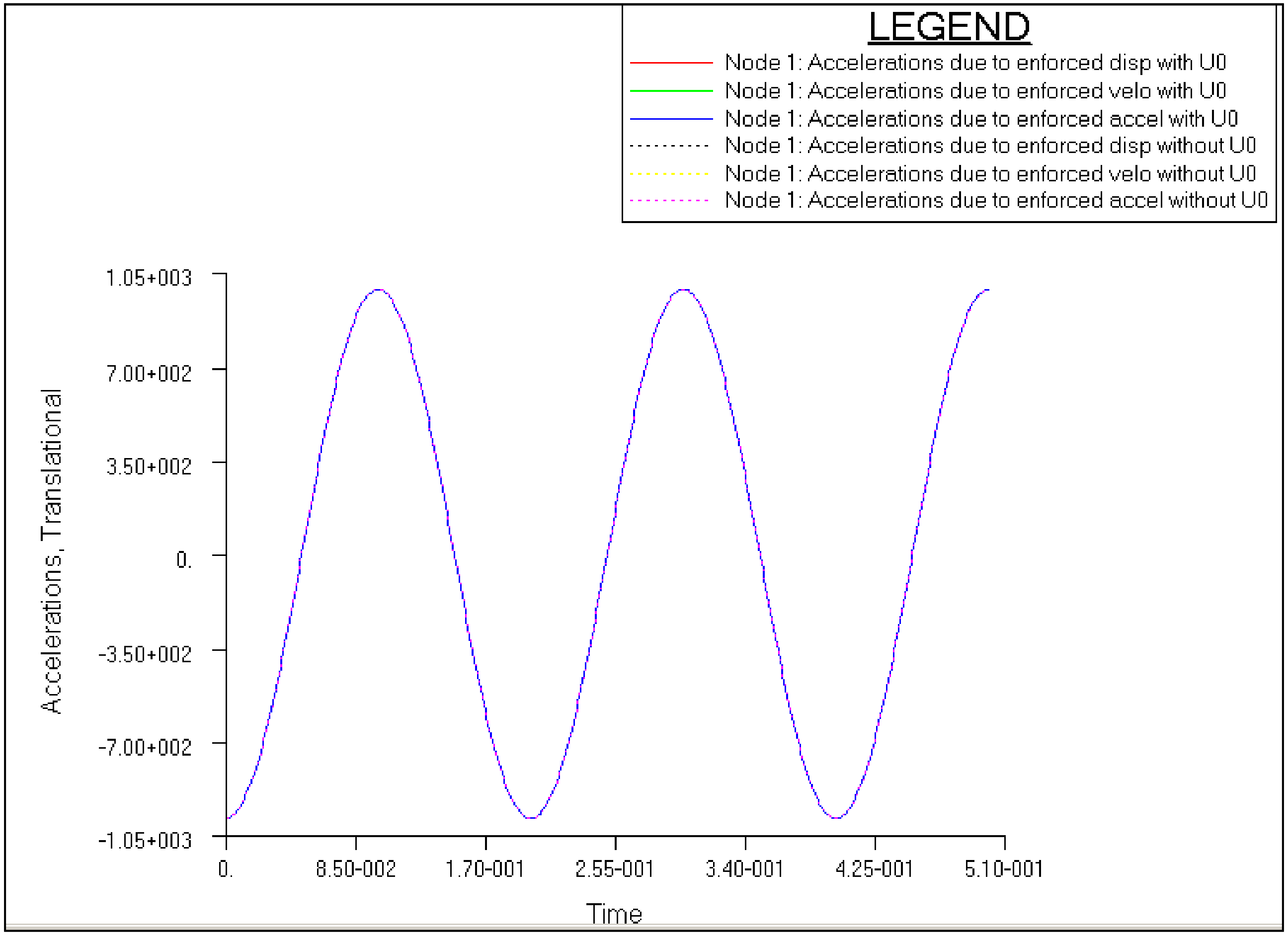
EXAMPLE SPECIFYING INITIAL DISPLACEMENT

- **Velocity output at grid 1**



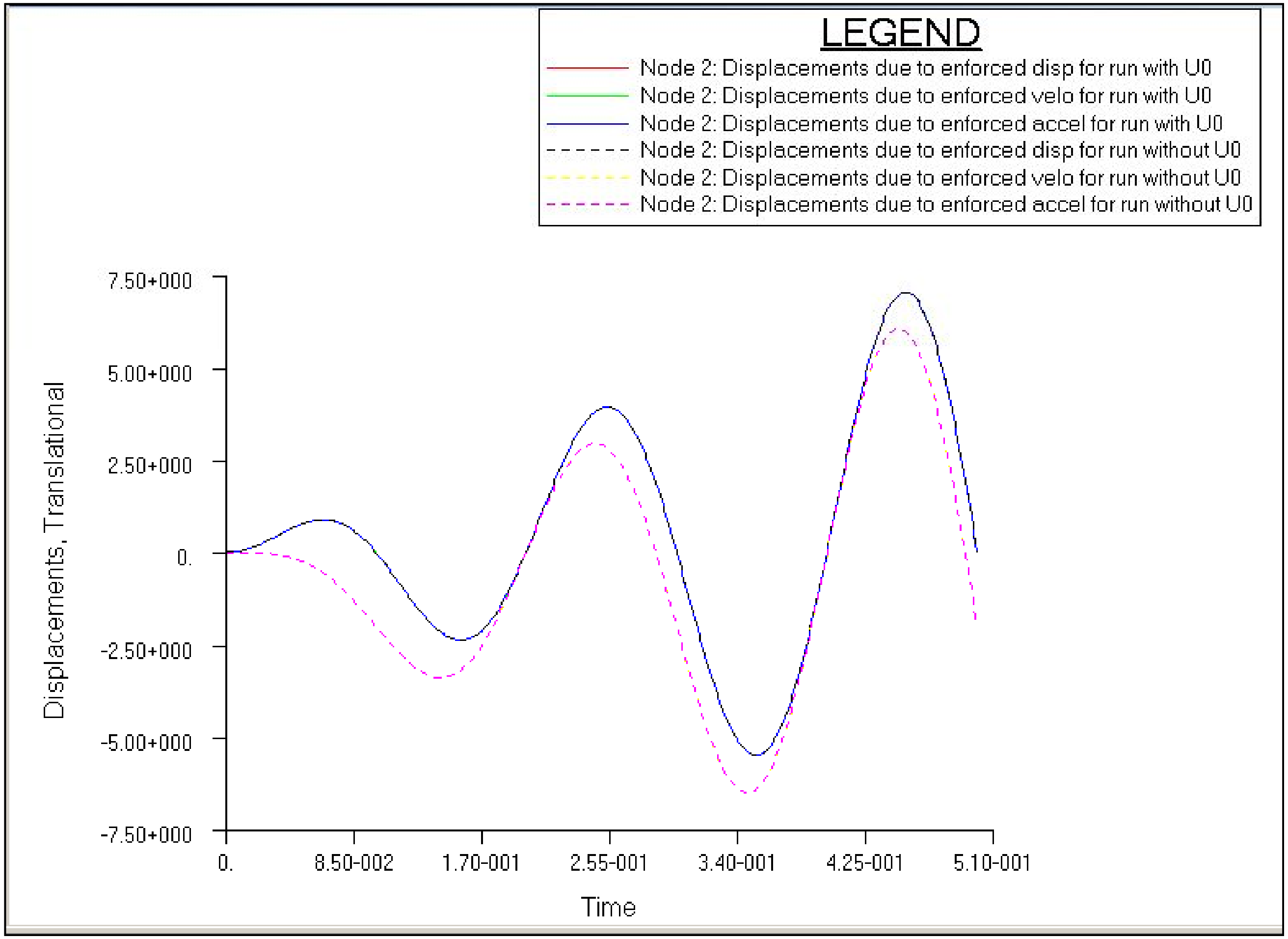
EXAMPLE SPECIFYING INITIAL DISPLACEMENT

- **Acceleration output at grid 1**



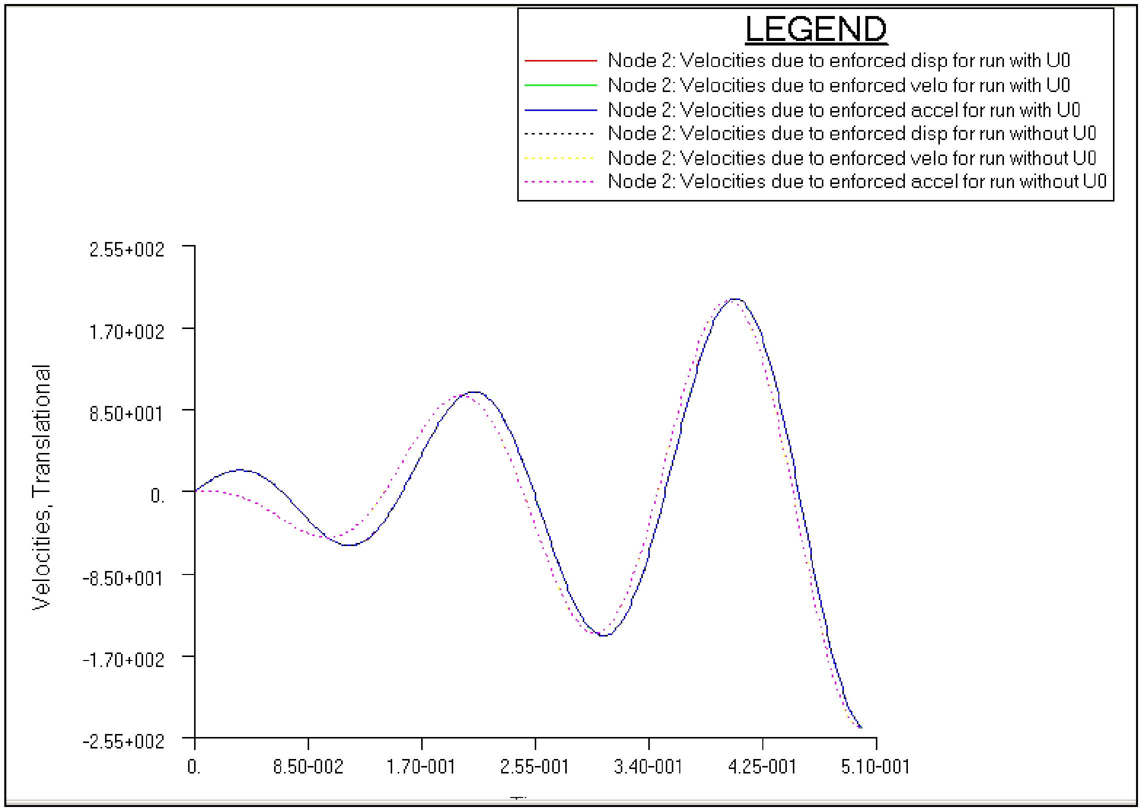
EXAMPLE SPECIFYING INITIAL DISPLACEMENT

- **Displacement output at grid 2**



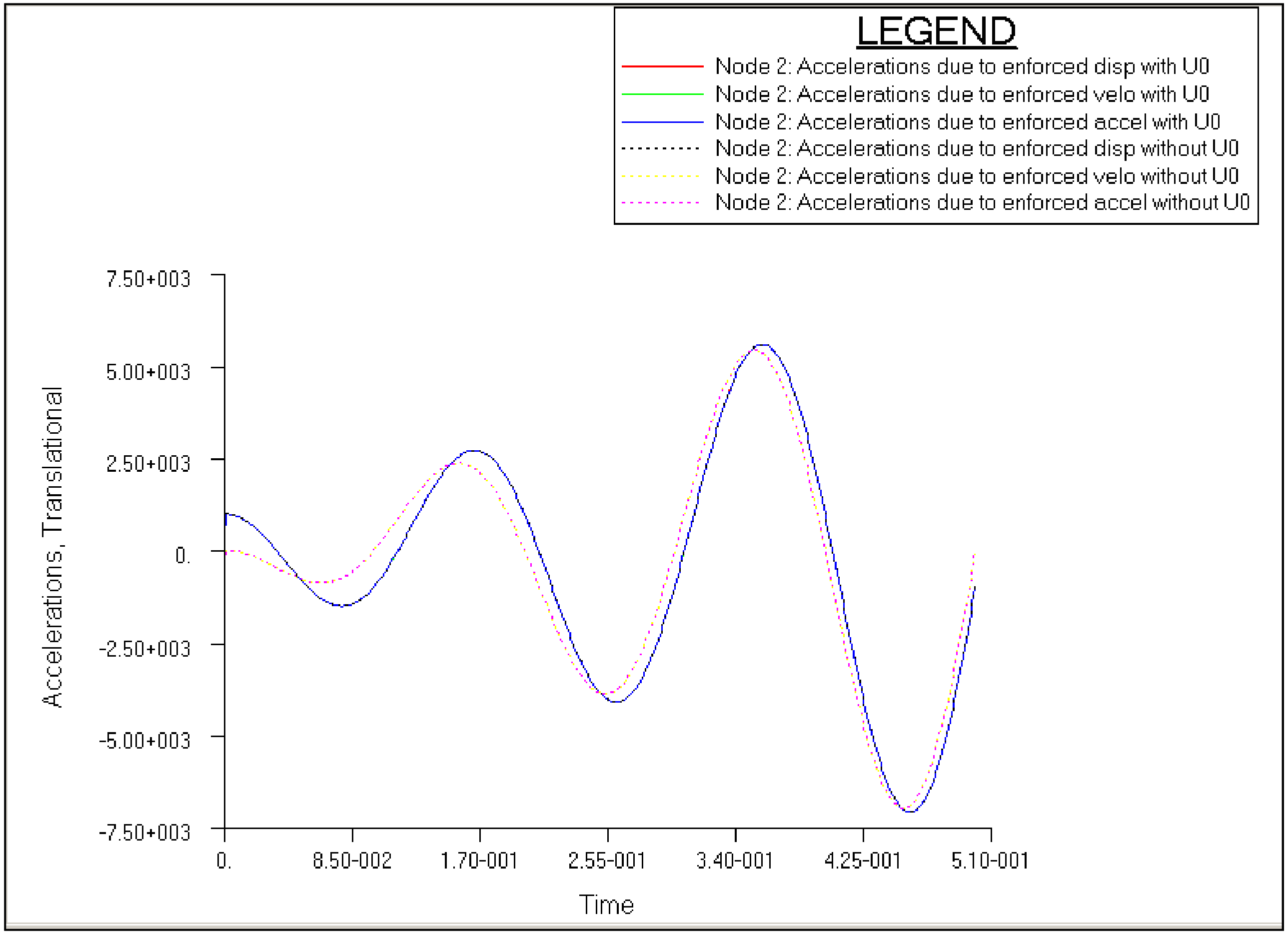
EXAMPLE SPECIFYING INITIAL DISPLACEMENT

- **Velocity output at grid 2**



EXAMPLE SPECIFYING INITIAL DISPLACEMENT

- **Acceleration output at grid 2**



EXERCISES

- **Now Perform the following workshops:**
  - Workshop 8A, Direct Transient Response with Enforced Acceleration
  - Workshop 8B, Modal Transient Response with Enforced Acceleration
  - Workshop 9A, Direct Frequency Response with Enforced Acceleration
  - Workshop 9B, Modal Frequency Response with Enforced Acceleration

SECTION 13

SHOCK AND RESPONSE SPECTRUM ANALYSIS

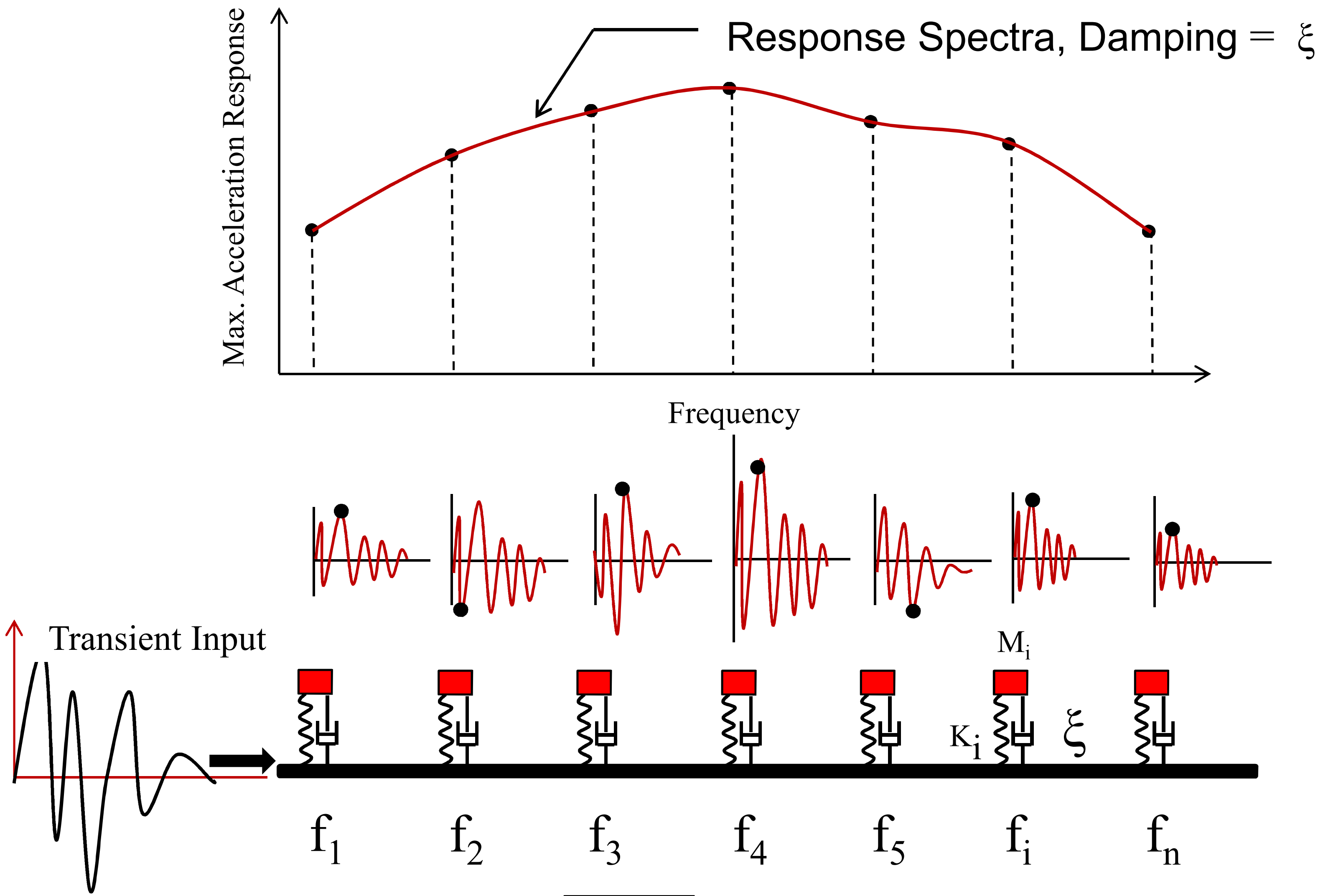
SHOCK SPECTRUM ANALYSIS

- What is Response (Shock) Spectra
- What is the difference between Harmonic and Shock Spectrum Input
- How to create Shock Spectra from acceleration transient input
- How to analyze structure subjected to shock input
- Various methods to combine modal response
- Comparison between transient and shock spectrum analysis.
- “Poor man’s” Transient Analysis
- Approximate method to predict the peak response of a structure
- Linear Analysis only
- Widely used in Seismic analysis of structures (building, nuclear power plants, civil engineering)

WHAT IS RESPONSE SPECTRA?

- **Response spectrum depicts the maximum response of an SDOF system as a function of its resonant frequency for base excitation**
- **The peak response of series of SDOF oscillators (each with different frequency, same damping) subjected to transient input**
- **This can then be repeated for a different damping**

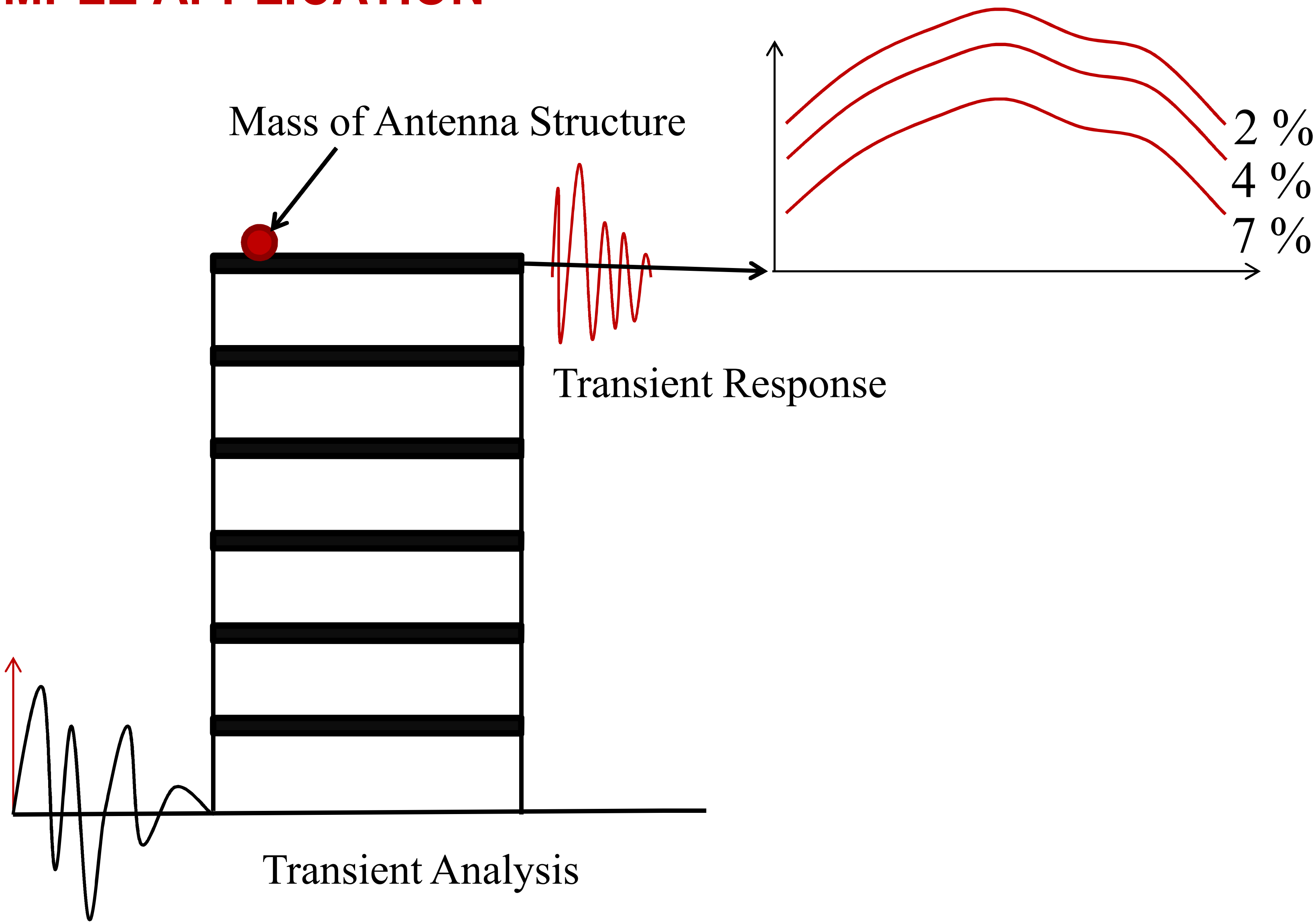
WHAT IS RESPONSE SPECTRA?



$$F_i = (1/2\pi)\sqrt{K_i/M_i}$$

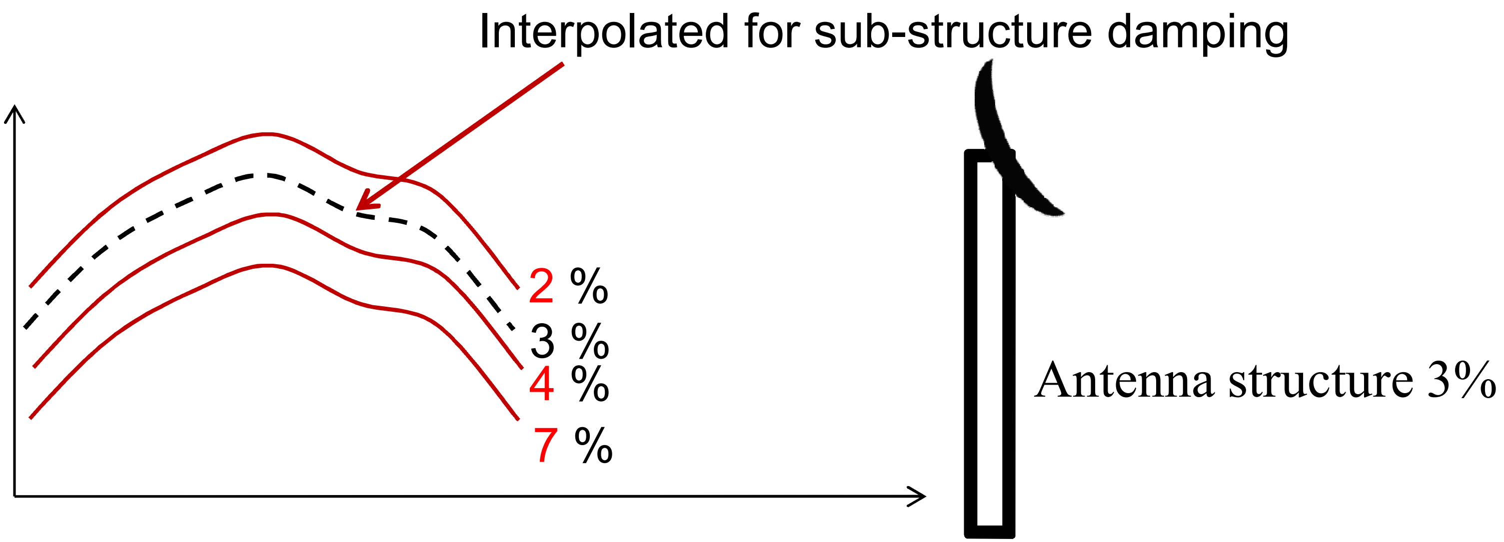
ξ = Constant Damping

SIMPLE APPLICATION



SIMPLE APPLICATION

- **Multiple transient analyses**
  - Multiple representative seismic motion
  - Input in different direction
- **Envelope all Spectra**
- **Apply enveloped spectra at base of small equipment**

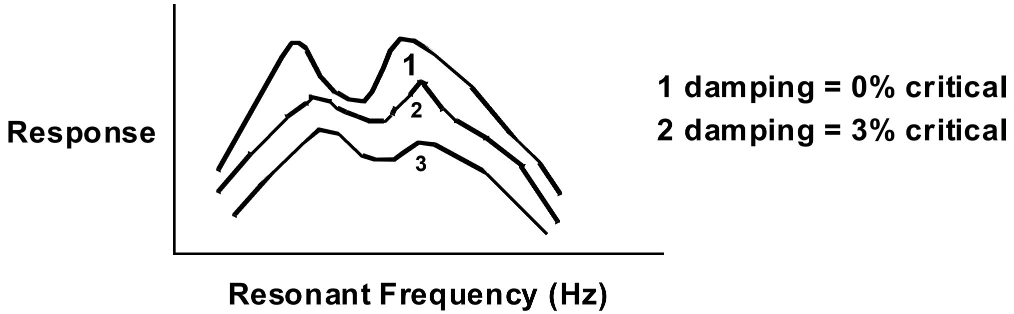


RESPONSE SPECTRUM

- **The peak response 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 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)**
- **<u>Example</u>:**
  - 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 (that is, machinery and piping systems).

RESPONSE SPECTRUM

- **Example (continued)**
  - Analysis is repeated for several damping values to generate a family of curves



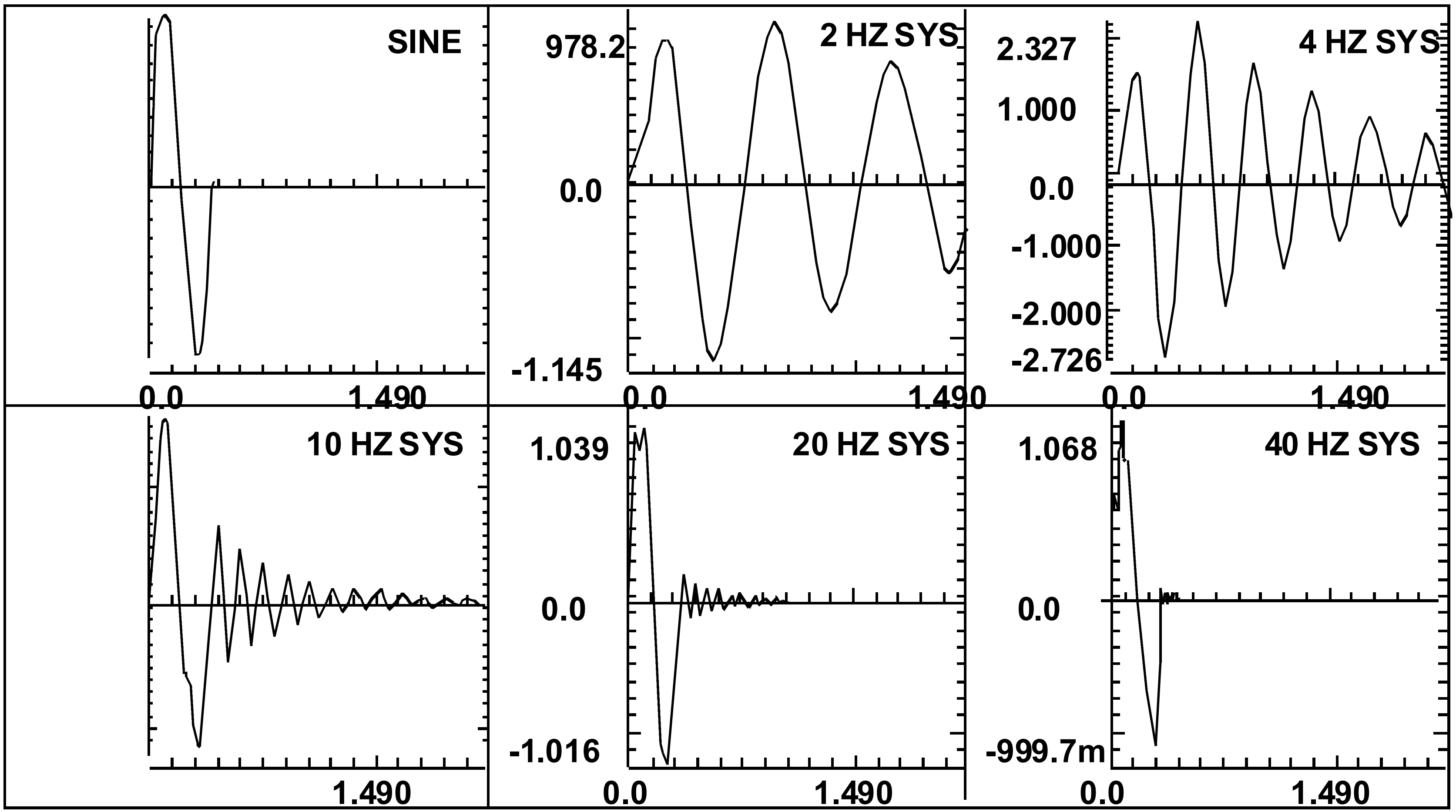
    - Damping applies to each oscillator, not to the vibrating structure
  - Maximum displacement response from X(t) is calculated for each oscillator. The maximum relative displacement between each oscillator and its base (a point on the vibrating structure) is also computed.
    - X = maximum intertial (absolute) displacement
    - X_r = maximum relative displacement
  - Relative velocity and absolute acceleration are approximately related to the relative displacements by

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

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

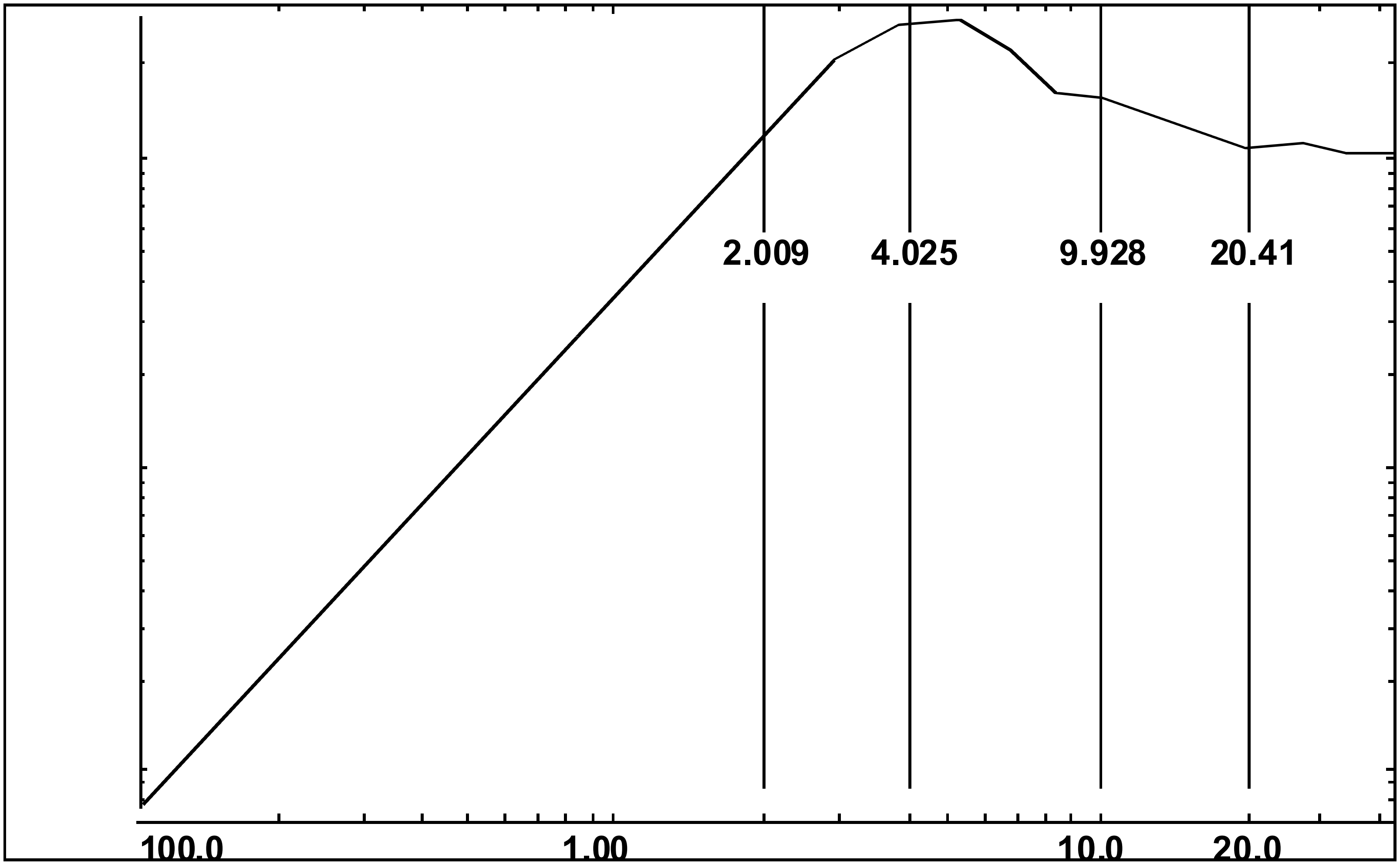
RESPONSE SPECTRUM

ZETA=.05 SDOF RESPONSE TO 4 HZ SINE PULSE



RESPONSE SPECTRUM

SHOCK SPECTRUM OF 4 HZ PULSE FOR ZETA = .05



RESPONSE SPECTRUM

- **For very low oscillator frequencies:**

$$X \rightarrow 0 \qquad \omega \rightarrow 0$$
$$X_r \rightarrow U_B$$

- **For very high oscillator frequencies:**

$$X \rightarrow U_B \qquad \omega \rightarrow \infty$$
$$X_r \rightarrow 0$$

- **The approximate relationships between $\dot{X}_r$, $\ddot{X}_r$, and X are not valid at very high or very low frequencies or for large damping values**
- **Note that only the magnitudes of response are computed with no phase information**

RESPONSE SPECTRUM

- **Response spectra may be generated in any transient solution (for example, SOLs 109, 112)**
- **The transient response for selected DOFs in the model is used as the input time history for the generation of the response spectra curves**
- **Further information on this subject is available in the *MSC.Nastran Dynamics User's Guide.***

RESPONSE SPECTRUM GENERATION

- **Required input**
  - Executive Control Section
    - SOL command selecting a transient solution (for example, SOL 109)
  - Case Control Section
    - XYPLOT SPECTRAL - Compute spectra
    - XYPUNCH SPECTRAL - Punch spectra

- **Example:**
  - XYPUNCH ACCELERATION SPECTRAL 1/1(T1RM)
  - This XYPLOT command creates a set of absolute (RM) acceleration spectra based on record number 1 on the DTI,SPSEL entry using the motion of Grid Point 1 in the x (T1) direction.

RESPONSE SPECTRUM GENERATION

- **Bulk Data Section**
  - PARAM,RSPECTRA,0 – Requests calculation of spectra
  - DTI,SPSEL – Correlates frequency and damping requests
  - FREQ – Used to specify frequencies and damping values (one FREQ set each)
- **Example Input:**

```
PARAM   RSPECTRA 0
$
$ Tells MSC.Nastran to perform spectra creation
$
$               RECNO  DAMP   FREQ   G1     G2     G3     G4
DTI     SPSEL   1      1      2      1      2      3      4      +SPSEL1
+SPSEL1 ENDREC
$
$ Tells the program that if RECNO 1 is selected by the XYPLOT command,
$ then DAMPing set 1 and FREQuency set 2 are to be used for GRIDs 1,
$ 2,3, and 4 (if requested)
$
FREQ    1       0.      .01     .02
$
$ This is the FREQ entry used by SPSEL RECNO 1 to specify damping ratios
$ to be used. In this case, spectral will be generated for damping
$ ratios of 0%, 1%, and 2% of critical
$
FREQ1   2       .5      .5      200
$
$ This is the FREQ entry used by SPSEL RECNO 1 to specify the
$ frequencies at which data points on the spectra will be
$ generated. In this case, spectra points will be generated
$ for oscillators with natural frequencies of .5Hz to 100.5Hz
```

APPLYING SPECTRA

- **Available in SOL 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 (the only problem is that the 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 model (including the 0.0 Hz modes) with the ‘SUPORT’ DOFs unconstrained
  - This approximates the “cantilevered” modes of the model attached to the “exciting” structure
  - 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:

$$\phi^T M D_m$$

    - Where: ϕ = the set of “elastic” modes

APPLYING SPECTRA

- **Procedure Continued**
  - 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), and the results are printed

APPLYING SPECTRA

- **Xr, 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:**

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

APPLYING SPECTRA

- **Absolute Option** $\bar{u}_k \cong \sum_i \sum_r |\phi_{ik}| |\Psi_{ir} \bar{X}_{ri}(\omega_i, g_i)|$
  - where
    $$\bar{X}_{ri}(\omega_i, g_i) = \max |X_{ri}(\omega_i, g_i, t)|$$
    - i represents a mode
    - r represents a direction

- **SRSS option** $\bar{u}_k \cong \sqrt{\sum_i (\phi_{ik} \bar{\xi}_i)^2}$
  - where the average peak modal magnitude, $\bar{\xi}_i$ is $\bar{\xi}_i \cong \sqrt{\sum_r (\Psi_{ir} \bar{X}_r(\omega_i, g_i))^2}$

- **NRL option** $\bar{u}_k \cong |\phi_{ik} \bar{\xi}_j| + \sqrt{\sum_{i \neq j} (\phi_{ik} \bar{\xi}_i)^2}$
  - Where $|\phi_{jk} \bar{\xi}_j|$ is the peak modal magnitude

APPLYING SPECTRA

- **Required Input**
  - Executive Control Section
    - SOL statement selecting SOL 103
  - Case Control Section
    - SDAMP To select modal damping ratios
    - DLOAD To select input spectra
    - METHOD To select eigenvalue solver
- **Example**
  - METHOD = 1
    - Selects eigenvalue solution method 1 from the Bulk Data (be sure that the range includes 0.0)
  - SDAMP = 1
    - Selects modal damping to be used for the calculated modes. Refers to a TABDMP1 entry in the Bulk Data
  - DLOAD = 1
    - Selects DLOAD Bulk Data entry that describes which spectra are applied at which 'SUPORT' DOF

APPLYING SPECTRA

- **Bulk Data**
  - PARAM,SCRSPEC,0 – Requests application of response spectra
  - DLOAD – Selects spectra and ‘SUPORT’ DOF at which to apply them
  - DTI,SPECSEL – Selects spectra, states associated damping and type of spectrum
  - TABLED1 – Provides input spectra
  - SUPORT – Selects spectrum input locations
  - TABDMP1 – Describes modal damping for the calculated modes
  - PARAM,OPTION – Selects modal combination method

APPLYING SPECTRA

- **Sample Input:**

```
SUPORT   1          3
$
$ Define input dof for the spectra - in this case, dof  3 for GRID 1
$ is selected
$
CONM2     1001      3       0         1000000.
$ apply large masses in the directions of the spectra input
TABDMP1  1       CRIT                                                    +DMP1
+DMP1    0.0     .01     100.    .01     100.01  .02     1000.   .02     +DMP2
+DMP2    ENDT
$
$ Select damping ratios for the calculated modes - in this case, a ratio of
$ 1% of ctrtical $ is used for all modes from 0hz to 100hz and 2% of critical
$ is used for all modes above $ 100.01hz
$
$
PARAM,SCRSPEC,0
$ Tells MSC.Nastran to perform shock spectrum analysis
$ DEFINE WHERE AND HOW TO APPLY SPECTRA
$
$ NOTE THAT SPECTRA ARE APPLIED USING INTERNAL SORT...NOT ASCENDING ORDER
$
$       SID     S       S1      L1      S2      L2 ....
DLOAD   1       1.0     1.0     1
$
$ Define where spectra are to be applied - this entry is called from
$ Case Control by a $ 'DLOAD=1' command - for this entry, an overall
$ scale factor of 1.0 (S) is applied, $ a factor of 1.0 (S1) is used
$ to apply spectrum 1 (L1) at 'SUPORT' dof 1.
$
$ (It should be noted that the order of the 'SUPORT' dof used on this
$ entry is the MSC.Nastran internal sort. If only one GRID point is used,
$ this is no problem, but if more than one GRID point is used,
$ then PARAM,USETPRT,1 should be used to obtain the internal order)
$
```

APPLYING SPECTRA

- **Sample Input Continued:**

```
DTI      SPECSEL 0
DTI      SPECSEL 1              A       2       0.0     3       .01     +SP1
+SP1     4       .02
$
$  This table defines the relationship between the input tables
$ (from the spectrum creation run) and the spectra sets. For example,
$ record 1 defines a spectra set representing acceleration spectra,
$ containing spectrum 2 for 0% of critical damping, spectrum 3 for 1% of
$ critical damping, and spectrum 4 for 2% of critical damping.
$ The program will interpolate between the spectra if a mode has a
$ damping value other than those defined in the table.
$
$                          GRID   Component (XYPLOT terminology)
$ACCE           0       1       3                                          1
$   0.000000E+00
TABLED1         2
         .5      3.156-4   1.0    .001263 1.5     .002842 2.      .005056
         2.5     .007905 3.      .011393 3.5     .015524 4.      .020303
         4.5     .025738 5.      .031839 5.5     .038615 6.      .046073
         6.5     .054219 7.      .063052 7.5     .072569 8.      .082766
         .
         .
         100.5   3.87229ENDT
$ Table representing the input spectra
```

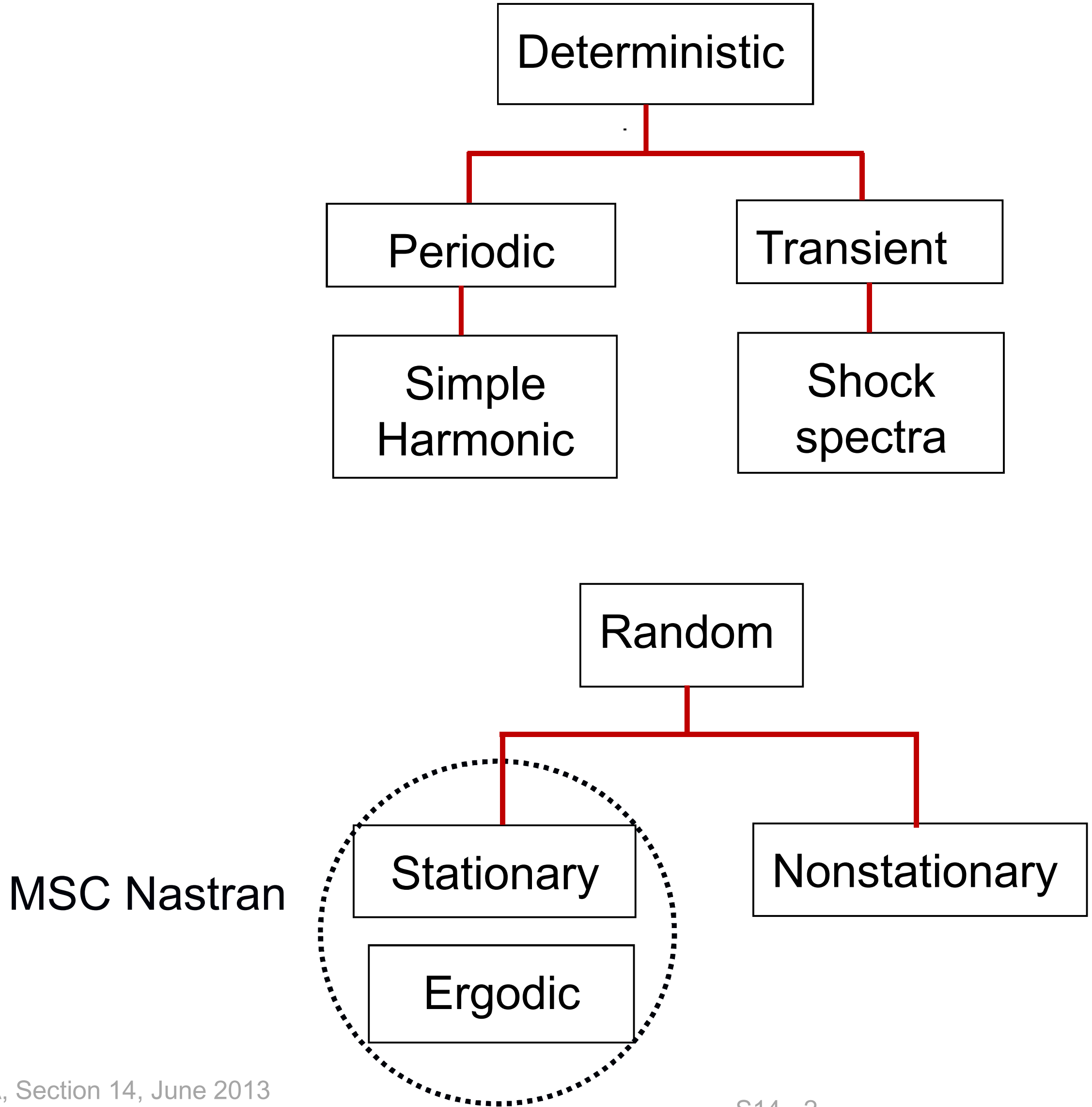
EXERCISES

- **Now Perform the following workshops:**
  - Workshop #10A, Generate Shock Spectrum Input
  - Workshop #10B, Apply Shock Spectrum Input

SECTION 14

RANDOM RESPONSE ANALYSIS

CLASSIFICATION OF DYNAMIC ENVIRONMENTS



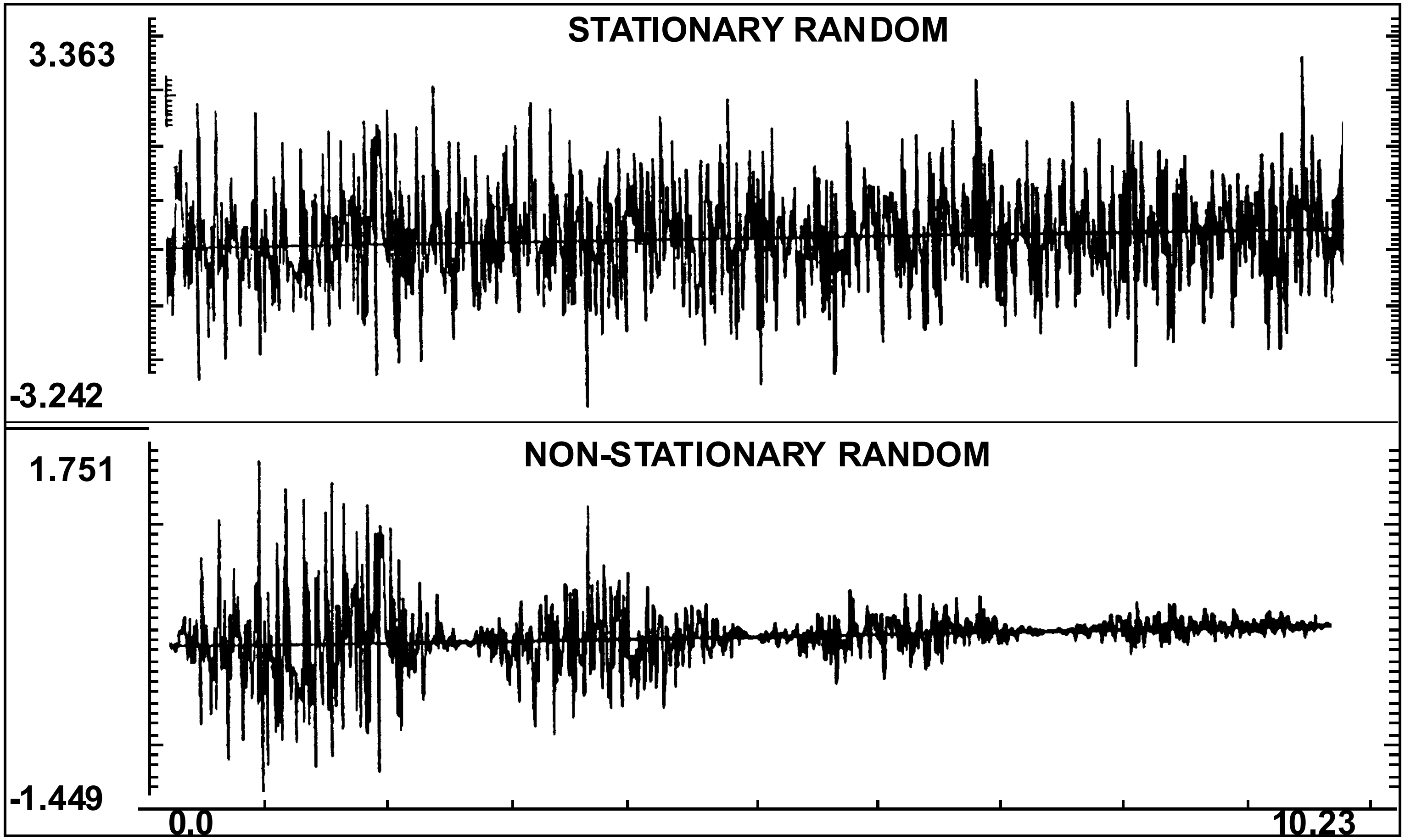
RANDOM RESPONSE ANALYSIS

- **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**
- **MSC Nastran performs random response analysis as post processing to frequency response. Inputs include the output from frequency response analysis as well as user-supplied loading conditions in the form of auto- and cross spectral densities. Outputs are response power spectral densities, autocorrelation functions, number of zero crossings with positive slope per unit time, and the RMS values of response.**
- **The theory is described in *Random Vibration in Mechanical Systems*, by S. H. Crandall and W. D. Mark, Academic Press, 1963**
- **Further information is available in the *MSC.Nastran Dynamics User's Guide***

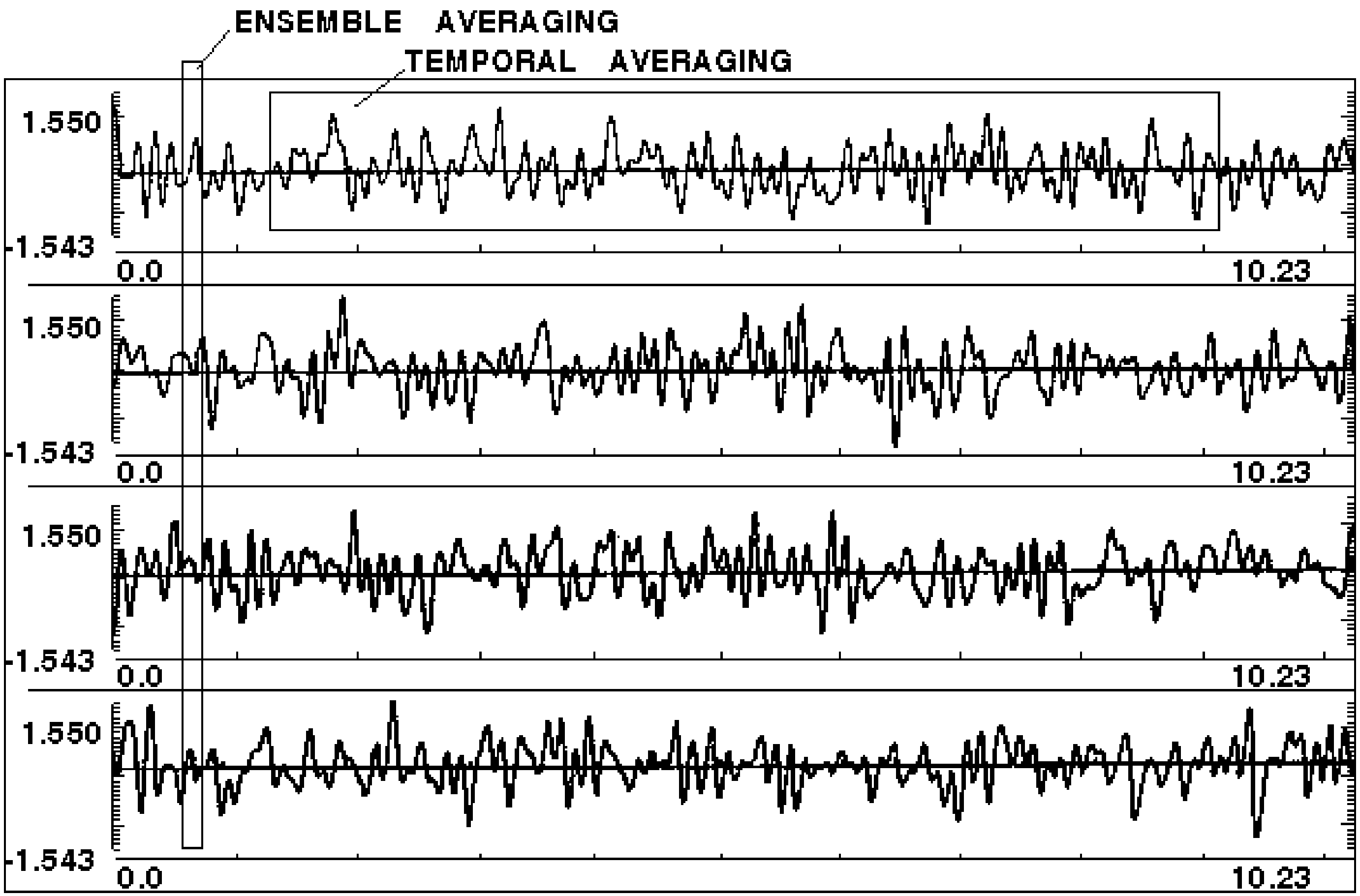
RANDOM ANALYSIS THEORY

- **There are several conventions used to define random analysis quantities. Care must be taken to use MSC Nastran random capability properly (see the *MSC.Nastran Advanced Dynamics User's Guide* for details and Bendat and Piersol (Reference 13) to understand the conventions).**
- **MSC Nastran random analysis assumes <u>ergodic processes</u>**
- **The concepts of autocorrelation, autospectrum (power spectrum), cross-correlation, and cross-spectrum must be defined**
- **The mean square value and apparent frequency are the key statistical quantities**

EXAMPLES OF RANDOM DYNAMIC ENVIRONMENT



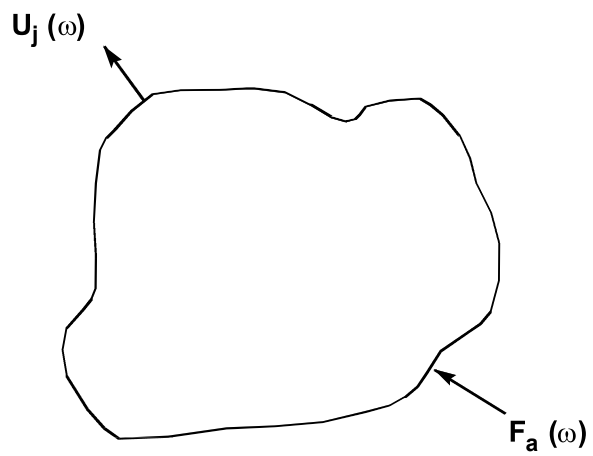
EXAMPLE OF ENSEMBLE OF ERGODIC RANDOM DATA



AUTOCORRELATION AND AUTOSPECTRUM

- **Autocorrelation function:** $$R_j(\tau) = \lim_{T \to \infty} \frac{1}{T} \int_0^T u_j(t) u_j(t - \tau) dt$$
  - Note: $R_j(o)$ is the mean-square value of $u_j(t)$
- **Autospectrum function:** $$s_j(\omega) = \lim_{T \to \infty} \frac{2}{T} \left| \int_0^T u_j(t) e^{-i\omega t} dt \right|^2$$
- **Fourier transform pairs:** $$R_j(t) = \frac{1}{2\pi} \int_0^\infty s_j(\omega) \cos(\omega\tau) d\omega$$
- **Mean square value:** $$\overline{u_t(t)^2} = R_j(o) = \frac{1}{2\pi} \int_0^\infty s_j(\omega) d\omega$$
- **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 we have several inputs, then:**

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

DEFINITION OF MULTIPLE INPUT-OUTPUT SPECTRAL RELATIONSHIP FOR A LINEAR SYSTEM

- **In matrix form we have:**

$$u_j(\omega) = \left[H_{ja}(\omega) H_{jb}(\omega) \cdots \right] \begin{Bmatrix} F_a(\omega) \\ F_{b(\omega)} \\ . \\ . \end{Bmatrix}$$

- **The output autospectrum is:**

$$S_{ujuj} = T\left[H_{ja} H_{jb} \cdots \right] \begin{Bmatrix} F_a(\omega) \\ F_b(\omega) \\ . \\ . \end{Bmatrix} \left[F_a^*(\omega) F_b^*(\omega) \cdots \right] \begin{Bmatrix} H_{ja}^* \\ Hjb^* \\ . \\ . \end{Bmatrix}$$

DEFINITION OF MULTIPLE INPUT-OUTPUT SPECTRAL RELATIONSHIP FOR A LINEAR SYSTEM

- **The individual input spectra are:**

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

$$T\,\overline{F_a(\omega)F_b^*(\omega)} = S_{ab}(\omega)$$

$$TF_b(\omega)F_b^*(\omega) = S_{bb}(\omega)$$

- **The multiple input-output spectral relationship is therefore:**

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

  - Where

$[H_j]^T = [H_{ja} H_{jb} \cdots]$ and $[H_j^*] = \begin{Bmatrix} H_{ja}^* \\ H_{jb}^* \\ \cdot \\ \cdot \\ \cdot \end{Bmatrix}$

DEFINITION OF MULTIPLE INPUT-OUTPUT SPECTRAL RELATIONSHIP FOR A LINEAR SYSTEM

- **The input cross-spectral matrix is:**

$$[S]_{in} \begin{pmatrix} S_{aa}(\omega) & S_{ab}(\omega) & \cdots \\ S_{ba}(\omega) & S_{bb}(\omega) & \cdots \\ . & . & \cdots \\ . & . & \cdots \end{pmatrix}$$

- **and it has the special properties:**

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

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

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

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

RANDOM ANALYSIS AS IMPLEMENTED IN MSC 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 $F(\omega) = 1.0$**

INPUT REQUIRED FOR RANDOM RESPONSE ANALYSIS

- **Executive Control Section**
  - SOL(required)

	Structured Solution
Direct	108
Modal	111

- **Case Control Section**
  - RANDOM – selects Bulk Data RANDPS, RANDT entries and entries for frequency response, and must be placed above the subcases
- **Bulk Data Section**
  - RANDPS – PSD specification
  - RANDT1 – autocorrelation time lag entries for frequency response

RANDPS ENTRY

RANDPS   Power Spectral Density Specification

Defines load set power spectral density factors for use in random analysis having the frequency dependent form

$$S_{jk}(F) = (X + iY)G(F)$$

Format:

1	2	3	4	5	6	7	8	9	10
RANDPS	SID	J	K	X	Y	TID			

Example:

RANDPS	5	3	7	2.0	2.5	4			

Field	Contents
SID	Random analysis set identification number. (Integer > 0)
J	Subcase identification number of the excited load set. (Integer > 0)
K	Subcase identification number of the applied load set. (Integer $\geq$ 0; K $\geq$ J)
X, Y	Components of the complex number. (Real)
TID	Identification number of a TABRNDi entry that defines $G(F)$. (Integer $\geq$ 0)

TABRND1 ENTRY

TABRND1 **Power Spectral Density Table**

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

Format:

1	2	3	4	5	6	7	8	9	10
TABRND1	TID	XAXIS	YAXIS						
	f1	g1	f2	g2	f3	g3	-etc.-		

Example:

TABRND1	3								
	2.5	.01057	2.6	.01362	ENDT				

Field	Contents
TID	Table identification number. (Integer > 0)
XAXIS	Specifies a linear or logarithmic interpolation for the x-axis. (Character: “LINEAR” or “LOG”; Default = “LINEAR”)
YAXIS	Specifies a linear or logarithmic interpolation for the y-axis. (Character: “LINEAR” or “LOG”; Default = “LINEAR”)
fi	Frequency value in cycles per unit time. (Real ≥ 0.0)
gi	Power spectral density. (Real)

RANDOM RESPONSE OUTPUT

- **<u>Output numerical values:</u>**
  - To output Auto-PSDF, auto correlation functions, and CRMS (cumulative root mean square) use the PRINT and RPUNCH options available from the following Case Control commands:
    - ACCELERATION
    - DISPLACEMENT
    - VELOCITY
    - FORCE
    - OLOAD
    - SPCF
    - STRESS
    - STRAIN
  - Log-Log option available when computing RMS, N0 from PSDF
    - Param,rmsint,log-log
  - Format for Displacement:

$$\textbf{DISPLACEMENT}\left[\left(\begin{bmatrix}\textbf{SORT1}\\\textbf{SORT2}\end{bmatrix},\begin{bmatrix}\textbf{PRINT, PUNCH}\\\textbf{PLOT}\end{bmatrix},\begin{bmatrix}\textbf{REAL or IMAG}\\\textbf{PHASE}\end{bmatrix}\begin{bmatrix}\textbf{PSDF, ATOC, CRMS}\\\textbf{RALL}\end{bmatrix},\begin{bmatrix}\textbf{PRINT}\\\textbf{NORPRINT}\end{bmatrix}, \textbf{RPUNCH}\right], \cdots\right)\right] = \begin{Bmatrix}\textbf{ALL}\\\textbf{n}\\\textbf{NONE}\end{Bmatrix}$$

- where
  - PSDF—request output for auto power spectral density function
  - ATOC—request output for auto correlation function
  - CRMS—request output for cumulative root mean square
  - RALL—request output for psdf, atoc, and crms
  - RPRINT—request printed output in the f06 file
  - RPUNCH—request punch output
  - NORPRINT—none of the above output

RANDOM RESPONSE OUTPUT

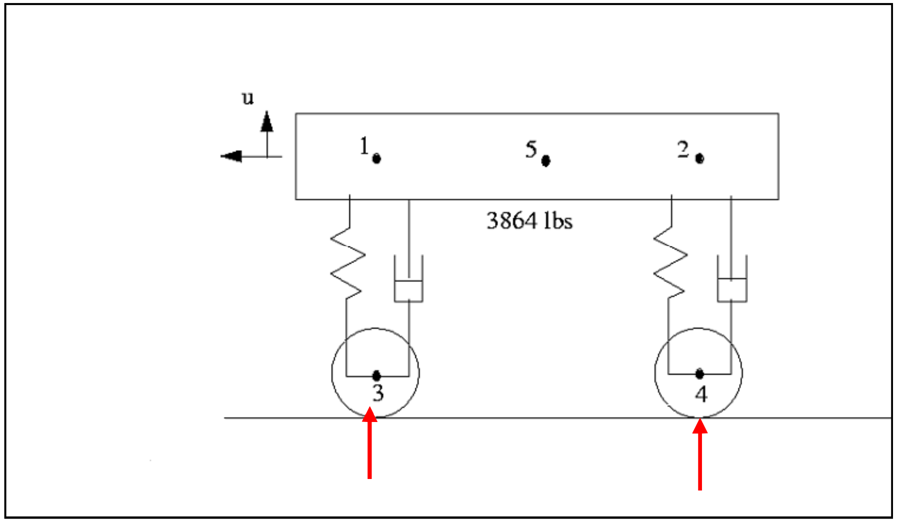
- **Output plots, x-y pairs, or maximum – minimum values**
  - To output Auto-PSDF, auto correlation functions use one of the following X-Yplot commands:
    - XYPLOT
    - XYPEAK
    - XYPUNCH
- **Additional Output:**
  - cross power spectral, cross-correlation functions
  - These output require RCROSS or RANDOM Case Control commands
  - Format for RCROSS case control command

$$\mathbf{RCROSS}\left[\left(\begin{bmatrix}\mathbf{REAL\ or\ IMAG}\\ \mathbf{PHASE}\end{bmatrix}\begin{bmatrix}\mathbf{PRINT}\\ \mathbf{NOPRINT}\end{bmatrix},[\mathbf{PUNCH}],[\mathbf{PSDF, CROF, RALL}]\right)\right]=\mathbf{n}$$

    - Where:
      - PSDF – request output for cross power spectral density function
      - CROF – request output for cross correlation function
      - RALL – request output for both psdf and crof
  - RCROSS and RANDPS bulk Data Entries

RANDOM RESPONSE ANALYSIS

- **Example:**
  - The following simplified car model is subjected to random loadings that are fully correlated at the front and back wheels



  - Output request:
    - The auto psdf disp, RMS, CRMS, N0 at grid points 1,2, and 5
    - The cross psdf displacement output between grid 1 (t2) and grid 2 (t2)

RANDOM RESPONSE ANALYSIS

- **Example continued:**

```
SOL 108 $
CEND
TITLE = SIMPLE CAR WITH RANDOM INPUT
SPC = 100
FREQUENCY = 1000
set 50 = 1,2,5
disp(phase,psdf,crms) = 50
rcross(phase,psdf) = 100
$
random = 1000
SUBCASE 1
  DLOAD = 111
SUBCASE 2
  DLOAD = 112
$
output (xyplot)
xtitle = frequency (hz)
ytitle = disp psd at grid pt 5
xypunch disp psdf /5(t2)
$
BEGIN BULK
```

```
$
$        2      3      4      5      6      7      8      9      10
RCROSS   100    DISP   1      2      DISP   2      2
$
FREQ1    1000   0.1    .05    40
$
$ DEFINE THE INPUT PSD
$        2      3      4      5      6      7      8      9      10
RANDPS   1000   1      1      1.     0.     145
RANDPS   1000   2      2      1.     0.     145
RANDPS   1000   1      2      1.     0.     146
RANDPS   1000   1      2      0.     1.     147
TABRND1  145
         .1     .1     5.     1.     10.    .05    ENDT
$
 .
 .
ENDDATA
```

RANDOM RESPONSE ANALYSIS

- **Example continued:**

```
0                                                                                                                  SUBCASE 1
1                                                                                                   MSC.NASTRAN

0                                                                                                                  RANDOM 1000
      POINT-ID =         1
                                                  D I S P L A C E M E N T   V E C T O R
                                                  ( POWER SPECTRAL DENSITY FUNCTION )

      FREQUENCY    TYPE          T1             T2              T3             R1             R2             R3
   1.000000E-01     G      0.0            1.039229E-01   0.0            0.0            0.0            1.415818E-07
   1.500000E-01     G      0.0            1.191483E-01   0.0            0.0            0.0            3.623494E-07
   2.000000E-01     G      0.0            1.385270E-01   0.0            0.0            0.0            7.176624E-07
                                                .
                                                .
   2.100000E+00     G      0.0            4.566946E-02   0.0            0.0            0.0            1.268936E-05

                                                .
                                                .
0                                                                                                                  RANDOM 1000
      POINT-ID =         5
                                                  D I S P L A C E M E N T   V E C T O R
                                                  ( POWER SPECTRAL DENSITY FUNCTION )

      FREQUENCY    TYPE          T1             T2              T3             R1             R2             R3
   1.000000E-01     G      0.0            1.034150E-01   0.0            0.0            0.0            1.415822E-07
   1.500000E-01     G      0.0            1.178516E-01   0.0            0.0            0.0            3.623497E-07
   2.000000E-01     G      0.0            1.359687E-01   0.0            0.0            0.0            7.176617E-07
                                                .
   2.100000E+00     G      0.0            5.515987E-05   0.0            0.0            0.0            1.268930E-05
```

RANDOM RESPONSE ANALYSIS

- **Example continued:**

0 RANDOM 1000

D I S P L A C E M E N T   V E C T O R
(ROOT MEAN SQUARE)

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	0.0	2.287596E+00	0.0	0.0	0.0	2.810828E-02
2	G	0.0	2.584790E+00	0.0	0.0	0.0	2.810428E-02
5	G	0.0	1.764859E+00	0.0	0.0	0.0	2.810628E-02

1 MSC.NASTRAN

0 RANDOM 1000

D I S P L A C E M E N T   V E C T O R
(NUMBER OF ZERO CROSSINGS)

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	0.0	8.958783E-01	0.0	0.0	0.0	1.057565E+00
2	G	0.0	8.972940E-01	0.0	0.0	0.0	1.057594E+00
5	G	0.0	7.188841E-01	0.0	0.0	0.0	1.057580E+00

1 MSC.NASTRAN

0 RANDOM 1000

.

0 RANDOM 1000

POINT-ID = 5

D I S P L A C E M E N T   V E C T O R
(CUMULATIVE ROOT MEAN SQUARE)

FREQUENCY	TYPE	T1	T2	T3	R1	R2	R3
1.000000E-01	G	0.0	0.0	0.0	0.0	0.0	0.0
1.500000E-01	G	0.0	7.437517E-02	0.0	0.0	0.0	1.122421E-04
2.000000E-01	G	0.0	1.089824E-01	0.0	0.0	0.0	1.989939E-04
2.500000E-01	G	0.0	1.387802E-01	0.0	0.0	0.0	2.981692E-04
3.000000E-01	G	0.0	1.673445E-01	0.0	0.0	0.0	4.141037E-04
3.500000E-01	G	0.0	1.964734E-01	0.0	0.0	0.0	5.496583E-04
			.				
			.				
2.100000E+00	G	0.0	1.764859E+00	0.0	0.0	0.0	2.810628E-02

RANDOM RESPONSE ANALYSIS

- **Example continued:**

```
0
1    SIMPLE CAR WITH RANDOM INPUT                                                          MSC.NASTRAN

0                                                                                                     RANDOM 1000
      SEQUENTIAL CURVE-ID =          1
                      C O M P L E X   C R O S S - P O W E R   S P E C T R A L   D E N S I T Y   F U N C T I O N
                                                           (MAGNITUDE/PHASE)
0          RCROSS              RTYPE1           ID1         COMP1            RTYPE2           ID2         COMP2          CURID
0           100                 DISP             1            2               DISP             2            2              0

           FREQUENCY                      CPSDF                         FREQUENCY                      CPSDF
          1.000000E-01       1.039372E-01 /   8.0841                  1.500000E-01       1.191402E-01 /  11.9813
          2.000000E-01       1.385522E-01 /  15.7046                  2.500000E-01       1.638877E-01 /  19.1527
          3.000000E-01       1.977741E-01 /  22.2584                  3.500000E-01       2.448712E-01 /  24.9524
          4.000000E-01       3.132123E-01 /  27.0288                  4.500000E-01       4.180073E-01 /  28.4139
                                          .
                                          .
          2.000000E+00       5.595233E-02 / 173.0778                  2.050000E+00       5.050250E-02 / 174.7362
          2.100000E+00       4.571699E-02 / 176.3955

0
0                                X Y - O U T P U T   S U M M A R Y  ( A U T O   O R   P S D F )
0 PLOT  CURVE FRAME    CURVE ID./       RMS       NO. POSITIVE  XMIN FOR   XMAX FOR   YMIN FOR    X FOR     YMAX FOR    X FOR*
  TYPE   TYPE   NO.   PANEL  : GRID ID   VALUE       CROSSINGS   ALL DATA   ALL DATA   ALL DATA    YMIN     ALL DATA    YMAX
0
  PSDF DISP       0         5(   4)    1.764859E+00  7.188841E-01 1.000E-01  2.100E+00  5.516E-05  2.100E+00  2.714E+01  7.000E-01
```

RANDOM ANALYSIS RECOMMENDATIONS

- **Most spectra are given as a log function. Use the log features on the TABRND1 entry if 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 summary output blindly.**
- **Use several frequencies in the vicinity of each mode. For the modal method, a combination of FREQ1 (or FREQ2) and FREQ4 usually works best.**
- **For low frequencies (<20 Hz), use many frequencies since the displacement spectra is changing rapidly for a constant input acceleration.**

EXERCISES

- **Now Perform the following workshops:**
  - Workshop #11, Random Response with Single Input
  - Workshop #12, Random Response with Multiple Inputs