

PSD Random Vibration Analysis in FEMAP

An Introduction to the Hows and Whys of PSD Adrian Jensen, PE | Senior Staff Mechanical Engineer Kyle Hamilton | Staff Mechanical Engineer



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.	11	NTRODUCTION	4
2.	Т	HE PSD FUNCTION	5
3.	Т	HE NX NASTRAN METHOD	7
4.	Ρ	SD UNITS	8
5.	E	XAMPLE 1: CANTILEVER BEAM	9
5	1	PROBLEM DEFINITION	.9
5	2	ANALYTICAL SOLUTION	10
5	3	Defining the System Damping	11
5	4	CREATING THE PSD FUNCTION	12
5	5	CREATING THE MODAL FREQUENCY TABLE/SETTING UP THE LOAD SET OPTIONS FOR DYNAMIC ANALYSIS	13
5	6	LOADING & CONSTRAINING THE MODEL	15
5	7	Specifying Groups for Nodal and Elemental Output	16
5	8	CREATING AN ANALYSIS SET – SIMPLE PSD	17
5	9	INTERPRETING THE OUTPUT	24
5	10	Positive Crossings	25
5	11	Fatigue Analysis Using RMS Stress and Positive Crossings	27
5	12	Fatigue Analysis – Time to Failure	28
6.	E	XAMPLE 2: SOLID MESHED BEAM	29
6	1	ANALYTICAL SOLUTION	30
6	2	PSD FUNCTION INPUT	31
6	3	PSD Stress Results	32
6	4	COMPARING MILES' APPROXIMATION AND PSD RESULTS	33





7.	С	ONCLUSION	. 34
8.	R	ECOMMENDED READINGS	. 34
9.	А	PPENDIX	. 35
9	.1	FLOW CHART FROM NX NASTRAN THEORETICAL MANUAL	35
9	.2	CREATING MODAL FREQUENCY TABLE	36
9	.3	Autocorrelation Function	39
9	.4	MULTIPLE EXCITATION SPECTRUMS	42
9	.5	Why we do a PSD Analysis	45





1. INTRODUCTION

Random vibration is vibration which can only be described in a statistical sense. The magnitude at any given moment is not known, 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 random vibrations are usually described in terms of a power spectral density (PSD) function.







2. 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:



For more of the nitty gritty math details see NASA's webpage on random vibration here: <u>https://femci.gsfc.nasa.gov/random/</u>



Applied CAx / Predictive Engineering White Paper

Please share with your friends and visit us online



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.

Image By Dan Kernler (Own work) [CC BY-SA 4.0 (http://creativecommons.org/licenses/by-sa/4.0)], via Wikimedia Commons





2017

3. 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 systems 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 upon 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





4. 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	m	Ра
SI (mm,Mg,sec)	6.89e4 MPa	2.71e-9 Mg/mm ³	1 g²/Hz	9807 mm/s ²	mm	MPa
Imperial (in, snail, sec)	10.0 e6 psi	2.54e-4 snail/in ³	1 g²/Hz	386.1 in/s ²	in	psi



5. EXAMPLE 1: CANTILEVER BEAM

5.1 **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 (PSD_{IN}) function used by Steinberg was

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

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/s^2 .



Steinberg, Dave S. Vibration Analysis for Electronic Equipment. 2nd ed. New York: John Wiley & Sons, 1988. 226-231.





5.2 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 G_{out}(RMS) value:

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

This output 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/s²

In English units, the max $PSD_{out} = 13.14e6 \text{ in}^2/\text{s}^4$. This can also be verified against the FE Model.

Note: When approximating transmissibility (Q), the square root of the natural frequency should be scaled by 0.5 – 2 per Steinberg. In this case there is 100% mass participation for the first mode, thus 2 is appropriate.



5.3 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.

Function Defi	nition									-		×
ID 2	Title Damping					1	ype 7	Critical D	amp v	s. Freq		~
X - Frequency	Y - Frac Crit						X Axis L	og Scale		Y Axis	s Log Sca	le
). I.	0.0238	Frac Crit .0476 - .0357 - .02975 - .0238 - .01785 - .0119 - .00595 -										
		0.	0.1	0.2	0.3	0.4	0.5	0.6	0.7	0.8	0.9 Free	quen
Data Entry		0.	0.1	0.2	0.3	0.4	0.5	0.6	0.7	0.8	0.9 Free	quen
Data Entry ● Single <u>V</u> alue	◯ Edit P <u>h</u> ase (X)	0.	0.1	0.2	0.3 Add	0.4	0.5 <u>C</u> opy Fr	0.6 unction	0.7	0.8 Get Da	0.9 Free ata Series	quen Data
Data Entry ● Single <u>V</u> alue ○ Linear <u>R</u> amp	◯ Edit P <u>h</u> ase (X) ◯ Edit Mag <u>n</u> itude	0. + 0 0 0 0 0 0 0	0.1	0.2	0.3 Add	0.4	0.5 Copy Fi	0.6 unction n Library	0.7	0.8 Get Da	0.9 Free ata Series from Clip	Data
Data Entry Single Value Linear Ramp Equation	 ○ Edit Phase (X) ○ Edit Magnitude ○ Periodic 	(Y) Delta X Variable	0.1 × 1 ×	0.2	0.3 Add	0.4	0.5 Copy Fi	0.6 unction n Library	0.7	0.8 Get Da Paste f	0.9 Free ata Series from Clip	Data board
Data Entry Single Value Linear Ramp Equation X 1.	 Edit Phase (X) Edit Magnitude Periodic Y 0.0234 	(Y) Delta X Variable	0.1 × 1 ×	0.2	0.3 Add Update Delete	0.4	0.5 Copy Fi Load fror Save to	0.6 unction m Library Library	0.7	0.8 Get Da Paste f	0.9 Free ata Series from Clip	Data board

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.





5.4 CREATING THE PSD FUNCTION

The input to the cantilever beam is a whitenoise vibration with a PSD input of 0.20 G^2/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).

E Function Defi	nition				— D	Х
ID 3	Title PSD Function			Type 3vs. Frequency	1	~
X - Frequency	Y - Factor			X Axis Log Scale	Y Axis Log Scale	е
10. 2000.	0.2 0.2	Factor .4 .36 .32 .28 .24 .2 .16 .12 .08 .04 .0 .10. .209. .408.		1 I I I 1005. 1204. 1403.		
						2000 requency
Data Entry	O Edit Phase (X)		٨،٩٩	Conv. Eurotian	Cat Data Sarias I	2000 requency
Data Entry Single Value	◯ Edit P <u>h</u> ase (X) ◯ Edit Magnitude	Y) Delta X 1	Add	Copy Function	Get Data Series I	2000 requency Data
Data Entry Single Value Linear Ramp Equation	◯ Edit P <u>h</u> ase (X) ◯ Edit Ma <u>gn</u> itude (◯ Periodic	(Y) Delta X 1 X Variable X	Add <u>U</u> pdate	Copy Function Load from Library	Get Data Series I Paste from Clipbo	Data
Data Entry Single <u>Value</u> Linear <u>Ramp</u> Equation X	C Edit Phase (X) Edit Magnitude (Periodic	(Y) Delta X 1 X Varia <u>b</u> le X	Add Update Delete	Copy Function Load from Library Save to Library	Get Data Series I Paste from Clipbo Copy to Clipboa	Data oard



5.5 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						
<u>T</u> itle <u>Eigenvalue</u>						
Analysis <u>P</u> rogram	36NX Nastran					
<u>A</u> nalysis Type	2Normal Modes/E	igenvalue v				
	Run Analysis Usi	ng VisQ				
Ne <u>x</u> t	<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.





5.6 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.

The Acceleration on Not	le		(Coord Sys 0.	.Basic Rectangular	
Color 10 Palette	Layer 1					
Acceleration	Direction Comp Vector Along Norm Norm	oonents or j Curve al to Plane al to Surface	Specify		Method Constant Variable Data Surface Adva	e nced
	Load	Value	Time/Freg Depend	lence	Data Surf	ace
	AX [✓ 0.	0None	~ f _{xy}	0None	~
	AY [386.			0None	\sim
	AZ [✓ 0.			0None	~

The **Load Constraint** constrains the base node in all six 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 Y-direction).

Create Nodal Constraints/DOF X								
Constraint Set 1 Load Constraint								
Title Coord Sys 0Basic Rectangular ~								
Color 120 Palette	Color 120 Palette Layer 1							
	The d	Discust	X Symmetry	X <u>A</u> ntiSym	01/			
	Fixed	Pi <u>n</u> ned	Y Symmetry	Y An <u>t</u> iSym	<u>U</u> K			
	Free	No <u>R</u> otation	Z Symmetry	Z AntiSy <u>m</u>	Cancel			





5.7 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.





5.8 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	×							
Iitle PSD Analysis								
Analysis Program 36NX Nastran								
Analysis Type 6Random Response								
Solve Using								
 Integrated Solver 								
○ Linked Solver								
Solver is undefine	ed. Go to File Preferences Solvers.							
⊖⊻isQ	⊖⊻isQ							
Ne <u>x</u> t	QK 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 X						
Skip EIGx	Range of Interest	Method ID	1			
	in inge er sineret	Real	Imaginary			
	F <u>r</u> om (Hz)	0.	0.			
Legacy Real Solution Methods	<u>T</u> o (Hz)	2000.	0.			
○ <u>G</u> ivens ○ Modified Givens	Eigenvalues and Eig	envectors	-			
	Number <u>E</u> stimated		0			
O Inverse Power/Sturm	Number Desired	0				
	Normalization Metho	bd	Mass			
O Modified Householder	Mass	0	Default			
Complex Solution Methods		0				
OHessenberg		0				
○ Complex Inverse Power	O Forme		Occupied			
O Complex Lanczos	Complex Solution Op	otions				
Solution Type	Convergence		0.			
	Region Width		0.			
Modal	Overall Damping (O	5)	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.

NASTRAN Dynamic A	Analysis			×
Use Load Set Optio	ons		Frequency ID	2
Options for Dynamic A	Analysis Solution	Frequencies		
Equivalent Viscous Da	mping	Limit Response Based	on Modes	
Overall Structural Da	amping Coeff (G)	0.	Number of Modes	0
Modal Damping	2Damping	~ 5 xy	Lowest Freq (Hz)	0.
As Structural	(KDAMP)		Highest Freq (Hz)	2000.
Equivalent Viscous Da	mping Conversion		Transient Time Step 1	ntervais
Convert using Sol	ution Freq (WMOE	DAL)	Number of	0
🗌 Rigid Body Zero N	Modes(FZERO)	1.E-4	Time per	0.
Freq for System Dar	nping (W3 -	0.	Output Interval	0
Freq for Element Da	mping (W4 -	0.		
Frequency Response			Response/Shock Spec	trum
Frequencies 4Mo	dal Frequency Tab	le v 😡	Damping/Freq Corre	lation
Modal Freq			0None	~ f _{av}
- Houdi Hoqiii				
_	•		01/	
Prev	Ne <u>x</u> t		<u>O</u> K	Cancel
	Frequency Table	From Modal Resu	lts X	
	Modal Results			
	Eirst Freq	1Mode 1, 110.4497	′Hz ~	
	<u>L</u> ast Freq	1Mode 1, 110.4497	Hz ~	
	Additional Solutio	n Frequency Points		
	Number of Poir	nts per Existing Mode	5	
	Frequency Ban	id <u>S</u> pread (+/-)	10. %	
		<u>o</u> k c	Cancel	





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 fo	r Modal Analysi	is X			
Output Requests						
Summar	Y					
Modal P	articipation F	actors				
Modal Effective Mass						
Modal Effective Weight						
Modal E	ffective Mass	s Fraction				
Reference	Node 0					
Prev	Ne <u>x</u> t	<u>O</u> K	Cancel			
NASTRAN Output for Random Analysis X Nodal Output Requests None Power Spectral Density Functions Autocorrelation Functions Root Mean Square All						
Elemental Output Requests None Power Spectral Density Functions Autocorrelation Functions Root Mean Square All						
Prev	Ne <u>x</u> t	<u>O</u> K	Cancel			





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-σ 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				
Nodal Output Requests				
Displacement T1 Velocity T1 Acceleration T1	T2 T3 R1 R2 R3 T2 T3 R1 R2 R3 T2 T3 R1 R2 R3 T2 T3 R1 R2 R3			
Elemental Stresses				
Springs Axial				
Rods Axial	Torsional			
Bars Axial				
End A Loc 1 End B Loc 1	Loc 2 Loc 3 Loc 4			
Beams End A Loc 1 End B Loc 1	Loc 2 Loc 3 Loc 4			
Plates Bottom □ X Nor Top □ X Nor	rmal Y Normal XY Shear rmal Y Normal XY Shear			
Solids X Not	rmal Y Normal Z Normal hear YZ Shear ZX Shear			
Axisym 🗌 Radia	al 🗌 Azim 🗌 Axial 🗌 Shear			
Forces				
Summary Data Only				
Prev Ne <u>x</u> t	<u>O</u> K Cancel			





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

NASTRAN Power Spectral Density Factors				
Correlation Table				
Master=>Master 1.(3) :Int1=0	Excited Subcase: Master			
	Load Set: 1PSD Load			
	Applied Subcase: Master			
	Load Set: 1PSD Load			
Edit Correlation Table				
Factor PSD Fund	ction PSD Interpolation			
Real 1. x 3PSD Function	 ✓ 0Log Log ✓ 			
Imaginary 1. X 0None	✓ 0Log Log ✓			
Apply				
Autocorrelation Function Time Lag				
Lag Intervals 0 Starting	Lag 0. Max Lag 0.			
Prev Ne <u>x</u> t	<u>O</u> K Cancel			

Choose your constraint set and load created for the PSD analysis

Boundary Conditions	X
Primary Sets	
<u>C</u> onstraints	1Fixed ~
<u>L</u> oads	1PSD Excitation Node <->
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 ~
<u>B</u> SET	0None
Prev Ne <u>x</u> t	<u>Q</u> K Cancel



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

NASTRAN Output Re	equests				×
Nodal			Elemental		
Displacement	0Full Model	\sim	Eorce	0Full	Model ~
Applied Load	0Full Model	\sim	⊡ <u>S</u> tress	0Full	Model ~
Constraint Force	0Full Model	~	Strain	0Full	Model ~
Equation Force	0Full Model	~	Strain Energy	0Full	Model ~
Force Balance	0Full Model	~	Heat Flux	0Full	Model ~
	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					
Element Corner Re	esults	Re	sults Destination		Prev
Output Modes (a,b,c	THRU d)		2PostProcess Only	~	
		Ec	ho <u>M</u> odel		<u>O</u> K
Magnitude/Phase	○ Real/Imaginary				Cancel
Relative Enforced	Motion Results				





5.9 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-σ PSD stress value. This represents how much stress the beam will experience 68.3% of the time.









5.10 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"



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.

Select PostP	Processing Data	a		_	×
View 1	default				Dynamic Max/Min
O <u>u</u> tput Sets	10Positive Cro	ssings	V v K	Program : NX Nastran alysis Type : Random Set Value : 0.	
Output Vecto	ors				
Deform	3139Beam En	dA Pt1 Comb Stress	× ¥ *	Transform	Vector Info
	Max : El	ement 1 110.4	009 Min :	Element 1	110.4009
Contour			✓ V _d ≪	Transform	Vector Info
	Max :		Min :		
	Double-Sideo	Planar Contours		Multiple Contor	ur Vectors
Contour	Options	Contour Arrows	Trace Locations	Expand Complex	<u>O</u> K
Laminat	e Options	Section Cut	Streamline Options		Cancel





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		×
🏚 • 📩 🚯 🖳 🛼 • 🔡 • 🗆 • 🛃 🍏		
Deform		×
Contour		×
Style	Contour	\sim
Results		
Output Set	17Positive Crossings 🔹 🖲	+
Output Vector	3154Beam EndB Pt4 Comb St 🗧 🛽	+
Additional Vector(s)		
Options		
Transform		
Data Conversion	Average	\sim
Data Selection	Contour Group	\sim
🕫 Туре	Elemental	\sim
Show On Groups	Full Model / Visible Groups	æ
Show As	Filled	\sim
🗉 Levels		
🗉 Legend		
Freebody		×
-		
Output Set		

Select an Output Set for Contour Deform and Freebody results ≝Model Info ■Meshing ■PostProcessing





5.11 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 G^2 /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- σ 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- σ 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 4 σ 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- σ 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.

 $R_n = \frac{n_1}{N_1} + \frac{n_2}{N_2} + \frac{n_3}{N_3}$





5.12 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	es infinite 11.0E6 cycles		14.0E4 cycles

1 _	0.6831·n	$0.271 \cdot n$	_ 0.0433·n
Τ-	0	11.0E5	14.0E4



Please share with your friends and visit us online



6. EXAMPLE 2: SOLID MESHED BEAM

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.

The beam properties are shown below:

w = 2 in
T = 0.25 in
$$L_{beam}$$
 = 5 in
W = 0.5 lbf
E = 10e6 psi







6.1 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}

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

Based upon this end 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 G_{rms} we see that G_{rms} is approximately 27 Gs:

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





6.2 **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.







Modal Frequency Table







6.3 **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.





6.4 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.



Applied CAx / Predictive Engineering White Paper Please share with your friends and visit us online



7. 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."

8. **RECOMMENDED READINGS**

- Linear Dynamics for Everyone, Three Part Series on Vibration (<u>www.PredictiveEngineering.com</u>)
- Principles of Vibration Analysis: Normal Modes to PSD to Direct Transient (<u>www.AppliedCAx.com</u>)
- Vibration Analysis for Electronic Equipment by Dave S. Steinberg
- NX Nastran Basic Dynamic Analysis User's Guide





9. APPENDIX

9.1 FLOW CHART FROM NX NASTRAN THEORETICAL MANUAL



Figure 1. Flow diagram for random analysis module.

12.2-4



Applied CAx / Predictive Engineering White Paper

Please share with your friends and visit us online





9.2 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.





Applied CAx / Predictive Engineering White Paper

Please share with your friends and visit us online



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.





Applied CAx / Predictive Engineering White Paper

Please share with your friends and visit us online



9.3 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 Spectra	al Density Factor	5			×
Correlation Table					
Master=>Master 1.(3) :	Int1=0	Excited Subca	se:	Master	
		Load Set:	1F	SD Excitation Node	
		Applied Subca	ise:	Master	
		Load Set:	1F	SD Excitation Node	
Edit Correlation Table					
Factor	PSD Fund	tion		PSD Interpolation	
Real 1. x	3PSD Function		\sim	0Log Log	~
Imaginary 1, X	0None		\sim	0Log Log	~
Apply					
Autocorrelation Function 1	Fime Lag				Ŧ.
Lag Intervals 10000 Starting Lag 1.E-6 Max Lag .16					
Prev <u>O</u> K 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 lagStarting Lag: Starting time stepMax 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.





Applied CAx / Predictive Engineering White Paper

Please share with your friends and visit us online



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 X	
	Nodal Output Requests	
	Displacement T1 T2 T3 R1 R2 R3 Velocity T1 T2 T3 R1 R2 R3	NASTRAN Power Spectral Density Factors X
NASTRAN Output for Random Analysis	Acceleration	Correlation Table
Nodal Output Requests None Power Spectral Density Functions Autocorrelation Functions Boot Mean Square Image: All state of the state of	Acceleration III II2 II3 RI R2 R3 Elemental Stresses Springs Axial Torsional Bars Axial Torsional Bars Axial Icoc 1 Icoc 2 Icoc 3 Icoc 4 End A Icoc 1 Icoc 2 Icoc 3 Icoc 4 Beams End A Icoc 1 Icoc 2 Icoc 3 Icoc 4 Beams End A Icoc 1 Icoc 2 Icoc 3 Icoc 4 Plates Bottom X Normal Y Normal XY Shear Top X Normal Y Normal Z Normal Solids X Normal Y Normal Z Normal XY Shear YZ Shear ZX Shear Axisym Radial Azim Axial Forces Summary Data Only PSDF AUTO	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 V 0Log Log Imaginary 1. X 0None Apply Autocorrelation Function Time Lag Lag Intervals 10000 Starting Lag 1.E-6 Max Lag .16
	Prev Next OK Cancel	





9.4 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.







2017

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 V
Loads	0None v
Temperatures	0From Load Set 🛛 🗸
Initial Conditions	0None v
Constraint Equations	0From Constraint Set 🛛 🗸
Bolt Preloads	0From Load Set 🗸
Other DOF Sets	
M <u>a</u> ster (ASET)	0None v
Kinematic (<u>S</u> UPORT)	0None ~
SUPORT1	0None ~
OMIT	0None ~
QSET	0None ~
CSET	0None ~
BSET	0None ~
Prev Ne <u>x</u> t	<u>Q</u> K Cancel

Analysis Set Manager (Active: 4Random Respons	
Analysis Set : 1Normal Modes Analysis Analysis Set : 2Random Response 10 modes (w ATOC) Analysis Set : 3Random Response 10 modes (no ATOC) Analysis Set : 4Random Response Example	<u>A</u> nalyze <u>A</u> nalyze Multiple
Solver : NX Nastran Type : Random Response Integrated Solver : NX Nastran	Expo <u>r</u> t Active
Options Master Requests and Conditions	Preview Input
Case : 12 Side Case : 2+Z Side	MultiSet
	<u>C</u> opy Delete
	Renumber
	Load
	New
	<u>E</u> dit
	Done





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 Factors		
Correlation Table		
1=>1 1.(3) :Int1=0 Excited Subcase: 1		
$\begin{array}{cccccccccccccccccccccccccccccccccccc$	2	Z Side Excitation
Applied Subcase: 2		
Load Set:	1+	-Z Side Excitation
Edit Correlation Table		
Factor PSD Function		PSD Interpolation
Real 1. x 42x PSD Function	\sim	0Log Log 🗸 🗸
Imaginary 1. X 54x PSD Function	\sim	0Log Log 🗸 🗸
Apply		
Autocorrelation Function Time Lag		
Lag Intervals 0 Starting Lag 0. Max Lag 0.		
Prev Ne <u>x</u> t	<u>о</u> к	Cancel





9.5 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.





