

Dynamic Analysis User Guide

A user guide for FEMAP and NX Nastran Users



LS-DYNA Sales, Support & Consulting

www.PredictiveEngineering.com



Siemens PLM Software Sales & Support CAD I CAM I CAE I Teamcenter www.AppliedCAx.com



TABLE OF CONTENTS

1.	INTRO	DDUCTION TO LINEAR DYNAMICS (NX NASTRAN)	6
	1.1	some examples of linear vibration analysis (predictive)	7
	1.2	eigenvalue or normal modes analysis (gotta have mass)	8
2.	FOUN	IDATIONS OF FREQUENCY ANALYSIS	11
	2.1	baby's first beam model	11
		2.1.1 here's your modal analysis checklist	12
		2.1.2 setting up the model for normal modes with mass participation	13
		2.1.3 interpreting results based on orthogonality and mass participation	14
		2.1.4 symmetry and frequency analysis	16
		2.1.5 significance of strain energy for frequency analysis	17
3.	STAN	DARD NORMAL MODES ANALYSIS	18
	3.1	model setup	19
	3.2	natural frequency results and interpretation	20
		3.2.1 mass participation	22
4.	MOD	AL FREQUENCY ANALYSIS	24
	4.1	running a modal frequency analysis in femap and nx nastran	25
5.	INTRO	DDUCTION TO RANDOM VIBRATION	33
	5.1	the psd function	34
	5.2	the nx nastran method	36



	5.3	psd units	37
6.	EXAN	MPLE 1: PSD ANALYSIS OF PCB WITH TWO HEAVY ELECTRICAL COMPONENTS	
7.	EXAN	MPLE 2: CANTILEVER BEAM	
	7.1	problem definition	41
	7.2	analytical solution	42
	7.3	defining the system damping	43
	7.4	creating the psd function	44
	7.5	creating the modal frequency table/setting up the load set options for dynamic analysis	45
	7.6	loading & constraining the model	47
	7.7	specifying groups for nodal and elemental output	48
	7.8	creating an analysis set – simple psd	49
	7.9	interpreting the output	56
	7.10	positive crossings	57
	7.11	fatigue analysis using rms stress and positive crossings	59
	7.12	fatigue analysis – time to failure	60
8.	EXAN	MPLE 3: SOLID MESHED BEAM	61
	8.1	analytical solution	62
	8.2	psd function input	63
	8.3	psd stress results	64
	8.4	comparing miles' approximation and psd results	65
9.	DIRE	ECT TRANSIENT ANALYSIS	



10.	QUES	TIONS AND ANSWERS ABOUT FREQUENCY ANALYSIS	72
11.	BEING	G AN EXPERT: VIBRATION IS ABOUT MASS AND CONSTRAINTS	73
	11.1	check fo6 for mass summation and know what you know	73
	11.2	ground check if you are doing aerospace quality work	74
12.	RAND	OM VIBRATION CONCLUSION	75
13.	INTRO	DDUCTION TO RESPONSE SPECTRUM ANALYSIS	76
	13.1	the accelerogram	77
	13.2	creating a response spectrum	78
	13.3	nx nastran method	81
14.	RESPO	ONSE SPECTRUM ANALYSIS EXAMPLE: CANTILEVER BEAM	83
	14.1	problem definition	84
	14.2	analytical solution	85
	14.3	step 1: creating the fe model	88
	14.4	step 2: defining the response spectrum	89
	14.5	step 3: creating interpolation table	.90
	14.6	step 4: creating a modal damping function	91
	14.7	step 5 creating the excitation node	92
	14.8	step 6: constraining the model	93
	14.9	step 7: setting up the analysis	94
	14.10	post processing the results	101



	14.11	results comparison	102
15.	EXTRA	A CREDIT: SOLID MESHED BEAM	103
16.	APPE	NDIX	113
	16.1	flow chart from nx nastran theoretical manual	113
	16.2	creating modal frequency table	114
	16.3	autocorrelation function	117
	16.4	multiple excitation spectrums	120
	16.5	why we do a psd analysis	123



1. INTRODUCTION TO LINEAR DYNAMICS (NX NASTRAN)

Vibration analysis is a huge topic and is easily the second most common type of FEA analysis after the basic static stress analysis. Within the field of vibration analysis, the most common type of analysis is that based on the linear behavior of the structure or system during its operation. That is, its stress/strain response is linear and when a load is removed, the structure returns to its original position in a stress/strain free condition. Although this might sound a bit restrictive, it actually covers a huge swath of structures from automobiles, planes, ships, satellites, electrical circuit boards and consumer goods. If one needs to consider a nonlinear response of the structure during operation, there exist codes such as LS-DYNA that can solve for the complete nonlinear vibration response. But that is not simple or basic and is left for another seminar sometime in the future.

Vibration Rich Environment



Linear FEA Model (courtesy Predictive Engineering)



1.1 SOME EXAMPLES OF LINEAR VIBRATION ANALYSIS (PREDICTIVE)



1.2 EIGENVALUE OR NORMAL MODES ANALYSIS (GOTTA HAVE MASS)

When the structure can be considered linear and we are interested in its vibration response, NX Nastran provides a broad spectrum of analysis solution sequences to investigate its response. The starting point for all of this work is the EOM for the dynamic behavior of a structure:

Linear Dynamics: E.O.M.
$$m \frac{\partial^2 u}{\partial t^2} + c \frac{\partial u}{\partial t} + ku = r(t)$$

Eigenvalue problem: undamped free vibration: $m \frac{\partial^2 u}{\partial t^2} + ku = 0$
Assuming a solution of the form: $u = u_0 \sin \omega t$
Then: $\left[k - \omega^2 m\right] \left\{u_0\right\} = 0$
For non-trivial solutions (i.e., $u_0 \neq 0$):

$$\left[k - \omega^2 m\right] = 0$$
 Giving us the well-known frequency relationship: $\omega = \sqrt{\frac{k}{m}}$

Normal Modes / Eigenvalue problem: undamped free vibration



$$\omega = \sqrt{\frac{k}{m}} = \sqrt{\frac{23,000}{100}} = 15.16$$
 rad/sec

NX Nastran reports frequencies in cycles per second. Hence, 15.16 radians/sec is equal to 2.41 cycles/sec. This is a beautifully simple relationship but it assumes that the stiffness of your structure stays constant or does not change due to load application. From the normal modes analysis, one can determine the natural frequencies of the structure (ω) but also the form of its vibration response or its mode shape.

For now, here's a short list of what one can learn from a normal modes analysis:

The natural frequencies (since no load is applied, the response is "natural")

How the structure will deform at the natural frequencies but since there is no load, the mode shapes do not indicate the magnitude of the vibration response only its shape)

The amount of mass that is associated with that particular frequency

Strain energy plots to determine where the structure is flexing or straining the most at that particular frequency

Given that this seminar covers prior material, if these items don't immediately make sense to you, you'll find a more detailed explanation in my article "Linear Dynamics for Everyone.pdf".

Before we leave this subject, a static stress analysis is nothing more than the above equation with acceleration and velocity at zero and time = zero:



2. FOUNDATIONS OF FREQUENCY ANALYSIS

2.1 BABY'S FIRST BEAM MODEL

Normal modes only needs three material properties and some FEA lash up that will create a stiffness / mass relationship. A constraint set is optional.

Define Material - ISOTRO	OPIC		X				
ID 1	Aluminum Color	104 Palette	Layer 1 Type	Forsay with NX Nastan - JBuan Model modified - [Prediction] [Pia: 2005 Quentity Connect Model Migh Might]	the frejoweng Dit Delete Group Yee Window Holp Nedicties Ingineering		- 0 - 3
General Function Referen Stiffness Youngs Modulus, <u>E</u> Shear Modulus, <u>G</u> Poisson's Ratio, n <u>u</u> Thermal Expansion Coeff, <u>a</u> Conductivity, <u>k</u>	Inces Nonlinear Ply/Bond Failure 10300000. 0. 0. 0.33 0. 0. 0. 0.	Creep Electrical/O Limit Stress Tension Compression Shear Mass Density Damping 2C/Co.	ptical Phase 0. 0. 0. 0. 2.64243E-4 0.	The their Generation of Lorent Mark Multip and at 24 = 5 = 24 = 24 at 14 at 24 = 5 = 24 at 14 at 24 at 24 at 24 at 24 at 14 at 24 at 24 at 24 at 24 at 14 at 24 at 24 at 24 at 24 at 24 at 14 at 24 a	10 Delle Delle Delle Verder på Rudske bigvenny i 		Branches Branches Branches (1997)
Specific <u>H</u> eat, Cp Heat Generation Facto	0. r 0.	Reference Temp	0.	19 06-01-00 ° g trading g Publication Integrate Ready-Tacket 11, Damanes 10	The second secon	мł	Page Life Ges Ges Oct
fw Load	Save Co	рру	QK Cancel				

Elastic Properties and Mass Density (Snails)



2.1.1 HERE'S YOUR MODAL ANALYSIS CHECKLIST

- o Elastic and mass properties are in consistent units
 - The weight of your structure can be checked by summing the mass of the model and multiplying it by gravity (for US units of lbf, inch and seconds, it would be 386 in/s2)
- o FEA model with a sufficient mesh density to capture the frequencies of interest (see below)
- o Constraint set that reflects reality as close as one can with a numerical simulation



Twelve Element Mesh (Fine)







2.1.2 SETTING UP THE MODEL FOR NORMAL MODES WITH MASS PARTICIPATION

Title NV Nastran Modes Analysis Set		Skip EIGx		Method ID	1
Litte INX Nasu all Modes Analysis Sec		Real Solution Methods	Range of Interest		
Analysis Program 36 NX Nastran	T	Lanczos		Real	Imaginar
Analysis Frogram Sound Husball		<u>Auto (HOU/MHOU)</u>	From (11a)	0	0
Analysis Type 2. Normal Modes/Fig	envalue 🔻	© Subspace	From (Hz)	0	0.
		Legacy Real Solution Methods	<u>⊥</u> o (Hz)	0.	0.
Run Analysis Using	g VisQ	© <u>G</u> ivens	Eigenvalues and E	igenvectors	
		Modified Givens	Number Estimate	d	0
Next OK	Cancel	Inverse Power	Number Esumate		
		Inverse Power/Sturm	Number Desired		10
		O Householder	Name limbing Mai		Marra
		Modified Householder	Normalization Me	thod	Mass
		Complex Solution Methods	Mass Node	e ID 0	 Derault
sis Set Manager (Active: 1NX Nastran Modes An	nalysi	Hessenberg	O Max DOE	0	C Lumped
		Complex Inverse Power	O Point		Coupled
/sis Set : 1NX Nastran Modes Analysis Set	Analyze	Complex Lanczos	Complex Solution	Options	
/pe : Normal Modes/Figenvalue	Analyza Multipla		Convergence		0.
alyze : Local	Analyze Huldpie	Olivert	Region Width		0.
tions	Export	Modal	Oursell Demoine		0
-xecutive/Solution Bulk Data		U HOUAI			0.
GEOMCHECK	Active	Prev Ne <u>x</u> t		OK	Cancel
MODELCHECK	Preview Input				
Modal/Buckling		1			
Rotor Dynamics	MultiSet	NASTRAN XY Outp	ut for Modal Ar	nalysis	X
Modal XYPlot	Copy		al failurates		
- Not Defined		Output Requests			
Response Spectrum Application	Delete	output nequests			
Response Spectrum Application External Superelement Reference					
Response Spectrum Application External Superelement Reference ster Requests and Conditions		Summary			
Response Spectrum Application External Superelement Reference ster Requests and Conditions Cases Defined		Modal Partici	nation Factors		
External Superlearner Reference ster Requests and Conditions Cases Defined	Save	Modal Partici	oation Factors		
Response Spectrum Application External Superelement Reference ster Requests and Conditions Cases Defined	Save	Summary Summary Modal Partici Modal Effecti	oation Factors ve <u>M</u> ass		
Response Spectrum Application External Superelement Reference ster Requests and Conditions Cases Defined	Save	Summary Summary Modal Partici Modal Effecti Modal Effecti	pation Factors ve <u>M</u> ass ve <u>W</u> eight		
Response Spectrum Application External Superelement Reference ster Requests and Conditions Cases Defined	Save	Summary Summary Modal Partici Modal Effecti Modal Effecti Modal Effecti Modal Effecti	pation Factors ve <u>M</u> ass ve <u>W</u> eight ve Mass Fraction		
Response Spectrum Application External Superelement Reference ster Requests and Conditions (Cases Defined	Save Save New Edit	Summary Summary Modal Partici Modal Effecti Modal Effecti Modal Effecti	pation Factors ve <u>M</u> ass ve <u>W</u> eight ve Mass <u>F</u> raction		
Lesponse Spectrum Application External Superelement Reference ter Requests and Conditions Cases Defined	Save New Edit Done	Summary Summary Modal Partici Modal Effecti Modal Effecti Modal Effecti Reference Node	oation Factors ve <u>M</u> ass ve <u>W</u> eight ve Mass <u>Fraction</u> 0		

2.1.3 INTERPRETING RESULTS BASED ON ORTHOGONALITY AND MASS PARTICIPATION

Cylindrical structures will have orthogonal modes that indicate that the structure actually has an infinite number of mode shapes at that frequency. But if you ain't using "rods" – you'll never see this in your analysis.

AppliedCAx.com / Predictive Engineering - Please share with your friends and visit us online at www.AppliedCAx.com



Output Set: Mode 2, 386.7562 Hz Deformed(62.09): Total Translation

Second Frequency 386.8 Hz



Mass participation tells you which modes have the "umph" and how many frequencies you need (modes) to accurately capture the dynamic response of the structure. On this later subject, a modal frequency analysis (e.g., PSD) formulates its response based on the number of modes chosen for the analysis. To ensure that you have captured the dynamic response of the structure, you'll want to use enough modes that you have at least 90% of the mass of the structure covered. What does this mean? Take a look at this screen shot showing the Mass Participation versus Number of Modes for the simple rod model. The bending modes capture 90% of the mass after 6 modes while to get the axial mass, it takes 26 modes.





2.1.4 SYMMETRY AND FREQUENCY ANALYSIS

This is just a little note to remind everyone that you can rarely use symmetry in a frequency analysis since the mode shapes are rarely symmetric. It sounds off but the higher frequency mode shapes are not symmetric. One might be able to use symmetry if you are only interested in the most basic mode shape.





2.1.5 SIGNIFICANCE OF STRAIN ENERGY FOR FREQUENCY ANALYSIS

If one wants to move your natural frequencies up or down, sometimes intuition is good enough but it never hurts to have a quantitative tool. When a structure flexes or vibrates, there will be regions within the structure that are deforming more and other regions less. Since a natural frequency analysis provides you with the mode shape (dimensionless deformation); it can also easily provide you with a contour plot of the relative strains within that structure. It sounds simple but can be tricky. Just to make sure that we understand this concept, we'll use a very simple model to explain this concept.

A center strip of the model has been thinned. This allows us to clearly see the effect of how strain energy plots can show us how to modify the structure to increase or decrease its natural frequencies.





3. STANDARD NORMAL MODES ANALYSIS

To see how this is applied in practice, we will run through an analysis project from start to finish (Normal Modes, Modal Frequency, PSD and Direct Transient). The model has been tweaked to protect the innocent.

We are starting with a PCB with two heavy electrical components. The PCB is a plate structure and the electrical components are modeled with solid elements. The PCB is screwed into a heavy component at the ends. The client must demonstrate that their PCB component can survive GM's vibration, PSD and Direct Transient (pothole) specifications (but that has been modified to confuse any automotive spies).



FEMAP Model

Units: N, mm, Tonne, s

3.1 MODEL SETUP

Since we know in advance that we will be doing more advanced frequency analyses, we can set up the constraints such that we don't have to mess with them in downstream analyses. The RBE2 element is setup to mimic a pinned connection at each of the PCB mounting holes. This is done by releasing the dependent DOF's of the RBE2. If you are not up-to-speed on multi-point-constraint (MPC) theory, take a look at our Seminar "Connections 2013: RBE2, RBE3 and CBUSH Elements".



3.2

NATURAL FREQUENCY RESULTS AND INTERPRETATION

A normal modes/eigenvalue analysis is the starting point for all linear dynamics work. It is simple to setup but difficult to interpret the results.

			Tit	le XXX-0814-0	1 Normal M	odes Analysis						
			An	alysis Program	36NX N	astran	•					
			An	alysis Type	2Norma	I Modes/Eigenvalu	Je 👻					
					Run Ar	alvsis Using VisO						
			_			(interesting the second						
				Next	Oł	Car	ncel					
undary Conditions		X	NASTRAN Modal Analysis	e (aniliana		X	Nastran Output Regu	iests	-	the Auditoria		X
Primary Sets			Skip EIGx		Method ID	1	Nodal			Elemental		
Constraints	1Universal All-Purpose	Con: 🔻	Real Solution Methods	Range of Interest		Texandianeu	✓ Displacement	0Full Model	-	Force	0Full Model	
Loads	0None	•	Auto (HOU/MHOU)		roeal	Imaginary	Applied Load	0Full Model	-	Stress	0Full Model	
Temperatures	0From Load Set		🔘 Subspace	From (Hz)).	0.		0Full Model	•	Strain	0Full Model	
Initial Conditions	0None	-	Cegacy Real Solution Methods	10 (HZ)				0Full Model	-	Chaile Second	0. Full Model	
Constraint Equations	0From Constraint Set	-	Modified Givens	Eigenvalues and Eige	nvectors		Equation Force	0. Full Model	-	Strain Energy	0 Full Model	
Bolt Preloads	0From Load Set	-	Inverse Power Inverse Power/Sturm	Number Desired		10	Force Balance	0. Full Model	-	Heat Hux	0. Full Model	
			C Householder	Name limbles Mathews			Velocity	0 Full Model		Enthalpy	0 Full Model	
Other DOF Sets Master (ASET)	0 None	-	Modified Householder	Mass Made to		Default	Acceleration	0. Full Medel	•	Enthalpy Rate	0. Full Medel	
Visconatic (CURORT)	0. None	_	Complex Solution Methods	Max		C Lumped	Kinetic Energy	0Full Model		Temperature	0Full Model	-
Kinematic (SUPORT)	UNone	-	Complex Inverse Power	© Point		Coupled	Temperature	0Full Model	*	Kinetic Energy	0Full Model	*
SUPORT1	0None	-	Complex Lanczos	Complex Solution Opt	tions					Energy Loss	0Full Model	*
TIMC	0None	-	Solution Type	Convergence		0.				Fluid Pressure	0Full Model	Ŧ
QSET	0None	-	O Direct Modal	Overall Damping (G)		0.	Customization					Danas
CSET	0None	-		oricran ban ping (a)			Element Corner R	esults	Re	esults Destination		rev
	0 None	-	Prev Next		ОК	Cancel	Output Modes (a,b,c	THRU d)		2PostProcess Only	· _	
BSET	Mar Mulle											01/



The mode shapes indicate the shape of that particular natural frequency. Since we are solving the EOM that has no {Force} or {Load}, the mode shapes have an arbitrary magnitude but they do tell us something very important. For example, the first mode flexes in the Y-direction and if excited in that direction, the structure would have a very strong response.



3.2.1 MASS PARTICIPATION

As engineers, we like to quantify our work and just to say it has a "strong response" is not exactly very qualitative. To remove some of this subjectiveness, it is useful to ask the model how much mass is associated with each natural frequency. That is, each natural frequency moves or captures a certain mass percentage of the structure. Its total dynamic response is the summation of all its natural frequencies (which can be a lot or just a few depending on the structure).

III Analysis Set Manager (Active: 1..XXX-0814-01 Normal ... - Analysis Set : 1..XXX-0814-01 Normal Modes Analysis Analyze Solver : NX Nastran Type : Normal Modes/Eigenvalue Analyze Multiple... Analyze : Local - Options Executive/Solution Export Bulk Data Active.. GEOMCHECK MODELCHECK Preview Input Hodal/Buckling DDAM Rotor Dynamics MultiSet... - Modal XYPlot Not Define Сору Response Spectrum Application Delete External Superelement Reference Master Requests and Conditions Title : Untitled Load ... External Superelement Creation Boundary Conditions Save... - Output Requests -No Cases Defined Analysis Set : 2..XXX-0814-01 Y-Axis PSD Analysis New. Analysis Set : 3..XXX-0814-01 Y-Axis Transient Shock (Potho Edit .. Done

With the "Not Defined" item highlighted, hit the edit button and the NASTRAN XY Output for Modal Analysis dialog box will appear. This box requests the mass participation factors.

utput Requests	
Summary	
🗸 Modal Participat	tion Factors
Modal Effective	Mass
Modal Effective	Weight
Modal Effective	Mass Fraction
Reference Node	0

AppliedCAx.com / Predictive Engineering - Please share with your friends and visit us online at www.AppliedCAx.com

Analysis Set Manager/Normal Modes

Once the mass participation items have been requested, the results are output as functions. I like to plot the SUM functions in the T1, T2 and T3 directions. As can be seen, the first natural frequency captures 90% of the mass of the structure in the T2 direction (Y-direction) and would be scary if excited.





4. MODAL FREQUENCY ANALYSIS

What does it mean to have mass and shape? It means that if your vibratory load is aligned in that direction and near that frequency, you have the perfect storm.

A modal frequency analysis is driven by a sinusoidal varying load. Its EOM is given as:

$$F_o \sin(\omega t - \theta) = m \frac{\partial^2 u}{\partial t^2} + c \frac{\partial u}{\partial t} + ku$$

And since it has a force, we get displacements and stresses from a model; however there is a hitch, results from this type of analysis are given in the form of magnitudes and phase angles. For example, displacement at any node is given as u_0 and Θ , and when requested, FEMAP can calculate the time varying response at any solved frequency (ω) as:

$$u = u_o \sin(\omega t - \theta)$$

Thus, a modal frequency analysis assumes that the forcing function is sinusoidal and solves the EOM in the frequency domain with results kicked-out in the form of absolute magnitudes and phase analysis. This makes interpretation of the results somewhat challenging and requires a bit of understanding of how the sinusoidal varying load is interacting with the mode shapes within each frequency.



2019

4.1 RUNNING A MODAL FREQUENCY ANALYSIS IN FEMAP AND NX NASTRAN

Title XXX-0814	I-01 Modal Frequency Analysis
Analysis <u>P</u> rogram	36NX Nastran
<u>A</u> nalysis Type	4Frequency/Harmonic Respon: 🔻
	Run Analysis Using VisQ

We'll Start with this option and explore what happens when you hit this circuit board with a sinusoidal varying 1 g acceleration in the Y-direction. Since we know from our junior level mechanical engineering vibration class that if we don't apply a bit of damping to the analysis the response goes to near infinity; hence we'll use the engineer's standard of 2% critical damping

For this analysis, we'll create the critical damping function and let the program determine the solution frequencies.

ID 3	equency Critical Damping		Type 7Critical Damp vs. Freq -		
X -Frequency Y -Frac Crit			X Axis Log Scale	Y Axis Log Scale	
0.02	04 - 035 - 03 - 025 - 02 - 015 -				
	.01 - .005 - 0. 10. 3341.66	7 6673.333	10005. 13336.67	16668.33 20000. Frequency	
Data Entry ● Single Value ── Edit Phase (X)	0	7 6673.333	1 10005 13336.67	16668.33 20000. Frequency Get Data Series Data	
Data Entry Single Value C Edit Phase (X) Linear Bamp Edit Maggitude) (Y) Delta X 1	7 6673.333	10005. 13336.67 Copy Function Load from Library	16668.33 20000 Frequency Get Data Series Data Paste from Clipboard	
Data Entry ③ Single Yalue ② Edit Phase (X ; ③ Linear gamp ③ Edit Maggtude ③ Eguetion ③ Periodic X 10. Y 9.02	01 - 005 - 10. 3341.66 (Y) Delta × 1 × Variație x	7 6673.333 Add Update Delete	10005. 13336.67 Copy Function Load from Library Save to Library	16668.33 20000. Frequency Get Data Series Data Paste from Clipboard Copy to Clipboard	

Damping is given as a function and is constant over the complete range of interest and since it doesn't matter, I just set it at 0.02 from 10 to 20,000 Hz. If one wants to know more, take a look at the documents: NX Nastran basic Dynamic Analysis

&

NX Nastran Advanced Dynamic Analysis User's Guide



Units for dynamic analysis can be challenging. In this example model, the units are a modified SI system of N, tonne, mm and second. For the acceleration load of 1 g we have 9,800 mm/s2. The load is applied at the independent node of the RBE2 element. It may seem funny that one can apply an acceleration load to a node that has all six DOF fixed but the modal frequency analysis understands the request and ignores the T2 SPC tag.

.oad Set 1 Motor Fr	equency				
Title Acceleration on Node Color 10 Palette	in mm/sec^2		Coord Sys	0Basic Rectangular	٠
Acceleration	Direction Component Vector Along Curv Normal to 1 Normal to 2 Load AX 0.	ts e Plane Surface ∑alue	Specify Time/Freq Dependence 0.None	Method Constant Variable Data Surface Advanced. Data Surface 0None	
	AZ 0.	500		0None	•
	Phase 0.		0None 🔹 🖍	r	



This is the heart and soul of the Modal Frequency Analysis setup. As one walks through the screens, we chose the Modal solution type, and request that 10 Eigenvalues and Eigenvectors be used to form the solution set. The next screen, we set damping to use our 0.02 critical damping curve and we request the solution frequencies. This can be done by creating your own function or letting FEMAP calculate the solution requests based on the natural frequencies. We chose the later by pressing the Modal Freq button and requesting solutions over the first four natural frequencies with a band spread of 10% (default).

Skip EIGx	Meth	od <u>I</u> D 1	Use Load Set Options	
Real Solution Methods	Range of Interest Rea	Imaginary	Options for Dynamic Analysis Advanced Options	Limit Response Based on Modes
 <u>Auto (HOU/MHOU)</u> Subspace 	F <u>r</u> om (Hz) 0.	0.	Overall Structural Damping Coeff (G) 0.	Number of Modes 0
Legacy Real Solution Methods	<u>T</u> o (Hz) 0.	0.	Modal Damping 3PSD Modal Frequer 👻 f _{xy}	Lowest Freq (Hz) 0.
© <u>G</u> ivens Modified Givens	Eigenvalues and Eigenvectors		As Structural (KDAMP)	Highest Freq (Hz) 0.
Inverse Power	Number Estimated	0	Equivalent Viscous Damping Conversion	Transient Time Step Intervals
Inverse Power/Sturm	Number Desired	10	Convert using Solution Freq (WMODAL)	Number of 0
<u>H</u> ouseholder Modified Householder Complex Solution Methods	Normalization Method	Mass O Default Lumped	Rigid Body Zero Modes(FZERO) 1.E-4 Freq for System Damping (W3 - Hz) 0. Freq for Element Damping (W4 - 0.	Time per 0. Output Interval 0
Complex Inverse Power	O Point	Coupled	Frequency Response	Response/Shock Spectrum
Complex Lanczos	Complex Solution Options Convergence	0.	Frequencies f _{xy}	Damping/Freq Correlation
Direct Modal	Overall Damping (G)	0.	Modal Freq	UNone

Since the linear dynamic response of a structure is determined or composed of its natural frequencies it often makes the most sense to request solutions at and around (Frequency Band Spread) these natural frequencies. Once this is done, the program creates a function showing how these solutions are spaced apart. The numerical value of the function is only for graphical utility since Nastran solves at each requested frequency.

•
•
%

D 5 Title Modal Fre	quency Table		Type 3vs. Frequen	cy 🗸
-Frequency Y - Factor			X Axis Log Scale	Y Axis Log Scale
80.41994 0. 80.41947 1. 90.46649 2. 10.46982 1. 90.20.51314 0. 42.17966 0. 61.18964 1. 80.19962 2. 99.2096 1. 81.21959 0. 95.31412 0. 93.894268 1. 93.824366 0. 14.26364 0. 65.05606 1. 105.8485 2. 066.6409 1. 117.4333 0.	Factor 2. 1.8 1.6 1.4 1.2 1. 1. - 1. - 1. - 1. - - 1. - - 1. - - - - - - - - - - - - -	4.6732 531.7999 648.9266	766.0533 883.18	1000.307 1117.43 Frequenc
oata Entry ● Single <u>V</u> alue	(X)	Add	Copy Function	Get Data Series Data
Clinear <u>R</u> amp Clinear <u>Ramp</u>	tude (Y) Delta X 1	Update	Load from Library	Paste from Clipboard
Equation Periodic	X Varia <u>b</u> le 🛛 🗙	Delete	Save to Library	Copy to Clipboard
A Y		Delete		

2019

Then, one applies the boundary conditions and then lastly, one sets the output requests.

Boundary Conditions	X
Primary Sets	
Constraints	1Universal All-Purpose Con: 🔻
Loads	1Motor Frequency -
Temperatures	0From Load Set
Initial Conditions	0None
Constraint <u>E</u> quations	0From Constraint Set 🛛 👻
Bolt Preloads	0From Load Set 🔹
Other DOF Sets	
M <u>a</u> ster (ASET)	0None 👻
Kinematic (<u>S</u> UPORT)	0None 🔻
SUPORT1	0None 🔻
OMIT	0None 🔻
QSET	0None 🔻
CSET	0None 🔻
BSET	0None
Prev Next	<u>O</u> K Cancel

Nodal		Elemental		
Displacement	0Full Model	<u>Force</u>	0Full Model	Ŧ
Applied Load	0Full Model	✓ <u>S</u> tress	0Full Model	•
Constraint Force	0Full Model	St <u>r</u> ain	0Full Model	
Equation Force	0Full Model	Strain Energy	0Full Model	-
Force <u>B</u> alance	0Full Model	Heat Flux	0Full Model	-
Velocity	0Full Model	Enthalpy	0Full Model	-
 Acceleration 	0Full Model	Enthalpy Rate	0Full Model	-
Kinetic Energy	0Full Model	Temperature	0Full Model	
Temperature	0Full Model	Kinetic Energy	0Full Model	-
		Energy Loss	0Full Model	-
		Fluid Pressure	0Full Model	-
Customization				
✓ E <u>l</u> ement Corner Re	esults	Results Destination	Prev	
Output Modes (a,b,c	THRU d)	2PostProcess Only	-	
		Echo Model	ОК	
Magnitude/Phase	Real/Imaginary		Cancel	



And the results show no surprises with the response peaking at the first normal mode at 200 Hz. If it is your first time with Modal Frequency, then the output results will seem a bit odd since you don't have Total Translation or a von Mises stress. All that you have are individual displacement and stress components. This goes back to the nature of the solution where the output is in magnitudes (uo) and phase angles (Θ). Hence, to get the time varying nature, you need to expand the complex results.





To obtain the time varying response from a Modal Frequency, one goes to Model / Output / Expand Complex and pick your solution of interest. For this structure it is the maximum response at 200 Hz and then we'll request that it is expanded into 12 solutions.

Model - PCB Analysis from Modal to MF to PSD to DT -	 Final Modal Frequency Analysis.modfem 				
Output Sets	Output Vectors				
	All Output Vectors From Output S	Set 11Case 1 Freq 180.4198	•		
 1Mode 1, 200.4665 Hz 2Mode 2, 380.1996 Hz 3Mode 3, 672.5712 Hz 4Mode 4, 1015.848 Hz 5Mode 5, 2022.406 Hz 6Mode 6, 2625.677 Hz 7Mode 7, 4156.374 Hz 8Mode 8, 4366.092 Hz 9Mode 9, 5601.611 Hz 10Mode 10, 6685.72 Hz 11Case 1 Freq 180.4198 12Case 2 Freq 190.4432 V 3Case 3 Freq 200.4665 14Case 4 Freq 210.4698 15Case 5 Freq 220.5131 16Case 6 Freq 342.1797 17Case 7 Freq 361.1896 18Case 8 Freq 300.1996 20Case 10 Freq 418.2196 21Case 11 Freq 605.3141 22Case 12 Freq 380.4766 23Case 13 Freq 720.4726 23Case 13 Freq 726.5712 24Case 14 Freq 706.1998 25Case 15 Freq 739.8284 26.Case 16 Freq 914.2636 27Case 17 Freq 961.5511 28Case 18 Freq 1015.848 29Case 20 Freq 1117.433 	Constant of the second se	dick Filter 0None - Ignore 70012SolidC1 Z Normal Stress 70013SolidC1 XY Shear Stress 70014SolidC1 XY Shear Stress 70015SolidC1 XY Shear Stress 70015SolidC2 X Normal Stress 70211SolidC2 X Normal Stress 70211SolidC2 X Normal Stress 70211SolidC2 X Normal Stress 70213SolidC2 XY Shear Stress 70214SolidC3 XY Shear Stress 70410SolidC3 X Normal Stress 70411SolidC3 X Shear Stress 70411SolidC4 X Normal Stress 70611SolidC4 X Normal Stress 70611SolidC4 X Normal Stress 70611SolidC4 X Normal Stress 70611SolidC4 X Normal Stress 70614SolidC5 X Normal Stress 70811SolidC5 X Normal Stress 70811SolidC5 X Normal Stress 70811SolidC5 X Shear Stress 70811S	TotalSolidC6 Y Normal Str 71012SolidC6 Z Normal Str 71013SolidC6 X Shear Str 71014SolidC6 X Shear Str 71015SolidC7 X Normal Str 71210SolidC7 X Normal Str 71211SolidC7 X Normal Str 71211SolidC7 X Normal Str 71213SolidC7 X Shear Str 71214SolidC7 X Shear Str 71214SolidC8 X Normal Str 71214SolidC8 X Normal Str 71411SolidC8 X Normal Str 71413SolidC8 X Shear Str 100007PIC1 Top Y Normal 100221PIC1 Bot X Normal 100221PIC1 Bot X Normal 100222PIC1 Bot X Normal 100222PIC2 Bottom Fiber 150008PIC2 Bottom Fiber 150008PIC2 Bottom Fiber 150008PIC2 Top Fiber 150008PIC2 Top Y Normal 150221PIC2 Top Y Normal	Expand Complex Output Expand For Single Phase Phase Range	t Data

ncel

After expanding the solution, we have the full-field solution with Total Translation and von Mises stresses. Keep in mind that this maximum response requires that the excitation is in the direction of the mode shape (Y-direction) and that this particular mode has mass (mass participation 90%).





5. INTRODUCTION TO RANDOM VIBRATION

Random Vibration is vibration which can only be described in a statistical sense. The magnitude at any given moment is not know, but is instead described in a statistical sense via mean values and standard deviations

Random vibration problems arise due to earthquakes, tsunamis, acoustic excitation (e.g., rocket launches), wind fluctuations, or any loading which is inherently random. Often random noise due to operating or transporting conditions can also be considered. These vibrations are usually described in terms of a power spectral density (PSD) function.



Page 33 of 123

5.1 THE PSD FUNCTION

Random vibration is unique because it can excite all frequencies at once, whereas a sine sweep will excite one frequency at a time (think slamming all keys on a piano instead of sliding your hand across them). The PSD function is created by subjecting a structure to white noise vibration and measuring the RMS amplitude of the response of the structure across a range of frequencies, squaring the response, and dividing it by the frequency range which results in units of G2/Hz.

A typical power spectral density is shown below:

		-	Fre	quency (Hz)	
1,100	0.01	0.001	100	1000	
900	0.05				
100	0.05	6 0.01 GS V			
20	0.01	(ZH/)			
requency (Hz)	PSD (G ² /Hz)	0.1			
		1	PSD Function		



2019

A system subject to random vibration does not have a single resultant stress. Luckily for us, the stress results do typically follow a Gaussian distribution (think bell-curve):



The Gaussian distribution allows stress results to be reported statistically. FEMAP will generate 1- σ stresses, which represent the stress that the system will likely see 68% of the time. The 2- σ stress level covers 95% of cases, and 3- σ covers 99.7%. Most of the time a system is designed to the 3- σ stress level.

5.2 THE NX NASTRAN METHOD

Given an input PSD function, an output response can be calculated by using the systems transfer function.

```
PSD_{out} = |g(w)|^2 PSD_{in}
```

The g(w) represents the system transfer function. A system transfer function simply represents its output to input ratio. NX Nastran performs a frequency response analysis on the system to obtain the system transfer function, and then does the random vibration analysis as a post processing step based on this transfer function.

There are several steps to setting up the analysis in FEMAP:

- 1. Defining the system damping
- 2. Creating the PSD function
- 3. Creating a Modal Frequency Table (or Requested Solutions Function)
- 4. Creating the excitation node and tying it into the model
- 5. Loading the model
- 6. Constraining the model
- 7. Specifying output groups for nodal and elemental output
- 8. Setting up the analysis in the Analysis Manager
5.3 PSD UNITS

It can be tricky keeping track of the units in any analysis; this is especially true with PSD analysis. The table below shows the input and output units for a few of the most common unit systems. It doesn't matter which system you go with but be sure you are consistent throughout your analysis. The table below shows the properties for aluminum in each unit system.

		Input	RMS Outputs			
Units	Young's Modulus	Mass Density	PSD Function	Acceleration Load	Deflection	Stress
SI (m,kg,sec)	6.89e10 Pa	2710 kg/m ³	1 g²/Hz	9.807 m/s ²	1 m	1000 Pa
SI (mm,Mg,sec)	6.89e4 MPa	2.71e-9 Mg/mm ³	1 g ² /Hz	9807 mm/s ²	1000 mm	1.0e-3MPa
Imperial (in, snail, sec)	10.0 e6 psi	2.54e-4 snail/in ³	1 g²/Hz	386.1 in/s ²	39.37 in	0.145 psi

2019

6. EXAMPLE 1: PSD ANALYSIS OF PCB WITH TWO HEAVY ELECTRICAL COMPONENTS

For completeness, let's do a simple PSD analysis on our circuit board. Again, units are very important. The PSD spectrum (load) is given as g2/Hz. In the center of the spectrum from 115 to 500 Hz, the PSD input is 1.31 g2/Hz and then tapers.





The PSD procedure is almost identical to the modal frequency analysis. There are some new screens but the only critical one is where you apply your PSD spectrum (lower-right-hand-corner). Otherwise, it is identical to that of the prior modal frequency analysis.

alusis Cot	X	NASTRAN Modal Analysis		X
liysis set		Skip EIG	Method JD	1
		Real Solution Methods Range (of Interest	
Title XXX-0814-01 V-Avis PSI) Analysis	Lanczos Auto (HOU/MHOU)	Raal	Imaginary
	Analysis	Subspace From (Hz) 0.	0.
		Legacy Real Solution Methods Io (Hz)) 0.	0.
Analysis Program 36NX Na	stran	© Givens Eigenva	alues and Eigenvectors	
		Modified Givens Number	r Estimated	0
Analysis Trans		Inverse Power Numge	r Desired	10
Analysis Type 6Randor	n Response 🔹 🔻	() Householder		
		Modified Householder	ss	Default
Run An	alvsis Using VisO	Complex Solution Methods	Node ID 0	C Lumped
		Hessenberg Deal	nt DOE 0	Coupled
		Complex Inverse Power	x Solution Options	
Next <u>O</u> K	Cancel	Solution Tyme Conven	gence	0.
		O Diregt Region	width	0.
		Mogal Overall	Damping (G)	0.
amic Control Options		NASTRAN Power Spectral Density Factors		
amic Control Options Use Load Set Options Dtions for Dynamic Analysis Advanced Options		NASTRAN Power Spectral Density Factors Correlation Table		
amic Control Options Use Load Set Options Utions for Dynamic Analysis Advanced Options Equivalent Viscous Damping	Limit Response Based on Modes	NASTRAN Power Spectral Density Factors Correlation Table Master=>Master 1.(1) :Int1=0	Excited	Master
amic Control Options Use Load Set Options toons for Dynamic Analysis Advanced Options quivalent Viscous Damping Overall Structural Damping Coeff (G) 0.	Limit Response Based on Modes Number of Modes	NASTRAN Power Spectral Density Factors Correlation Table Master=>Master 1.(1) :Int1=0	Excited Load Set: 1	Master Motor Frequency
amic Control Options Use Load Set Options Use Load Set Options ations for Dynamic Analysis Advanced Options aquivalent Viscous Damping Overall Structural Damping Coeff (G) Modal Damping 3PSD Modal Frequer	Limit Response Based on Modes Number of Modes Lowest Freq (Hz) 0.	NASTRAN Power Spectral Density Factors Correlation Table Master=>Master 1.(1) :Int1=0	Excited Load Set: 1	Master Motor Frequency
amic Control Options Use Load Set Options Stions for Dynamic Analysis Advanced Options Squivalent Viscous Damping Overall Structural Damping Coeff (G) 0. Modal Damping 3PSD Modal Frequer Squitzers (KRAMP) Set Structural (KRAMP)	Limit Response Based on Modes Number of Modes Lowest Freq (Hz) 0. Highest Freq (Hz) 0.	NASTRAN Power Spectral Density Factors Correlation Table Master=>Master 1.(1) :Int1=0	Excited Load Set: 1	Master Motor Frequency Master
amic Control Options Use Load Set Options Use Load Set Options Squivalent Viscous Damping Overall Structural Damping Coeff (G) 0. Modal Damping 3PSD Modal Frequer	Limit Response Based on Modes Number of Modes Lowest Freq (Hz) 0. Highest Freq (Hz) 0.	NASTRAN Power Spectral Density Factors Correlation Table Master=>Master 1.(1) :Int1=0	Excited Load Set: 1 Applied	Master Motor Frequency Master
amic Control Options Use Load Set Options Advanced Options Soutions for Dynamic Analysis Advanced Options Soutianal Demping Overall Structural Damping Coeff (G) As Structural (KDAMP) Soutianal Conversion Conversion Soutianal Conversion Soutiana Soutiana Soutiana Soutiana Soutiana Soutiana Soutiana So	Limit Response Based on Modes Number of Modes Lowest Freq (Hz) 0. Highest Freq (Hz) Transient Time Step Intervals	NASTRAN Power Spectral Density Factors Correlation Table Master=>Master 1.(1) :Int1=0	Excited Load Set: 1 Applied Load Set: 1	Master Motor Frequency Master Motor Frequency
amic Control Options Use Load Set Options Storms for Dynamic Analysis Advanced Options Storward Viscous Damping Overall Structural Damping Coeff (G) As Structural (KDAMP) Storward Viscous Damping Conversion Convert using Solution Freq (WMODAL)	Limit Response Based on Modes Number of Modes Lowest Freq (Hz) 0. Highest Freq (Hz) 0. Transient Time Step Intervals Number of 0	NASTRAN Power Spectral Density Factors Correlation Table Master=>Master 1.(1) :Int1=0 Fdit Correlation Table	Excited Load Set: 1 Applied Load Set: 1	Master Motor Frequency Master Motor Frequency
amic Control Options Use Load Set Options Use Load Set Options Advanced Options Calvalent Viscous Damping Overall Structural Damping Coeff (G) 0. Modal Damping As Structural (KDAMP) Calvalent Viscous Damping Conversion Convert using Solution Freq (WMODAL) Rigid Body Zero Modes(FZER0) 1.E-4	Limit Response Based on Modes Number of Modes Lowest Freq (Hz) 0. Highest Freq (Hz) 0. Transient Time Step Intervals Number of 0 Time per 0.	NASTRAN Power Spectral Density Factors Correlation Table Master=>Master 1.(1) :Int1=0 Edit Correlation Table	Excited Load Set: 1 Applied Load Set: 1	Master Motor Frequency Master Motor Frequency
amic Control Options Use Load Set Options Use Load Set Options Advanced Options Course of Dynamic Analysis Advanced Options Course of C	Limit Response Based on Modes Number of Modes Lowest Freq (Hz) 0. Highest Freq (Hz) 0. Transient Time Step Intervals Number of 0 Time per 0. Output Interval 0	NASTRAN Power Spectral Density Factors Correlation Table Master=>Master 1.(1) Edit Correlation Table Factor PSD Function	Excited Load Set: 1 Applied Load Set: 1	Master Motor Frequency Master Motor Frequency PSD Interpo
amic Control Options Use Load Set Options Use Load Set Options Advanced Options quivalent Viscous Damping Overall Structural Damping Coeff (G) As Structural (KDAMP) quivalent Viscous Damping Conversion Convert using Solution Freq (WMODAL) Rigid Body Zero Modes(FZERO) 1.E-4 Freq for System Damping (W3 - He) D. Freq for System Dampi	Limit Response Based on Modes Number of Modes Lowest Freq (Hz) 0. Highest Freq (Hz) 0. Transient Time Step Intervals Number of O Time per O. Output Interval O	NASTRAN Power Spectral Density Factors Correlation Table Master=>Master 1.(1) :Int1=0 Edit Correlation Table Factor PSD Function Real 1. x 1PSD Function for	Excited Load Set: 1 Applied Load Set: 1 on	Master Motor Frequency Master Motor Frequency PSD Interpo
amic Control Options Use Load Set Options Advanced Options Stations for Dynamic Analysis Advanced Options Stativalent Viscous Damping Overall Structural Damping Coeff (G) 0. Modal Damping 3PSD Modal Frequer As Structural (KDAMP) Guivalent Viscous Damping Conversion Convert using Solution Freq (WMODAL) Rigid Body Zero Modes(FZERO) 1.E-4 Freq for System Damping (W3 - H2) 0. Freq for Element Damping (W4 - 0.	Limit Response Based on Modes Number of Modes Lowest Freq (Hz) 0. Highest Freq (Hz) 0. Transient Time Step Intervals Number of 0 Time per 0. Output Interval 0	NASTRAN Power Spectral Density Factors Correlation Table Master=>Master 1.(1) Edit Correlation Table Edit Correlation Table Factor PSD Function Real 1. × 1PSD Function (g)	Excited Load Set: 1 Applied Load Set: 1 on g^2/Hz)	Master Motor Frequency Master Motor Frequency PSD Interpo
amic Control Options Use Load Set Options Advanced Options Stoins for Dynamic Analysis Advanced Options Stouwalent Viscous Damping As Structural Damping Conversion Convert using Solution Freq (VMMODAL) Rigid Body Zero Modes(FZERO) 1.E-4 Freq for System Damping (W3 - Hc) Rigid For Element Damping (W4 - 0, Frequency Response	Limit Response Based on Modes Number of Modes Lowest Freq (Hz) Highest Freq (Hz) Translent Time Step Intervals Number of Time per Output Interval Response/Shock Spectrum	NASTRAN Power Spectral Density Factors Correlation Table Master=>Master Linit=0 Edit Correlation Table Factor PSD Function Real I. × Imaginary 1. × 0None	Excited Load Set: 1 Applied Load Set: 1 on g^2/Hz) •	Master Motor Frequency Master Motor Frequency PSD Interpo 0Log Log
amic Control Options Use Load Set Options Use Load Set Options Statuns for Dynamic Analysis Advanced Options Equivalent Viscous Damping Overall Structural Damping Coeff (G) 0. Modal Damping 3PSD Modal Frequer Gavert using Solution Freq (WMODAL) Convert using Solution Freq (WMODAL) Rigid Body Zero Modes(FZERO) 1.E-4 Freq for System Damping (W3 - H2) 0. Freq for Element Damping (W3 - H2) Frequency Response Frequencies 5Modal Frequency Table for	Limit Response Based on Modes Number of Modes Lowest Freq (Hz) Highest Freq (Hz) Transient Time Step Intervals Number of Time per Output Interval Response/Shock Spectrum Damping/Freq Correlation	NASTRAN Power Spectral Density Factors Correlation Table Master=>Master 1.(1) Edit Correlation Table Factor PSD Function Real 1. × 1PSD Function Imaginary 1. × 0None	Excited Load Set: 1 Applied Load Set: 1 on g^2/Hz) •	Master Motor Frequency Master Motor Frequency PSD Interpo 0Log Log 0Log Log
amic Control Options Use Load Set Options Use Load Set Options Advanced Options Convert Viscous Damping As Structural Damping Conversion Convert using Solution Freq (WMODAL) Rigid Body Zero Modes(FZERO) 1.E-4 Freq for System Damping (W3 - H2) 0. Frequency Response Frequencies S.:Modal Frequency Table	Limit Response Based on Modes Number of Modes Lowest Freq (Hz) 0. Highest Freq (Hz) 0. Transient Time Step Intervals Number of 0 Time per 0. Output Interval 0 Response/Shock Spectrum Damping/Freq Correlation	NASTRAN Power Spectral Density Factors Correlation Table Master=>Master 1.(1) Edit Correlation Table Edit Correlation Table Factor PSD Function Real 1. X 1PSD Function (c) Imaginary 1. X 0None	Excited Load Set: 1 Applied Load Set: 1 on g^2/Hz) ~	Master Motor Frequency Master Motor Frequency PSD Interpo 0Log Log 0Log Log
amic Control Options Use Load Set Options Use Load Set Options Solutions for Dynamic Analysis Advanced Options Solutions for Dynamic Analysis Advanced Options Solution I Structural Damping Coeff (G) 0. Modal Damping 3PSD Modal Frequer Gamma Solution Freq (WMODAL) Gamma Solution Freq (WMODAL) Rigdi Body Zero Modes(FZER0) 1.E=4 Freq for System Damping (W3 - H2) 0. Freq for System Damping (W3 - H2) 0. Freq for System Damping (W4 - 0. Frequencies 5Modal Frequency Table frequencies 5Modal Frequency Table Modal Freq	Limit Response Based on Modes Number of Modes Lowest Freq (Hz) 0. Highest Freq (Hz) 0. Transient Time Step Intervals Number of 0 Time per 0. Output Interval 0 Response/Shock Spectrum Damping/Freq Correlation 0None for	NASTRAN Power Spectral Density Factors Correlation Table Master=>Master 1.(1) Edit Correlation Table Factor PSD Function (g) Imaginary 1. X 0None Apply	Excited Load Set: 1 Applied Load Set: 1 on g^2/Hz) ~	Master Motor Frequency Master Motor Frequency PSD Interpo 0Log Log
amic Control Options Use Load Set Options Sotions for Dynamic Analysis Advanced Options Squivalent Viscous Damping Overall Structural Damping Coeff (G) 0. Modal Damping 3PSD Modal Frequer As Structural (KDAMP) Convert using Solution Freq (WMODAL) Rigid Body Zero Modes(FZERO) 1.E-4 Freq for System Damping (W3 - H2) 0. Freq for Element Damping (W4 - 0. Frequency Response Frequencies SModal Frequency Table Kodal Freq	Limit Response Based on Modes Number of Modes Lowest Freq (Hz) 0. Highest Freq (Hz) 0. Transient Time Step Intervals Number of 0 Time per 0. Output Interval 0 Response/Shock Spectrum Damping/Freq Correlation 0None v freq	NASTRAN Power Spectral Density Factors Correlation Table Master=>Master 1.1) Edit Correlation Table Factor PSD Function (g Imaginary Apply	Excited Load Set: 1 Applied Load Set: 1 on g^2/Hz)	Master Motor Frequency Master Motor Frequency PSD Interpo 0Log Log 0Log Log



Given that a PSD analysis can be a numerically intensive calculation, FEMAP provides the ability to restrict your analysis output to just a few items or the complete model. For this analysis, all output requests are left blank except the very last screen where just displacements and stresses are requested. This is identical to that which was done for the modal frequency analysis. At the end, we have the RMS von Mises stresses contoured over the system and they are significantly greater than just the modal frequency result.



7. EXAMPLE 2: CANTILEVER BEAM

7.1 PROBLEM DEFINITION

A cantilevered aluminum beam 5 inches in length is used to support a 0.50 lb mass. Our objective is to determine the dynamic stresses and fatigue life of the beam for vibration along the vertical axis.

The FEA model is a single beam element. A picture of the beam element, with its cross section displayed is shown on the right.

We will compare the FEA results to an analytical solution ψ . The PSD **input used by St**

$PSD_{in} = 0.2 \ G^2/Hz$

This Excitation was applied to the fixed end of the beam (where the rectangle is drawn)

Our unit system is lb/in/s and 1 g=386 in/s²



^{*V*} Steinberg, Dave S. <u>Vibration Analysis for Electronic Equipment</u>. 2nd ed. New York: John Wiley & Sons, 1988. 226-231.



7.2 ANALYTICAL SOLUTION

A cantilever beam with the dimensions previously given and an end load of 0.5 lbf experiences an end deflection of:

$$Y_{St} = \frac{WL^3}{3EI} = 8.01E - 4$$

Based upon this end deflection, the beam's resonant frequency and transmissibility can be calculated as:

$$f_n = \frac{1}{2\pi} \sqrt{\frac{g}{Y_{St}}} = 110.5$$
 $Q = 2\sqrt{f_n} = 21$

Miles' equation can be used to approximate the Gout(RMS) value:

$$G_{out} = \sqrt{\frac{\pi}{2}} PSD_{in} * f_n * Q = 27.0$$

This output is in G, if an equivalent value is desired in English units, simply multiply this by gravity.

$$27G = 27 \frac{acceleration}{gravity} * gravity = 10,422 \frac{in}{s^2}$$

The max output PSD can also be obtained using:

$$PSD_{out} = Q^2 * PSD_{in} = 21^2 * (0.2 * G^2)$$
 where $G = 1g$ or $386 in/s^2$

In English units, the max PSD_{out}=13.14e6 in²/s⁴. This can also be verified against the FE Model

7.3 DEFINING THE SYSTEM DAMPING

Determining how the system is damped can be complicated. In NX Nastran there are three ways to do this:

- If the structural damping coefficient (G) is known then function type 6: Structural Damping vs. Frequency should be used.
- If the critical damping ratio is known, then function type 7: "Critical Damping vs Frequency" should be used.
- If the Quality/Magnification factor (Q) is known, then function type 8: "Q Damping vs. Frequency" should be used.

Punction Dem	nition								×
ID 2	Title Damping			Type 7	Critical Da	mp vs	. Freq		~
X - Frequency	Y - Frac Crit			X Axis L	.og Scale		Y Axis	Log Sca	le
). L.	0.0238	Frac Crit .0476 - .04165 - .0357 - .02975 - .0238 - .01785 -							
		.0119 - .00595 - 0 0. 0.1	0.2 0.3	0.4 0.5	0.6	0.7	0.8	0.9 Free	1 quenc
Data Entry	C Edit Phase ()	.0119 - .00595 - 0. 0. 0.1	0.2 0.3	0.4 0.5	1 0.6	0.7	0.8	0.9 Fred	1 quenc
Data Entry ● Single ⊻alue ◯ Linear Ramp	◯ Edit P <u>h</u> ase ()	x) de (Y) Delta X 1	0.2 0.3	0.4 0.5	0.6 unction	0.7	0.8 Get Date	0.9 Free a Series	1 quenc Data
Data Entry Single Value Linear Ramp Equation	 Edit Phase () Edit Magnitud Periodic 	X) K) X) X) X) X) X) X) X) X) X) X	0.2 0.3	0.4 0.5	unction	0.7	0.8 Get Dati	0.9 Free a Series om Clipt	1 quenc Data poard
Data Entry Single Value Linear Ramp Equation X 1.	C Edit Phase () C Edit Magnitud Periodic Y 0.0	x) de (Y) X Variable x 238	0.2 0.3 Add Update Delete	0.4 0.5	0.6 unction m Library	0.7	0.8 Get Date Paste fro Copy t	0.9 Frec a Series om Clipt o Cliptoc	1 quenc Data Doard

An approximation of the transmissibility of the beam is Q = 21. This value yields a critical damping ratio of 2.38%; this is what we will use.

7.4 CREATING THE PSD FUNCTION

The input to the cantilever beam is a white-noise vibration with a PSD input of 0.20 G2/Hz from 20 to 2000 Hz.

This is entered directly with no scaling. It will be scaled for the desired unit system in the Load Definition dialog (Section 5.6).



7.5 CREATING THE MODAL FREQUENCY TABLE/SETTING UP THE LOAD SET OPTIONS FOR DYNAMIC ANALYSIS

The Modal Frequency Table is a function which defines which frequencies NX Nastran will obtain a solution for; that is, each frequency represents a separate solution that is written out to the results file. The function can either be created manually, or FEMAP can create one for you. If you do not know about which frequencies you'd like the analysis to focus, it is preferable to have FEMAP set it up, otherwise you will most likely end up with a large amount of extraneous output.

Analysis Set		×	
Title Eigenvalue			
Analysis <u>P</u> rogram	36NX Nastran	~	
Analysis Type	2Normal Modes/Eigenvalue		
	<u>Run Analysis</u> Us	ing VisQ	
Next	<u>O</u> K	Cancel	

To have FEMAP set up the table for you, you must first run an eigenvalue analysis. Once the eigenvalue analysis has run, FEMAP will know about which frequencies to concentrate.

The normal modes will be used to define the solution frequencies of the Random Analysis. Think of it as guiding the Random Analysis such that only frequencies of interest (significant frequencies) are processed. This greatly limits the amount of post-processing that is required for the Random Analysis. More will be said on this later on....



It is good practice to run the normal modes analysis first to see how the structure will behave. In this simple beam model, we have fixed the end of the beam in all six DOF. The beam is also massless (material density of 0.0). This was done to allow us to exactly match the analytical solution.

After the analysis has finished running, you should have three modes. In Section 5.8 we will show you how these Normal Modes are used to generate the Solution Frequencies for the Random Analysis.



123456

7.6 LOADING & CONSTRAINING THE MODEL

An acceleration load must be given to the base node in the direction of the excitation. Since the PSD is given in G^2/Hz , we must scale the load by a 1 g gravitational acceleration in our unit system of choice. We want our deflection results in inches so we will enter an acceleration of 386 in/s².

Title	Acceleration on Nod	e .		Coord Sys	0 Basic R	ectangular	~
Color	10 Palette	Layer 1					
Accelera	ation	Direction © Comp O Vecto Along Norm	r r Curve al to Plane al to Surface	Specify		d ionstant ariable bata Surface Advanced	
		Load	Value	Time/Freg Dependence		Data Surface	
		AX [0.	0None v	‰ 0Nor	ne 🗸	
		AY [386.		0Nor	ne 🗸	
		AZ [0.		0Nor	ne 🗸 🗸	
		Dha		0.None	4		

The **Load Constraint** constrains the base node in all sex degrees of freedom

This constraint set should identical to the constraint set used for the eigenvalue analysis. The node used to constrain the model is the same node to which the unit acceleration was applied.

The idealization concept is that the base is fixed in the TX, TZ, RX, RY, RZ while the structure is excited in the Y-direction (i.e., there is displacement in the Ydirection).

Create Nodal Co	nstraints/D	OF					×
Constraint Set 1	Lo	ad Constraint					
Title				Coord Sys	0Bas	ic Rectangular	~
Color 120	Palette	Layer	. 1				
DOF	_	-		X Symmet	try	X AntiSym	
עדע עדע [√TZ	Fixed	Pinned	Y Symme	trv	Y AntiSym	<u>O</u> K
	(D7	Eree	No Detation		1000		Concol



7.7 SPECIFYING GROUPS FOR NODAL AND ELEMENTAL OUTPUT

A group can be created to specify certain nodes and elements to recover data from. For this analysis we will skip creating a group to simplify the analysis.

If we wanted a group for the beam element we could create a single group with our single element and two nodes. We are not recovering any data from the Mass Element, so we can leave it out of the group.



7.8 CREATING AN ANALYSIS SET – SIMPLE PSD

Next up is creating an analysis set. There are a lot of options to tailor the output to exactly what you need, but let's look at a straightforward analysis. This will allow you to see RMS Stress and positive crossings, which is enough information for a general PSD stress analysis and fatigue life estimate.

First, create a new Random Response Analysis Set.

Select Next...

Analysis Set		×
Title PSD Analysis		
Analysis <u>P</u> rogram	36NX Nastran	~
Analysis Type	6Random Respon	ise ~
Solve Using		
Integrated Solver		
\bigcirc Linked Solver		
Solver is undefine	ed. Go to File Prefere	ences Solvers.
⊖⊻isQ		
Ne <u>x</u> t	<u>O</u> K	Cancel

Keep pressing **Next...** until you arrive at the *NASTRAN Modal Analysis* window.

In the modal analysis tab you can decide between a Direct or Modal Solution Type. For this analysis, we will use a Modal solution. For more information about the difference in solution types take a look at the NX Nastran Basic Dynamic Analysis User Guide, Chapter 6.4 Modal Versus Direct Frequency Response.

For Range of Interest you can set the maximum frequency at your upper limit of the PSD spectrum. This will guarantee your entire PSD spectrum is covered and not spend extra computing power (and time) processing frequencies above that.

NASTRAN Modal Analysis			×		
Skip EIGx	Danaga of Internet	Method ID	1		
Eanczos Auto (HOU/MHOU)	Range of Interest	Real	Imaginary		
	From (Hz)	0.	0.		
Legacy Real Solution Methods	Io (Hz)	2000.	0.		
	Eigenvalues and Eigenvectors				
	Number Estimated		0		
O Inverse Power/Sturm	Number Desired		0		
 ○ Householder ○ Modified Householder 	Normalization Meth	od	Mass Default		
Complex Solution Methods Hessenberg Complex Inverse Power	O Max O Point	0	Coupled		
O Complex Lanczos	Complex Solution O	ptions			
Solution Type	Convergence		0.		
) Direct	Region Width		0.		
● Modal	Overall Damping (G)	0.		
Prev Ne <u>x</u> t		<u>O</u> K	Cancel		



In the Dynamic Analysis tab, one can specify the damping function and define the frequency range of the analysis (# of modes, or Lowest and Highest Frequency).

For Frequency Response, Select the "**Modal Freq...**" button, and then choose the modes you would like to create a Modal Frequency Table from. For this analysis only the first mode will be selected to match up with the analytical solution.

It is recommended to use the default values for the Points per Mode and Frequency Spread. See appendix for details.

IASTRAN Dynamic Analysis			
Use Load Set Options		Frequency ID	2
Options for Dynamic Analysis Solution	Frequencies		
Equivalent Viscous Damping		Limit Response Based	l on Modes
Overall Structural Damping Coeff (G)	0.	Number of Modes	0
Modal Damping 2Damping	~ 4	Lowest Freq (Hz)	0.
As Structural (KDAMP)		Highest Freq (Hz)	2000.
Equivalent Viscous Damping Conversion	ı	Transient Time Step 1	intervais
Convert using Solution Freq (WMC	DDAL)	Number of	0
Rigid Body Zero Modes(FZERO)	1.E-4	Time per	0.
Freq for System Damping (W3 -	0.	Output Interval	0
Freq for Element Damping (W4 -	0.		
Frequency Response		Response/Shock Spec	trum
Frequencies 4Modal Frequency Ta	ble 🗸 🙀	Damping/Freq Corre	elation
Modal Freq		0None	~ 69
Prev Next		OK	Cancel

Eirst Freq	1Mode 1, 110.4497 Hz		~
Last Freq	1Mode 1, 110.4497 Hz		~
ditional Solu	ition Frequency Points		
dditional Solu Number of P	tion Frequency Points toints per Existing Mode	5	



In the NASTRAN XY Output for Modal Analysis window, you can leave all of the options un-checked. This information can be gathered when you run a standard modal analysis so there is no need to request it here.

In the NASTRAN Output for Random Analysis window, select none for nodal and elemental output.

PSD Functions: Generates 'PSDF' output set for each frequency in the Modal Frequency Table

Autocorrelation Functions: Creates output for the autocorrelation functions if applicable

Root Mean Square: Generates 'CRMS' results for each frequency in the Modal Frequency Table

NASTRAN XY Output for	Modal Analysi	s X
Output Requests		
Summary		
Modal Participation F	actors	
Modal Effective Mass	actors	
Modal Effective Weig	ht	
Modal Effective Mass	Fraction	
	Traction	
Reference Node 0		
Prev Ne <u>x</u> t	<u>O</u> K	Cancel
Nodal Output Requests None Power Spectral Density Autocorrelation Function Root Mean Square All	y Functions	
Elemental Output Requests None Power Spectral Density 	y Functions	
O Autocorrelation Function	ons	
O Root Mean Square		
Prev Ne <u>x</u> t	<u>O</u> K	Cancel

Dynamic Analysis User Guide



If you are interested in getting data for your entire structure, deselect everything in the NASTRAN Output for Random Analysis window. This will give you 1-o stress results for your full model. For this example deselect all.

If you have an extremely large model and you only want specific nodal outputs, or results from certain elements, this is where you specify that. You can also use this window to request specific data such as T2 acceleration for a group of elements and nodes that you could have created in Section 5.7. If you select PSDF it will generate a function with the acceleration vs frequency for a group.

NASTRAN Output	for Random	Analysis			Х
Nodal Output Requ	ests				
Displacement	□T1 □T2	□Т3	R1	R2	R3
Velocity	□T1 □T2	□Т3	□R1	R2	🗌 R3
Acceleration	T1 T2	T 3	□ R1	R2	🗌 R3
Elemental Stresses					
<u>S</u> prings	Axial				
Rods	Axial		nal		
Bars	Axial				
End	A Loc 1	Loc 2		3	Loc 4
End	B Loc 1	Loc 2		3	Loc 4
Beams End	A Loc 1	Loc 2		3	Loc 4
End	B 🗌 Loc 1	Loc 2	Loc	3	Loc 4
Plates Botto	m X Norma	al 🗌 Y	Normal		Y Shear
Т	op 🗌 X Norma	al 🗌 Y	Normal		Y Shear
Solids	X Norma	al 🗌 Y	Normal	Z	Normal
	XY Shea	ar 🗌 YZ	Z Shear		X Shear
A <u>x</u> isym	Radial	Azim	Axia	l 🗌	Shear
Forces					
Torces					
Summary Data	Only	PSD	ΡF		UTO
Prev	Ne <u>x</u> t	<u>0</u>	к	(Cancel

Select your PSD Function and be sure to select Apply. If desired you can scale the PSD function in the "Factor" input here.

laster=>Master 1.(3)	:Int1=0 Excite	d Subcase: N	laster	
	Loa	d Set: 1PS	iD Load	
	Applie	d Subcase: N	laster	
	Loa	d Set: 1PS	D Load	
dit Correlation Table				
Factor	PSD Function		PSD Interpolation	ı
Real 1. x	3PSD Function	~	0Log Log	~
Imaginary 1. x	0None	~	0Log Log	~
Apply				
Autocorrelation Function T	Time Lag			
	Starting Lag 0	•	Max Lag 0.	
Lag Intervals				

Choose your constraint set and load created for the PSD analysis

Constraints	1 Eived	
Constraints		~
Loads	1PSD Excitation Node	~
Temperatures	0From Load Set	~
Initial Conditions	0None	~
Constraint <u>Equations</u>	0From Constraint Set	~
Bolt Preloads	0From Load Set	~
Other DOF Sets		
Master (ASET)	0None	~
Kinematic (<u>S</u> UPORT)	0None	~
SUPORT1	0None	~
OMIT	0None	~
QSET	0None	~
CSE <u>T</u>	0None	~
BSET	0None	~

Choose the output requests desired. For this analysis we will request Displacements, Equation Force, Acceleration, and Stress.

Magnitude/Dhase		Ec	ho <u>M</u> odel		∑K
Output Modes (a,b,c	THRU d)	[2PostProcess Only	×	
Element Corner Re	esults	Re	esults Destination	Pre	ev
Customization					
			Fluid Pressure	0Full Model	~
			Energy Loss	0Full Model	
Temperature	0Full Model	~	Kinetic Energy	0Full Model	~
Kinetic Energy	0Full Model	\sim	Temperature	0Full Model	
Acceleration	0Full Model	~	Enthalpy Rate	0Full Model	
Velocity	0Full Model	~	Enthalpy	0Full Model	
Force Balance	0Full Model	~	Heat Flux	0Full Model	
Equation Force	0Full Model	~	Strain Energy	0Full Model	
Constraint Force	0Full Model	~	Strain	0Full Model	
Applied Load	0Full Model	~	Stress	0Full Model	×
Displacement	0Full Model	~	Eorce	0Full Model	
Nodal			Elemental		

7.9

INTERPRETING THE OUTPUT

The PSD output sets are titled RMS Values and Positive Crossings. RMS Values will give all of the traditional stress, displacement, and acceleration data. Positive Crossings will output the frequency of positive crossings for each of the requested output vectors. This frequency is utilized to calculate fatigue damage based on the duration of excitation.

In the RMS Values output set you can contour all the usual output vectors. Beam EndA Pt1 Comb Stress is shown contoured over the beam. This output shown is the RMS Stress, and is also known as the $1-\sigma$ PSD stress value. This represents how much stress the beam will experience 68.3% of the time.



7.10 POSITIVE CROSSINGS

This is a vibration analysis, so of course we are also concerned about fatigue. We will use the output from positive crossings to calculate the fatigue life.

To access data for the positive crossings, Right click on the Positive Crossings result in the model info tree, and select "Post Data"



Select PostProcessing Data View 1 default Dynamic Max/Min Output Sets Program : NX Nastran 10.. Positive Crossings ~ 4 * Analysis Type : Random Set Value : 0 **Output Vectors** Deform 39 Beam EndA Pt1 Comb Stra ~ Vi Transform.. Vector Info... 110.4009 110,4009 Max : Element 1 nent 1 ~ 4 4 Contour Vector Info ... Transform. Max : Min Double-Sided Planar Contours Multiple Contour Vectors... Contour Options. Contour Arrows. Trace Locations. OK Laminate Options Section Cut.. Streamline Options. Cancel

In the Post Data toolbar select the Dynamic Max/Min box in the upper right

Select the output vector for the positive crossing frequencies desired. In this model, all stress recovery points on Beam EndA show the same frequency.

2019

Positive Crossings can also be contoured over the model. This can help the user understand how the positive crossing frequency changes throughout the model.

PostProcessing Toolbox			×
🍓 • 🚰 🚯 眠 ⊨ • 🖽 • 🛛 • 🚮 🍮			
Deform			×
Contour			×
Style	Contour		~
Results			
Output Set	17Positive Crossings +	0	+
Output Vector	3154Beam EndB Pt4 Comb St+	0	+
Additional Vector(s)			
Options			
Transform			
Data Conversion	Average		~
Data Selection	Contour Group		\sim
▪ Type	Elemental		~
Show On Groups	Full Model / Visible Groups	\sim	æ
Show As	Filled		~
• Levels			
• Legend			
Freebody			×
Output Set			
BModel Info Meshing	PostProcessing		1+~

7.11 FATIGUE ANALYSIS USING RMS STRESS AND POSITIVE CROSSINGS

We can see that Beam EndA Pt1 Comb Stress vector gives a positive crossing frequency of 110.4 Hz. This means that given the white noise PSD input of 0.2 G2/Hz, the beam will experience a fully reversible stress of 3,162 psi at a frequency of 110.4 Hz.

Statistically speaking, this stress value represents the $1-\sigma$ value and will be experienced 68.3% of the time. A 2- σ stress of 2*3,162 or 6,324 psi will be experienced 27.1% of the time and a $3-\sigma$ value of 9,486 psi will be experienced 4.33% of the time. These values represent 99.73% of the stresses the beam will see at point A. It is probable that the beam will see stresses at and above the 4σ level, but this will only happen 0.27% of the time, so we will ignore them.

All three σ level stresses fall into the "run-out" range on a fatigue curve for aluminum. To demonstrate how to treat the problem if this is not the case, let us assume that there is a small hole in the beam which causes a stress concentration factor of 3. This would put the $1-\sigma$ stress level at 9,486 psi. We can use Miner's cumulative damage index to get a sense of how long the beam will last under this condition. Miner's cumulative damage is given by the equation on the right.



7.12 FATIGUE ANALYSIS – TIME TO FAILURE

On the right is a table containing values taken from a fatigue curve for aluminum. For a given stress, the amount of cycles needed to cause failure is given.

These values can be substituted into Miner's equation to calculate how many cycles can occur until the beam fails. Substituting in the values and solving for n, yields a beam life of 1.80E6 cycles. If the beam is vibrating at a frequency (number of positive crossings) of 110.4 Hz, then it will take the beam approximately 16,300 seconds or about 4.5 hours to fail.

As long as the beam is exposed to the while noise vibration for less than 4.5 hours, it should not fail.



Point A	1σ	2σ	3σ
Stress	9,486 psi	18,972 psi	28,458 psi
# of Cycles to Fail	infinite	11.0E6 cycles	14.0E4 cycles

$$1 = \frac{0.6831 \cdot n}{\infty} + \frac{0.271 \cdot n}{11.0E5} + \frac{0.0433 \cdot n}{14.0E4}$$



8. EXAMPLE 3: SOLID MESHED BEAM

The beam properties are shown below:

Let's take a look at the same beam geometry modeled with solid elements. The beam is massless, with a point mass of 0.5lbf (1.30e-3 snails) attached via RBE2 on the end.





8.1 ANALYTICAL SOLUTION

Let's first take a look at the hand calculations to show how the beam is expected to behave. First up is maximum deflection Y_{max}

$$Y_{max} = \frac{WL^3}{3EI} = 8e - 4 in$$

Based upon this deflection, the beam's first natural frequency and transmissibility can be calculated as:

$$f_n = \sqrt{\frac{1}{2\pi} \left(\frac{g}{Y_{max}}\right)} = 110.6 \, Hz \qquad \qquad Q = 2\sqrt{f_n} \approx 21$$

Utilizing Miles' Equation to estimate Grms we see that Grms is approximately 27 Gs:

$$G_{outRMS} = \sqrt{\frac{\pi}{2} PSD_{in} f_n Q} = 27G's$$

8.2 PSD FUNCTION INPUT

Then we generate the functions necessary for the PSD Analysis. Note the Modal Frequency Table is centered at the first natural frequency with 10% spread in both directions.



8.3 PSD STRESS RESULTS

After running the analysis, let's take a look at the results. The PSD results can be validated by checking the resultant acceleration against the Miles' equation prediction. Miles' equation predicted 27 G's for the maximum acceleration. The results show an acceleration of 10,300 in/s2 which matches up with the Miles' equation prediction.



2019

8.4 COMPARING MILES' APPROXIMATION AND PSD RESULTS

An additional verification is done by comparing the PSD stress results to the static analysis with the acceleration given by Miles' equation. The images below show an 8% difference between the two results, with similar stress patterns. In addition, the hand calculations show ~10% higher stresses than the static analysis.



Hand Calculations:

$$F_d = 27 * W * S_a = 13.5 \, lbf$$

$$Stress = \frac{Mc}{I} = \frac{(F_d L)(\frac{T}{2})}{I} = 3,240 \ psi$$

This comparison between the PSD results, Miles' equation, and hand calculations offer some insight into the relative accuracy of the analysis.



9. DIRECT TRANSIENT ANALYSIS

Sometimes you just want to whack the structure and not mess around. In this scenario, we are going to hit the circuit board with a 100 g pulse at a frequency of 200 Hz in the Y-direction (one can detect a theme to this seminar?). The procedure just requires a function for the hit and then a few setup screens. The equation of motion is even simpler:

$$F_o(t) = m\frac{\partial^2 u}{\partial t^2} + c\frac{\partial u}{\partial t} + ku$$

Our equation is developed in FEMAP using a sin((!x/0.004988)*180) to create a 200.4 Hz half-sine wave:





For our work, we are just going to use half the sin wave to give the system a shock pulse. The load for this analysis is 100 g (98,000 mm/s2) with our half-sine function at 200.4 Hz.





Our transient analysis is based on the first ten Eigenvalues and Eigenvectors.

Skip EIGx		Method ID	1
Real Solution Methods	Range of Intere	st	
 Lanczos <u>Auto (HOU/MHOU)</u> 		Real	Imaginary
Subspace	F <u>r</u> om (Hz)	0.	0.
Legacy Real Solution Methods	<u>T</u> o (Hz)	0.	0.
Givens	Eigenvalues and Eigenvectors		
Inverse Power	Number Estima	ted	0
Inverse Power/Sturm	Number Desire	ł	10
O Householder	Normalization N	lethod	Mass
Modified Householder	Mass		Default
Complex Solution Methods	Max		C Lumped
Complex Inverse Power	© Point DO	0	Coupled
Complex Lanczos	Complex Solution	on Options	
Solution Type	Convergence		0.
Direct	Region Width		0.
Modal	Overall Dampin	g (G)	0.

Dynamic Control Options			X
Use Load Set Options			
Options for Dynamic Analysis Advanced	Options		
Equivalent Viscous Damping		Limit Response Base	d on Modes
Overall Structural Damping Coeff (G)	0.	Number of Modes	0
Modal Damping 3PSD Modal F	Modal Damping 3PSD Modal Frequer 👻 👧		0.
As Structural (KDAMP)	As Structural (KDAMP)		0.
Equivalent Viscous Damping Conversion	1	Transient Time Step	Intervals
Convert using Solution Freq (WMC	DAL)	Number of	1000
Rigid Body Zero Modes(FZERO) 1.E-4		Time per	1.E-5
Freq for System Damping (W3 - Hz) 0.		Output Interval	10
Freq for Element Damping (W4 -	0.		
Frequency Response		Response/Shock Spe	ctrum
Frequencies 0None	▼ f _{xy}	Damping/Freq Corro	elation
Modal Freq		0None	▼ f _{xy}
Prev Next		<u></u> K	Cancel



At the end of the simulation, one has a hundred result sets to claw through.







A much simpler way to process transient results is to use the FEMAP envelope function and then select all the output sets with "time" in the title:

Filter Title Contains	time OK Cancel	
Process Output Data		
Vhat to Process		
Com	vlete Output Sets One or More Selected Output Vectors	
rocessing Operations		
Copy Merge Linear Com	pination RSS Combination Envelope Error Estimate Convert	
Type	Envelope Approach Create Envelopes	
Max Value	© Envelope All Selected Vectors	
Min Value	Envelope All Locations For Each Vector Across Output Sets	
Max Absolute Value	Envelope Each Vector Independently	
	Store Set/Location Info	
	Envelope Every Vector across Multiple Output Sets	
Select Output To	Process Store Output in Set	
perations That Will Be Proce	sed - Review Before Pressing OK	
ax 45Case 1 Time 0> Ne ax 46Case 2 Time 1.E-4	w Set Envelope Each Vector Across Sets with SetInfo	Reset
ax 47Case 3 Time 0.0002		- Heart
ax 48Case 4 Time 0.0003 ax 49Case 5 Time 0.0004		= Delete
ax 50Case 6 Time 0.0005		
ax 51Case 7 Time 0.0006 ax 52Case 8 Time 0.0007		
ax 53Case 9 Time 0.0008		
ax 54Case 10 Time 0.0009 ax 55Case 11 Time 0.001		
ax 56Case 12 Time 0.0011		
ax 58Case 14 Time 0.0012		
ax 59Case 15 Time 0.0014		
lax 61Case 17 Time 0.0016		
ax 62Case 18 Time 0.0017		
av 62 Cace 10 Time 0 0019		
ax 63Case 19 Time 0.0018 ax 64Case 20 Time 0.0019		
ax 63Case 19 Time 0.0018 ax 64Case 20 Time 0.0019 ax 65Case 21 Time 0.002 ax 65Case 21 Time 0.002		
tax 63Case 19 Time 0.0018 tax 64Case 20 Time 0.0019 tax 65Case 21 Time 0.002 tax 66Case 22 Time 0.0021 tax 67Case 23 Time 0.0022		ОК
tax 63Case 19 Time 0.0018 tax 64Case 20 Time 0.0019 tax 65Case 21 Time 0.002 tax 66Case 22 Time 0.0021 tax 67Case 23 Time 0.0022 tax 68Case 24 Time 0.0024 tax 68Case 25 Time 0.0024		QK



With the envelope technique, one graphic can say it all.



10. QUESTIONS AND ANSWERS ABOUT FREQUENCY ANALYSIS

Question: What happens when a structure is loaded by harmonic load that is below the structures lowest natural frequency?

Answer: Let's say that we have a transmission where the motor has an operating speed of 1,800 RPM (30 Hz). The transmission's first natural frequency is 36 Hz (20% margin since we don't really trust our FEA results). The transmission is stable and the applied load has a magnitude effect equal to that of a static load.

Question: I have a very small natural frequency number (i.e., <<0.1), what happened?

Answer: Well, most likely you have something not constrained and NX Nastran is telling you that you have a rigid body motion. If one animates this frequency, one will see the complete model moving. Note: A structure that has no constraints or a constraint set attached to the solution, will have six low-number natural frequencies and likewise, if you have a part within your model that is not attached, it will exhibit a low frequency mode (rigid body motion). This is a super effective trick to find lose parts in your model that would cause a static stress analysis run to fail.
11. BEING AN EXPERT: VIBRATION IS ABOUT MASS AND CONSTRAINTS

11.1 CHECK FO6 FOR MASS SUMMATION AND KNOW WHAT YOU KNOW

Although this is just another check, we wanted to let you guys know

FO6 Check-Out Basics

- Do the element types and numbers make sense?
- Does the model mass exactly match that reported in the "OUTPUT FROM GRID POINT WEIGHT GENERATOR"
- Error and Warning Messages?

Vibration Analysis White Paper



11.2 GROUND CHECK IF YOU ARE DOING AEROSPACE QUALITY WORK

This check-out technique provides a numerical proof that your stiffness matrix is up-to-snuff. It is a rather dry subject and we'll leave it up to the seminar to flesh-out exactly how to do Ground Check, but if you have ever wondered what this screen does – this is your opportunity.

Model Check					-	X
Weight Check			Ground Ch	neck		
DOF SET	G	<u> </u>	DOF SET	🗖 <u>G</u>		E
E	<u>N</u>	Δ		<u>N</u>		A
	N+AUTOSPC	<u>v</u>		N+A	UTOSPC	
					Print Forces A	bove
CGI (Cente	er of Gravity)			EC	10	%
Re <u>f</u> Node	0		Ref Nod <u>e</u>		0	
Units	0We	eight 👻	Max Strajn	i Energy	0.	
Prev	. Next			<u>О</u> К	Canc	el

For more information see our User Guide: What is Groundcheck?



12. RANDOM VIBRATION CONCLUSION

The topic of Random Vibration is complex. What is presented here is a brief introduction to the theory and implementation of the subject. It is suggested that the user read a bit of the documentation provided on this subject within the NX Nastran library that is installed with every license of FEMAP & NX Nastran.

For a lot of FEA work, a straightforward recipe to accomplish your analysis task is seldom available and if it does, could easily lead you down the wrong path. Thus, I'm fond of saying that nothing beats having a good theoretical understanding of what you are doing and being highly suspicious of any result generated in "color". Or as I have read "Computer models are to be used but not necessarily believed."



13. INTRODUCTION TO RESPONSE SPECTRUM ANALYSIS

Response spectrum analysis is widely used for the design and assessment of structures that are subject to earthquakes or shock events. The reason we want to use a response spectrum is that it allows us to analyze transient events (time based events) without having to review hundreds or thousands of results sets. In essence, it allows us to assess the maximum dynamic response (stress, acceleration, velocity or displacement) of a structure using a very simple analysis technique (normal modes). Moreover all of this goodness can be had by only having to interrogate one (1) output set.

This tutorial will walk you through the theoretical background of response spectrum analysis and how to actually implement it within FEMAP & NX Nastran.



13.1 THE ACCELEROGRAM

The whole key to a response spectrum analysis is generating the response spectrum that is used to drive the FEA model. A response spectrum is the "load" to the FEA model that allows us to reduce a complicated transient analysis (time based) to simple normal modes analysis (frequency based). In general terms, response spectrums are generated from acceleration versus time measurements or accelerograms. An example of an accelerogram is shown below for an earthquake event. The three traces (Up, East and South) were generated by a tri-axial accelerometer mounted onto a concrete foundation. Similar traces can be obtained during shock events (e.g., rocket launches).

A response spectrum (an example is shown on the right) is created in a somewhat non-intuitive manner from an accelerogram. The conversion from time based data into a response spectrum is mathematically complex (Note: it is not a Fast Fourier Transform (FFT)). In the next section we will give an equation-less explanation of this mathematical procedure.



Accelerogram image courtesy of Charles Ammon, Penn State), http://earthquake.usgs.gov/learning/glossary.php?term=accelerogram

The response spectrum image is courtesy of the U.S. Taxpayers.

13.2 CREATING A RESPONSE SPECTRUM

As we mentioned before, converting an accelerogram (Time domain) to a response spectrum (frequency domain) is not done with a FFT. Let's explore this procedure using a simple example. We want to create a response spectrum that goes from 0 to 10 Hz using our acceleration versus time history data (i.e., accelerogram).



AppliedCAx.com / Predictive Engineering - Please share with your friends and visit us online at www.AppliedCAx.com

Page 78 of 123



To account for system damping (which exists to some degree in all structures), we can attach dampers to our springs and rerun the shaker table experiment. As you might expect, another curve is generated that follows the exact same profile but with lower acceleration values. Below is a classic example of a family of response spectrum curves for 1%, 2.5% and 5% of critical damping.







13.3 NX NASTRAN METHOD

Nastran starts off the response spectrum analysis by calculating the normal modes of the system and then the modal participation factors (PF). We seek the solution to;

$$([K] - \lambda [M])[\emptyset] = 0, \lambda = \omega^2$$

Which is an eigenvalue problem, the non-trivial solution to the problem is found from;

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

This gives non-zero eigenvectors $[\emptyset] \neq 0$. The response spectrum is used in conjunction with the PF's and a modal combination method to determine the system response to the spectrum. The PF's are calculated by;

$PF = [\emptyset]^T [M] [I]$

Where $[\emptyset]$ is the mode shape vector, M is the mass matrix and I is a unit vector of the same dimensions as M.

What we are saying is that each normal mode shape has a bit of mass associated with it. To generate displacements and stresses within the structure, each mode shape (and its PF) is factored against the response spectrum at that particular frequency. That is to say, a standard linear stress analysis is performed at each normal mode submitted in the response spectrum analysis. After all of the requested mode shapes have been processed, the resulting displacements and stresses are summed up using some sort of combination method (see next slide) and then written out into one final result set for subsequent post-processing.



As we had mentioned, at the end of the analysis, the individual results set generated at each mode are summed up. An idea of some of the methods that are available:

- 1. ABS This combination method is the most conservative, the absolute value resulting from each mode are summed.
- 2. SRSS Square root of the sum of the squares, also contains a provision to use the ABS method for mode combination that are within a user specified closeness (default is 1.0).
- 3. NRL U.S. Navy Shock Design Modal Summation Convention, this method combines the SRSS and the ABS methods
- 4. NRLO Updated NRL method to comply with NAVSEA-0908-LP-000-3010 specification.

The choice of the combination method is dependent on the analysis, the SRSS method is generally the method of choice since it includes the provision for using the ABS method for close modes.

14. RESPONSE SPECTRUM ANALYSIS EXAMPLE: CANTILEVER BEAM

Doing a Response Spectrum Analysis in FEMAP:

- 1. Creating the FE model
- 2. Defining the response spectrum or spectrums
- 3. Creating the interpolation table
- 4. Creating a modal damping function
- 5. Creating the excitation node and tying it into the model
- 6. Constraining the model
- 7. Setting up the analysis in the analysis manager
- 8. Post-Processing the results



14.1 PROBLEM DEFINITION

A cantilevered beam six inches in length is used to support a mass of 1.0 [lbs \cdot s2/in]. Our objective is to determine the dynamic stresses of the beam for vibration along the vertical axis.

The FEA model is a single beam element. A picture of the beam element, with its cross section displayed is shown on the right.

We will compare the FEA results to an analytical solution.

Our unit system is lb/in/s and

1 g = 386 in/s2. The elastic modulus of steel is taken as 2.9E+07 psi.



14.2 ANALYTICAL SOLUTION

The stiffness of the cantilever beam can be determined from the Young's modulus for steel, the moment of inertia and the length of the beam:

$$k = \frac{3EI}{L^3} = \frac{3 * 2.9E7 * (\frac{2 * 0.25^3}{12})}{6^3} = 1048.9\frac{lbs}{in}$$

With the stiffness and the weight, the circular natural frequency can be determined:

$$\omega_n = \sqrt{\frac{k}{m}} = \sqrt{\frac{1048.9}{1.0}} = 32.387 \; \frac{rad}{s}$$

And the natural frequency can be found:

$$f_n = \frac{\omega_n}{2\pi} = \frac{32.387}{2\pi} = 5.1545 \, Hz$$

We will determine the systems response to a simple response spectrum, to highlight the abilities of Nastran, we will use a damping value of 3%, this value is intermediary between the given response spectrum curves (just like a standard engineering problem, i.e., we never get exactly what we want for an analysis). This is no big deal since NX Nastran can interpolate between the given damping values.

Values for the problem have been chosen in order to provide a clean and straight forward solution. Since the beam model is a 1 degree of freedom system, we know that the modal participation factor (PF) for the above calculated natural frequency will be 1.0 (since all the mass of the system is at the only mode that the system has – never in real life but great for this example).





5.155 Hz

From the value that we previously determined from the natural frequency (5.155 Hz), using the response spectrum shown below, we can see that the response will be 1.2 and 1.0 G's. The system is linear, and as such linear interpolation can be used to determine the system response with 3% damping.

- Response with 1% damping (Sa) 1.2 G's
- Response with 3% damping (Sa) 1.1 G's (using linear interpolation) = 1.1*386 in/s2 = 424.6 in/s2
- Response with 5% damping (Sa) 1 G



Knowing that the spectral acceleration (Sa) is 1.1 G's or 424.6 in/s (the response spectrum acceleration at the normal mode frequency), we can find the spectral velocity (maximum velocity at end of the beam) and the spectral displacement (the maximum displacement at the end of the beam).

$$S_{\nu} = \frac{S_a}{\omega_n} = \frac{424.6\frac{in}{s^2}}{32.38\frac{1}{s}} = 13.11\frac{in}{s}$$
$$S_d = \frac{S_a}{{\omega_n}^2} = \frac{424.6\frac{in}{s^2}}{32.38^2\frac{1}{s}} = 0.405 \text{ in}$$

The dynamic force can be determined from the weight at the end of the beam and the maximum acceleration (Samax). Then the dynamic stresses can be found;

$$F = ma$$

$$F_{dyn_{max}} = m * S_a = 1.0 \frac{lbs * s^2}{in} * 424.6 \frac{in}{s^2} = 424.6 lbs$$

Finally the dynamic stress is given by;

$$\sigma = \frac{Mc}{I} = \frac{(424.6 * 6) * (\frac{0.25}{2})}{(\frac{2 * 0.25^3}{12})} = 122,285 \text{ psi}$$

14.3 STEP 1: CREATING THE FE MODEL

Create a beam element with the cross section as shown. The length of the beam is 6 in.

Add a point mass of 1.0 at one end of the beam. Add a large point mass of 10e6 to the end of the beam where the excitation will later be applied.

The cross section of the beam is chosen to be thin along the bending direction in order to minimize shear effects, this provides very close agreement between the FE results and the analytical results.

For this type of analysis, no load set is required, the analysis is performed by creating response spectrum, interpolation and damping functions. Then everything is tied together in the Analysis Manager as a Normal Modes/Eigenvalue analysis



14.4 STEP **2**: DEFINING THE RESPONSE SPECTRUM

A total of 2 functions will be created to define the response spectrum for the analysis, they are Type 3: vs. Frequency. One will be created for the 5% damping curve (Function ID 1)

and another for the 1% damping curve (Function ID 34). The curves are plotted below. Any number of response spectrums can be defined, depending on the requirements of the design. For this example, we will be using 3% of critical damping. The user can choose from Q damping, structural damping or critical damping







AppliedCAx.com / Predictive Engineering - Please share with your friends and visit us online at www.AppliedCAx.com

14.5 STEP **3**: CREATING INTERPOLATION TABLE

Nastran requires an interpolation table to determine the system response, if the system damping is not given directly from a response spectrum curve. IMPORTANT, the interpolation table is mandatory, even if no interpolation is required between curves.

A function is created that assigns damping values to the previously created functions.

The function is Type 16..Function vs. Critical Damp. We will give the function a title of "Acceleration Response Spectrum"

A value of 0.05 (5% of critical damping) is assigned to function ID 1 and a value of 0.01 (1% or critical damping) is assigned to function 2. In this manner, the system response to a damping value other than the damping associated to the defined response spectrum curves can be calculated via interpolation between the defined curves.

E Function Defin	nition				– 🗆 X
ID 3	Title Acceleration Re	sponse Spectrum		Type 16Function vs (Critical Damp 🗸 🗸
X - Function	Y - Frac Crit			X Axis Log Scale	Y Axis Log Scale
1 2	0.05	Frac Crit 05 046 042 038 034 033 034 035 026 022 018 014 1. 1. 1. 1. 1. 1. 1. 1. 1. 1.	1.3 1.4	, i, i 15 16 17	, , 18 19 Find
Data Entry	◯ Edit P <u>h</u> ase (X)		Add	Copy Function	Get Data Series Data
O Linear <u>R</u> amp	O Edit Magnitude	(Y) Delta X 1	Undate	Load from Library	Paste from Clipboard
O Eguation		X Varia <u>b</u> le x	Dhance	Save to Library	Copy to Cliphoard
ID	Y		Delete	Save to Library	Copy to Clipboard
10	-				

14.6 STEP **4**: CREATING A MODAL DAMPING FUNCTION

The system damping can be defined as a function of frequency; we will use a 3% of critical damping across the frequency range.

The damping function is defined as Type 7..Critical Damp vs. Freq. We will give the function a title of "Damping"

The function is used in association with the interpolation table. Since we have chosen a modal damping value of 3%, the system response will be found in the response spectrum between the 5% damping and the 1% damping curves. Note that if the system response at 5% damping was desired, the interpolation function previously created would only need to be defined for the function ID 1, and the damping function would then be defined as 5% over the frequency range.

E Function Defi	nition				- 0	×
ID 4	Title Response Spec	trum Damping		Type 7Critical Damp	o vs. Freq	~
X - Frequency	Y - Frac Crit			X Axis Log Scale	Y Axis Log	Scale
0. 1000.	0.03 0.03	FracCott .067 - .054 - .048 - .042 - .036 - .033 -				
		.024 - .018 - .012 - .006 - .0. 100. 200.	300. 400.	1 1 1 500. 600. 700.	800. 90	D. 100 Frequen
Data Entry	0		300. 400.	500. 600. 700.	800. 90), 100 Frequent
Data Entry ● Single <u>V</u> alue	Edit Phase (X)		300. 400. Add	500. 600. 700.	800. 90 Get Data Ser	0, 100 Frequenc
Data Entry Single Value Linear Ramp	○ Edit Phase (X) ○ Edit Mag <u>n</u> itude	(Y) Delta X 1	300. 400. Add	500. 600. 700. Copy Function Load from Library	Get Data Ser Paste from C	0. 100 Frequen Ties Data
Data Entry Single Value Linear Ramp Equation X 1000	 Edit Phase (X) Edit Magnitude Periodic Y03 	(Y) Delta X 1 X Variable X	300. 400. Add Update	500. 600. 700. Copy Function Load from Library Save to Library	Get Data Ser Paste from C Copy to Cli	0. 100 Frequent ies Data Ilipboard pboard



14.7 STEP **5** CREATING THE EXCITATION NODE

For the response spectrum analysis, a foundation mass must be added to the excitation node. The mass should be 103 to 106 times the mass of the system.

Since this is a base excitation problem, and the base of the structure consists of one node, it is that node to which we will apply our acceleration response spectrum. In the case where the base of the structure is not one node, a rigid link approach is used to tie the multiple nodes of the base to a single node.



Base Node

14.8 STEP **6**: CONSTRAINING THE MODEL

In order to perform the response spectrum analysis, 2 constraint sets must be created.

The first set will constrain the model in all degrees of freedom except the excitation direction (in this example, y-direction).

The second constraint set will be used as a kinematics DOF. This is known as a SUPPORT set in Nastran. The DOF that the excitation is in is constrained.

	Constra	sint Definitio	n				
Constrair	t Set	1 Sei	smic Constra	int			
Title					Coord Sys 413	Use Nodal Outpu	it System
Color	120	Palette	Layer	1			
DOF					X Symmetry	X AntiSym	NonZero Constraint >>
⊡TX	TY	⊡TZ	Fixed	Pigned	V Europeter	V Automation	1120
RX	RY	⊡ RZ	Eree .	No Rotation	r gymneu y	T Anguayin	QK
					Z Symmetry	Z AntiSym	Cancel
Editing Constrain	Constra	int Definitio	n smic Kinema	tic Constraint			>
Editing Constrain Title	Constra It Set	aint Definitio 2 Sei	n smic Kinema	tic Constraint	Court Sys -1.1	Use Nodel Outpu	t System -
Editing Constrain Title Color	Constra It Set	aint Definitio 2 Sei Palette	n smic Kinema Løyer	tic Constraint	Coset Sys1.1	Use Nodel Outpu	t System -
Editing Constrain Title Color DOF	Constru t Set 120	aint Definitio 2 Sei Palette	n smic Kinema Layer	tic Constraint	Court Sys -1.1	Use Nodel Output	t System
Editing Constrain Title Color DOF	Constru nt Set	Paletta	n smic Kinema Layer Fjored	tic Constraint	Coord Sys -1.	Use Nodel Outpo X AntrSym	t System
Editing Constrain Title Color DOF DDF	Constru et Set 120 DTY RY	Palette	n smic Kinema Layer Fixed Eree	tic Constraint 1 Poned No Rotation	Court Sys -1.J X Symmetry Y Symmetry	Use Nodal Outpo X AntiSym Y AntiSym	t System Co NonZero Constraint >> OK





14.9 STEP **7**: SETTING UP THE ANALYSIS

Set up a new analysis set as type 2Normal Modes/Eigenvalue.	Analysis Set Manager (Active: 1Random Spectr – Analysis Set : 1Random Spectrum (shock spectrum) Solver : NX Nastran Type : Normal Modes/Eigenvalue Integrated Solver : NX Nastran Options Synort
	No Case
	Analysis Set Title Random Spectrum (shock spectrum) Analysis Program 36NX Nastran Analysis Type 2Normal Modes/Eigenvalue Solve Using Integrated Solver C Linked Solver Solver is undefined. Go to File Preferences Solvers. VisQ Next OK Cancel Edit
	Done

Click Next button 5 times, in the Range of Interest,	NASTRAN Modal Analysis		×	
type in;	Skip EIGx	Method I	D 1	
	Real Solution Methods	Range of Interest		
1.0 in the From (Hz) field	Lanc <u>z</u> os	Real	Imaginary	
	Auto (HOU/MHOU)			
	◯ S <u>u</u> bspace	From (Hz)	0.	
50 0 in the Te (Hz) field	Legacy Real Solution Methods	<u>To (Hz)</u> 50.0	0.	
50.0 III the TO (12) held.	◯ <u>G</u> ivens	Figenvalues and Figenvectors		
	O Modified Givens	Ligenvalues and Ligenvectors		
This is very important since the first mode that will be	◯ Inverse Power	Number <u>E</u> stimated	0	
calculated will be the rigid body motion (natural	O Inverse Power/Sturm	Number Desired	0	
	Householder			
frequency of zero). If this field is left blank, the analysis	Modified Householder	Normalization Method	Mass	
will be performed from 0 Hz, this will give the incorrect	Complex Solution Matheda	• Ma <u>s</u> s _{Node ID} 0	Default	
value for the nodal velocity and displacement, and as		O Max		
such the stresses will be incorrect	Hessenberg	O Point	○ Coupled	
	Complex Inverse Power	Complex Solution Ontions		
	Complex Lanczos		0	
Note: The minimum frequency does not have to be	Solution Type	Convergence	0.	
1.0 it can be any small number such that mechanisms	Direct	Region Width	0.	
are excluded.	◯ Mo <u>d</u> al	Overall Damping (G)	0.	
	Prev Ne <u>x</u> t	<u>O</u> K	Cancel	

Press Next button once. For the modal analysis, it is very useful to ask Nastran to output the Modal Participation Factors (PF). Select Summary and Modal Effective Mass Fraction. Nastran will output the PF's to the .f06 file. This is a very important tool when analyzing larger models that have multiple DOF and many eigenvalues. The PF's are used in combining the response from the individual modes based on the combination method selected.

However, you don't need to request'em since they are automatically calculated. This toggle just dumps them out to the f06 file and creates functions in FEMAP.

NASTRAN XY Output for Modal Analysis	×
Output Requests	
Summary	
Modal Participation Factors	
Modal Effective Mass	
Modal Effective Weight	
Modal Effective Mass Fraction	
Reference <u>N</u> ode 0	
Prev <u>Nex</u> t <u>O</u> K	Cancel

Press Next button once. This is the step that ties all the functions together.

	NASTRAN Response Spectrum Application X
In the Spectrum – Spectrum Function ID drop down box, select "Acceleration Response Spectrum".	Spectrum Spectrum Function ID Acceleration
In the Scale Factor box, type in 386, this will scale the response spectrum curve from G's to actual acceleration values in in/s2.	Ovelocity Scale Factor Obisplacement 386. Modal Combination 0ABS
In the Modal Combination - Method box, select any type since only one mode is calculated in the 1.0 Hz to 50 Hz range. Here we are using ABS method, but for this example, the same results are obtained with all methods	Closeness 1.0 Base DOF SUPORT Set 2Seismic Kinetic Constraint
example, the sume results are obtained with an methods.	Modal Damping Damping Func 4Response Spectrum Dampir
In the Base DOF - SUPORT Set drop down box, select "Seismic Kinematic Constraint".	Prev <u>Next</u> <u>O</u> K Cancel
In the Modal Damping – Damping Func drop down box select "Damping"	

Page 97 of 123

	Boundary Conditions		×
	Primary Sets		
	<u>C</u> onstraints	1Seismic Constraint	~
	Loads	0None	~
	Temperatures	0From Load Set	~
Press Next button twice.	Initial Conditions	0None	~
	Constraint Equations	0From Constraint Set	~
In the Boundary Conditions – Primary Sets –	Bolt Preloads	0From Load Set	~
Constraints drop down box select "Seismic	Other DOF Sets		
Constraint".	Master (ASET)	0None	~
	Kinematic (SUPORT) >	2Seismic Kinematic Constrai	~
In the Other DOF Sets – Kinematic (SUPPORT) drop	SUPORT1	0None	~
down box select "Seismic Kinematic Constraint".	OMIT	0None	~
	QSET	0None	~
	CSET	0None	~
	BSET	0None	~
	Prev Ne <u>x</u> t	<u>O</u> K Cancel	

In the Nastran Output Request – Nodal select.	NASTRAN Output Requests					×
	Nodal			Elemental		
•		0Full Model	~	Eorce	0Full Model	× .
Displacement	Applied Load	0Full Model	~	Stress	0Full Model	~
Constraint Force	Constraint Force	0Full Model	~	Strain	0Full Model	~
	Equation Force	0Full Model	~	Strain Energy	0Full Model	×
Velocity	Force Balance	0Full Model	~	Heat Flux	0Full Model	~
Acceleration		0Full Model	~	Enthalpy	0Full Model	~
	Acceleration	0Full Model	~	Enthalpy Rate	0Full Model	~
	Kinetic Energy	0Full Model	_	Temperature	0Full Model	~
In the Nastran Output Request – Elemental select;	Temperature	0Full Model		Kinetic Energy	0Full Model	~
				Energy Loss	0Full Model	~
Christian				Fluid Pressure	0Full Model	~
Stress —	Customization				Deer	
In the Customization area:	Element Corner R	Results	Re	sults Destination	Prev	·
	Output Modes (a,b,	c THRU d)		2PostProcess Only	~	
		-	Ec	ho <u>M</u> odel	<u>O</u> k	(
Set Results Destination to	Magnitude/Phase	e 🔿 Real/Imaginary			Can	cel
	Relative Enforced	Motion Results				
"2PostProcess Only"						

Click Ok





14.10 POST PROCESSING THE RESULTS

Press F5 and select Beam Diagram from the Contour Style. Use the Post Data tool to look at the beam diagrams for the acceleration, velocity, stress etc. Below is the acceleration contour plot, note that the max acceleration is at the free end and has a value of 424.6 in/s2.



14.11 RESULTS COMPARISON

The results from the FE analysis follow very closely with the analytical results. The acceleration, velocity and displacement are exact (slight difference between analytical and FE velocity). Note that the natural frequency found in Nastran is slightly different than the analytical calculated value, this is because Nastran calculates the shear area of the beam. One can obtain the exact same results for the natural frequency in the FE model by excluding the shear area from the calculations, when this is done, the FE results match with the analytical results to 3 decimal places.

Results Comparison					
	Analytical	Nastran			
Natural Frequency	5.1545	5.1509			
Acceleration (in/s ²)	424.6	424.6			
Velocity (in/s)	13.11	13.12			
Displacement (in)	0.405	0.405			
Stress (psi)	122,285	122,293			

15. EXTRA CREDIT: SOLID MESHED BEAM

We can verify this method using a hex meshed solid beam. The beam is massless, with a point mass of 0.5lbf (1.30e-3 slugs) on the end. The beam properties are shown below:

W = 2 in W = 0.5 lbf T = 0.25 in E = 10e6 psi L_{beam} = 5 in I_{XX} = 2.6e-3 in⁴ An Acceleration vs. Frequency spectral plot was created for 1% and 5% damping, an interpolation function was created to find intermediate damping values, and a damping function was generated.





Let's first take a look at the hand calc's to show how the beam is expected to behave.

Deflection is calculated first:

$$Y_{max} = \frac{WL^3}{3EI_{xx}} = 8e - 4 in$$

Based upon this end deflection, the beam's resonant frequency can be calculated as:

$$f_n = \sqrt{\frac{1}{2\pi} \left(\frac{g}{Y_{max}}\right)} = 110.6 \, Hz$$

Spectral acceleration for this condition is

$$S_a = 1.1 * 386.09 = 424.6 \frac{in}{s^2}$$



Continuing, we can approximate the spectral velocity, displacement, and stress: Velocity:

$$S_{\nu} = \frac{S_a}{w_n} = 0.611 \frac{in}{s}$$

Displacement:

$$S_d = \frac{S_a}{w_n^2} = 8.80 * 10^{-4} in$$

Stress:

$$F_d = W * S_a = 0.55 \ lbf$$

$$Stress = \frac{Mc}{I} = \frac{(F_d L)(\frac{T}{2})}{I_{xx}} = 132 \ psi$$

AppliedCAx.com / Predictive Engineering - Please share with your friends and visit us online at www.AppliedCAx.com



First up is a normal analysis for the solid model, the first mode is at 113 Hz:

2..NX Nastran Modes Analysis Set
 2..Mode 1, 112.9961 Hz
 3..Mode 2, 845.2501 Hz
 4..Mode 3, 4466.077 Hz



With the normal modes calculated, next is a shock response analysis:



AppliedCAx.com / Predictive Engineering - Please share with your friends and visit us online at www.AppliedCAx.com
With the normal modes calculated, next is a random response analysis:



AppliedCAx.com / Predictive Engineering - Please share with your friends and visit us online at www.AppliedCAx.com



The FEA results for the solid model are not as close to the hand calculations as the beam element, but you can see the relative accuracy below:

Results Comparison			
	Analytical	Nastran	
Nautral Frequency (Hz)	110.6	113.0	
Acceleration (in/s ²)	425	425	
Velocity (in/s)	0.611	0.598	
Displacement (in)	8.42e-4	8.80e-4	
Stress (psi)	136	116	

Page 110 of 123

The error in the maximum stress of the FEA model is 12%, which seems a bit on the high side. Let's check a hand calculation of a 1G static load versus the finite element static load to explore the behavior. This beam with 1G static loading should see 120psi maximum stress based on the simple calculations.





It is interesting to compare the accuracy with the # of nodes required for analysis. In a solid element model it seems that the coarse hex mesh does not have enough definition to capture the stresses and has a relatively large error. The fine hex mesh drops the error down to 3% with just under 4,000 nodes. Meshing with a fine 10-node tetrahedral mesh gives similar accuracy but requires 26,000 nodes, a significant increase! The plate meshed beam is the closest with 0.2% error and under 900 nodes.

	Max Stress	% Difference	# of Nodes
Hand Calculation	120 psi	n/a	n/a
Coarse Hex Mesh	110	8%	236
Fine Hex Mesh	124	3%	3,782
Fine 10-node Tet Mesh	123	3%	26,274
Fine Quad Plate Mesh	119.8	0.2%	884

A summary of the findings for a 1 G static load is shown in the table below.



16. APPENDIX

16.1 FLOW CHART FROM NX NASTRAN THEORETICAL MANUAL



AppliedCAx.com / Predictive Engineering - Please share with your friends and visit us online at www.AppliedCAx.com

2019

16.2 CREATING MODAL FREQUENCY TABLE

When Nastran calculates the RMS stress value for a PSD analysis it first calculates the frequency response at each value on the Modal Frequency Table, and uses that response to calculate the stress due to PSD excitation.

The chart below shows the frequency response for a simply supported beam with the first mode at 227 Hz, and the next mode above 300 Hz. The response acceleration ramps up near the natural frequency and gradually drops off as you move away from it.





PSD analysis is statistical and the 1-sigma stress output is simply the stress the structure will likely see when subject to a specified acceleration spectrum. The random vibration solver doesn't calculate the stress at every frequency—it only solves for the stress at the frequencies specified in the Modal Frequency Table. The simplified process is that it solves for the stress at each value in the Modal Frequency Table, and then combines those stress results to give the RMS stress.

The red line in the image below shows the response at the natural frequency. The orange lines show the response at multiple points with a 10% spread from the natural frequency. As you can see, the response drops off as you move away from the natural frequency so adding more solve points, or a greater spread from the natural frequency does not improve accuracy of the results, but it does add significant computational cost.





The chart below shows Modal Frequency Tables (X-axis is frequency, Y-axis is arbitrary) with varying number of solution points and the resultant RMS stress on a simply supported beam. With a single solve point at the natural frequency it significantly overestimates the RMS stress. The default 5 points with 10% spread gives a more reasonable result, and you can see even going up to an impractical 55 points with 20% spread gives a result within a few percent of the default table configuration.



AppliedCAx.com / Predictive Engineering - Please share with your friends and visit us online at www.AppliedCAx.com

2019

16.3 AUTOCORRELATION FUNCTION

A newly supported feature in FEMAP 11.3 is the ability to generate Autocorrelation functions during random vibration analysis. The Autocorrelation Function Time Lag input is available in the NASTRAN Power Spectral Density Factors dialog as shown below:

NASTRAN Power Spectral Density Factors		
Correlation Table		
Master=>Master 1.(3) :Int1=0	Excited Subcase: Master	
	Load Set: 1PSD Excitation Node	
	Applied Subcase: Master	
	Load Set: 1PSD Excitation Node	
Edit Correlation Table		
Factor PSD Fund	ction PSD Interpolation	
Real 1. × 3PSD Function	n v 0Log Log v	
Imaginary 1. X 0None	 ✓ 0Log Log ✓ 	
Apply		
Autocorrelation Function Time Lag		
Lag Intervals 10000 Starting Lag 1.E-6 Max Lag .16		
Prev <u>OK</u> Cancel		

The three inputs available are Lag Intervals, Starting Lag, and Max Lag. These inputs do not change the way the random vibration analysis is conducted, it merely defines the autocorrelation function which will be generated in addition to the output.

Lag Intervals: How many times to chop up the time band between the starting lag and maximum lag

Starting Lag: Starting time step

Max Lag: Final time step



If we take our beam example and plot the autocorrelation function for displacement on a couple of nodes we can get a more intuitive idea of what is going on. No matter the lag time, the autocorrelation at the excitation node is very close to zero. At lag = 0, the autocorrelation for the node at the end of the beam is high while the node at the middle of the beam is lower amplitude, and follows the same sinusoidal pattern. From this plot we can infer that a small lag time results in a high autocorrelation at the beam end, and it tapers off as you increase the lag time. It is worth noting that the period of the sinusoidal response shown here is 0.009 seconds, which matches the 110 Hz natural frequency of the beam.





At this point you may be wondering how to generate the autocorrelation function in your analysis. In the first *NASTRAN Output* dialogs, select "Autocorrelation Functions" or "All". Choose the desired nodal or elemental outputs to plot, and then enter the desired Autocorrelation Function Time Lag values.

	NASTRAN Output for Random Analysis	
	Nodal Output Requests Displacement T1 T2 T3 R1 R2 R3 Valority	NASTRAN Power Spectral Density Factors
NASTRAN Output for Random Analysis X Nodal Output Requests None Power Spectral Density Functions Autocorrelation Functions Doot Mean Square Image: Constructions Power Spectral Density Functions None Power Spectral Density Functions Autocorrelation Functions Root Mean Square Image: Constructions Prev Next OK Cancel	Yelocity T1 T2 T3 R1 R2 R3 Acceleration T1 T2 T3 R1 R2 R3 Elemental Stresses Springs Axial Torsional R3 Bars Axial Torsional End A Loc 1 Loc 2 Loc 3 Loc 4 End A Loc 1 Loc 2 Loc 3 Loc 4 End B Loc 1 Loc 2 Loc 3 Loc 4 Beams End A Loc 1 Loc 2 Loc 3 Loc 4 Beams End A Loc 1 Loc 2 Loc 3 Loc 4 Plates Bottom X Normal Y Normal XY Shear Top X Normal Y Normal Z Normal XY Shear YZ Shear ZX Shear Axisym Radial Axim Shear Forces Summary Data Only PSDF AUTO	Correlation Table Excited Subcase: Master Master=>Master 1.(3) :Int1=0 Excited Subcase: Master Load Set: 1PSD Excitation Node Applied Subcase: Master Load Set: 1PSD Excitation Node Edit Correlation Table Factor PSD Function Real 1. x 3PSD Function Imaginary 1. X 0None Apply Autocorrelation Function Time Lag Lag Intervals 10000 Starting Lag 1.E-6 Max Lag .16
	Prev Ne <u>x</u> t <u>O</u> K Cancel	



2019

16.4 MULTIPLE EXCITATION SPECTRUMS

In some cases you may need to analyze a system with multiple excitation spectrums. This can be due to a difference in structure mounting points, wheels driving on different surfaces, or an array of other situations. In this example we will take a look at a hypothetical red wagon, where the wheels on the –Z side are on a smooth section of road, and the -Z side is rolling on over a rougher road.



First the boundary conditions must be set up as shown—with your excitation points defined in separate load sets and all of the constraints in one Constraint Set.



With the boundary conditions set up, prepare the random analysis in the usual manner, but do not specify boundary conditions. Instead, specify the boundary conditions as subcases in the Analysis Set Manager.

Boundary Conditions		×
Primary Sets <u>C</u> onstraints	0None	~
<u>L</u> oads	0None	~
Temperatures Initial Conditions	0From Load Set 0None	~
Constraint <u>E</u> quations Bolt Preloads	0From Constraint Set 0From Load Set	~
Other DOF Sets	0 Nano	
Kinematic (SUPORT)	0None	~
SUPORT1	0None	~
OMIT	0None	~
<u>Q</u> SET	0None	~
CSET	0None	~
<u>B</u> SET	0None	~
Prev Ne <u>x</u> t	<u>Q</u> K Can	icel

After that is set up, go back to edit the PSD Factors tab. Here you will notice it looks a little different than the previous method. You can now choose PSD Functions for each subcase, and you can correlate the two sub cases for coupled analysis if desired. If you do not wish to correlate the sub cases, leave the settings at their default values.

NASTRAN Power Spectral Density Factor	s X	
Correlation Table 1=>1 1.(3) :Int1=0 1=>2 1.(4) + 1.(5)i :Int1=0 :Int2=0 2=>2 1.(5) :Int1=0	Excited Subcase: 1 Load Set: 2Z Side Excitation Applied Subcase: 2 Load Set: 1+Z Side Excitation	
Edit Correlation Table Factor PSD Funct Real 1. x 42x PSD Funct Imaginary 1. X 54x PSD Funct	tion PSD Interpolation tion OLog Log	
Apply Autocorrelation Function Time Lag Lag Intervals 0 Starting Lag 0. Max Lag 0. Prev Next QK Cancel		

2019



16.5 WHY WE DO A PSD ANALYSIS

Dynamics are tricky. Structures that seem sturdy intuitively may have unexpected responses when excited dynamically. The images below show a circuit board which was subjected to an intense PSD spectrum where the assembly was expected to see accelerations near 700 g's! In this case, the circuit board was not designed for that level of excitation and the chips removed themselves from the board during testing.









Siemens PLM Software for the USA

AppliedCAx.com

FEMAP & STAR-CCM+ LICENSES I TRAINING I SUPPORT

CAE Support • CAD Workflows • CAM Posts • PLM Architecture



FEMAP • NX CAD • NX CAM • Solid Edge • Simcenter 3D • STAR-CCM+ • Teamcenter