# **Random Vibration Analysis in FEMAP**

An Introduction to the Hows and Whys

Adrian Jensen, PE | Senior Staff Mechanical Engineer Kyle Hamilton | Staff Mechanical Engineer





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



# **Table of Contents**

1.	IN	ITRODUCTION	. 4
2.	Tł	HE PSD FUNCTION	. 5
3.	Tł	HE NX NASTRAN METHOD	. 7
4.	E۷	XAMPLE 1: CANTILEVER BEAM	. 8
	4.1	PROBLEM DEFINITION	8
	4.2	Analytical Solution	9
	4.3	Defining the System Damping	.10
	4.4	CREATING THE PSD FUNCTION	.11
	4.5	CREATING THE MODAL FREQUENCY TABLE/SETTING UP THE LOAD SET OPTIONS FOR DYNAMIC ANALYSIS	.12
	4.6	CREATING THE EXCITATION NODE AND TYING IT TO THE MODEL	.14
	4.7	LOADING THE MODEL	.15
	4.8	CONSTRAINING THE MODEL	.16
	4.9	Specifying Groups for Nodal and Elemental Output	.17
	4.10	CREATING AN ANALYSIS SET – SIMPLE PSD	.18
	4.11	INTERPRETING THE OUTPUT	.26
	4.12	RMS VALUES	.27
	4.13	Positive Crossings	.28
	4.14	Fatigue Analysis Using RMS Stress and Positive Crossings	.30
	4.15	Fatigue Analysis – Time to Failure	.31
5.	E۷	XAMPLE 2: SOLID MESHED BEAM	32
	5.1	Analytical Solution	.33
	5.2	PSD FUNCTION INPUT.	.34





		ONCLUSION	, 7
		COMPARING MILE'S APPROXIMATION AND PSD RESULTS	â
5	.3	PSD Stress Results	5





# 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.



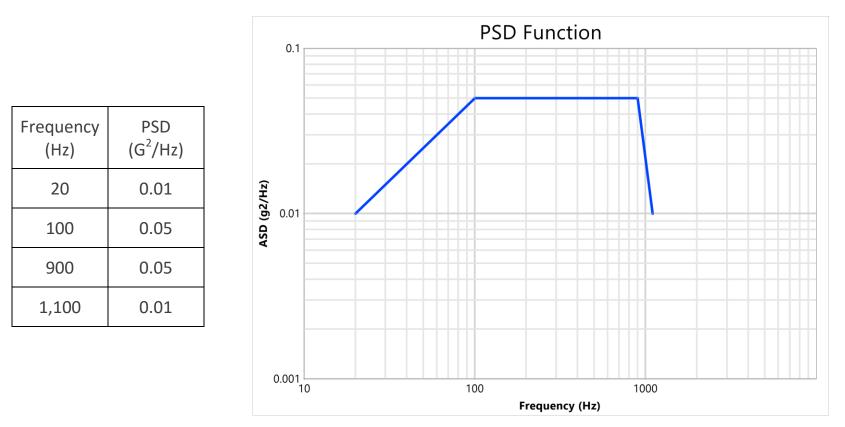




# 2. THE PSD FUNCTION

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 G<sup>2</sup>/Hz.

A typical power spectral density is shown below:

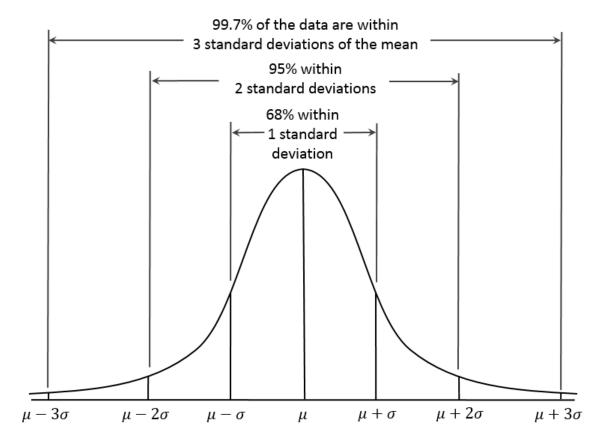


For more of the nitty gritty math details see NASA's webpage on random vibration here: <u>https://femci.gsfc.nasa.gov/random/</u>





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- $\sigma$  stresses, which represent the stress that the system will likely see 68% of the time. The 2- $\sigma$  stress level covers 95% of cases, and 3- $\sigma$  covers 99.7%. Most of the time a system is designed to the 3- $\sigma$  stress level.

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





# 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. EXAMPLE 1: CANTILEVER BEAM

# 4.1 **PROBLEM DEFINITION**

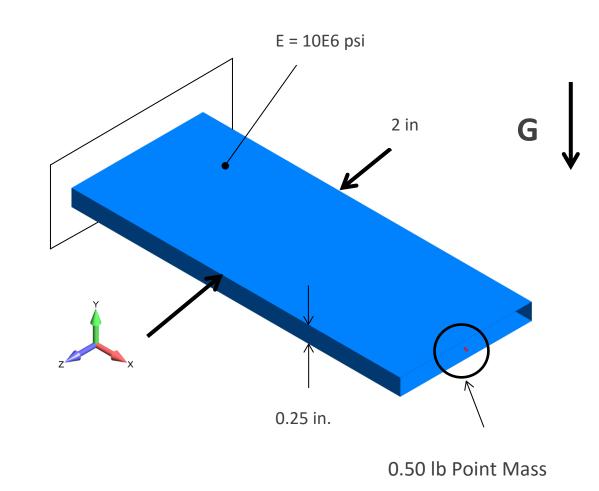
A cantilevered aluminum beam 5 inches in length is used to support a 0.50 lb mass. Our objective is to determine the dynamic stresses and fatigue life of the beam for vibration along the vertical axis.

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<sup> $\psi$ </sup>. The PSD input (PSD<sub>IN</sub>) 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 =  $386 \text{ in/s}^2$ .



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





#### 4.2 ANALYTICAL SOLUTION

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

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

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

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

For a beam, the transmissibility can be approximated as:

$$Q = 2\sqrt{f_n} = 21$$

Mile's equation can be used to approximate the G<sub>out</sub>(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  $G = 1g \text{ or } 386 \text{ in/s}^2$ 

In English units, the max PSD<sub>out</sub> = 13.14e6 in2/s4. 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.







#### 4.3 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.

E Function Defin	nition						-		×
<b>ID</b> 2	Title Damping				Type 7	Critical Da	mp vs. Freq		~
X - Frequency	Y - Frac Crit				X Axis Lo	og Scale	Y Axi	s Log Sca	le
0.	0.0238 0.0238	Frac Crit .0476 - .04165 - .0357 - .02975 - .0238 - .01785 - .01785 - .01795 - .00595 - 0. 0.1	0.2	0.3	0.4 0.5	0.6	0.7 0.8		1 quenc
Data Entry	O Edit Phase ( X )		Δ	dd	Copy Fr	unction	Cat D	ata Series	Data
◯ Linear <u>R</u> amp	Edit Magnitude (	Y) Delta X 1		date		n Library		from Clip	
C Equation	O Periodic	X Varia <u>b</u> le <b>x</b>				Library		to Clipb	
<b>Х</b> 1. То Х	<u>Ү</u> 0.0238 То Ү			lete set	S	ΣK		Cancel	

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.





# 4.4 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 4.7).

E Function Defin	nition									_		×
ID 3	Title PSD Function						Туре	3vs. Fre	quency			~
X - Frequency	Y - Factor						X Axis	Log Sca	le	Y Axis	Log Sca	ale
10. 2000.	0.2 0.2	Factor .4 - .32 - .28 - .24 - .2 - .16 - .12 - .08 - .04 - 0 .10.	209.	408.	607.	806.	ı 1005.	1204.	1403.	1602.	1801.	2000. Frequency
Data Entry										<b>C</b> + <b>D</b>		
● Single <u>V</u> alue ○ Linear <u>R</u> amp	Edit Phase (X) Edit Magnitude (	(Y)	Delta X	1	Add	_		Function			ta Series	
	○ Periodic	X Varia <u>b</u> le X		<u>U</u> pdate		Load from Library		ary	Paste from Clipboard			
Σ I X	Y				<u>D</u> elet	e	<u>S</u> ave	to Librar	y	Сору	to Clipbo	oard
To X	To Y				Rese			<u>O</u> K			Cancel	





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

The model 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.

To have Femap set up the table for you, you must first run an eigenvalue analysis. Once the eigenvalue analysis it 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

Analysis Set					
<u>T</u> itle <u>Eigenvalue</u>					
Analysis <u>P</u> rogram	36NX Nastran	~			
<u>A</u> nalysis Type	2Normal Modes/Eigenvalue				
	Run Analysis Usi	ng VisQ			
Ne <u>x</u> t	<u>O</u> K	Cancel			

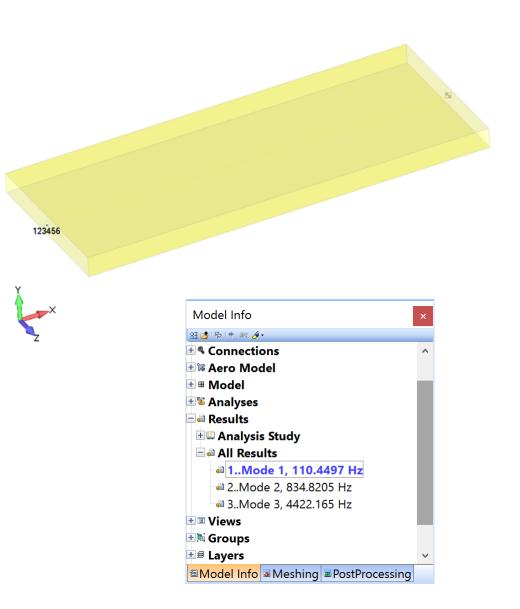
frequencies of interest (significant frequencies) are processed. This greatly limits the amount of postprocessing that is required for the Random Analysis. More will be said on this later on....





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 mass-less (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 4.10 we will show you how these Normal Modes are used to generate the Solution Frequencies for the Random Analysis.





# 4.6 CREATING THE EXCITATION NODE AND TYING IT TO THE MODEL

There are two ways to go about exciting the model. The traditional method is called the Large Mass Method. A more contemporary method has been developed called the Direct Method, wherein an acceleration is directly applied to a node. We will use the Direct Method.

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. In the case where the base of the structure is not one node, a rigid link approach can be used.



# 4.7 LOADING 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 results in inches (psi) so we will enter an acceleration of 386 in/s<sup>2</sup>.

Editing Load Definition			X
Load Set 1 PSD Excita	ation Node		
Title Acceleration on Node		Coord Sys	0Basic Rectangular v
Color 10 Palette	Layer 1	-	
Acceleration	Direction		Method
	Components		● Co <u>n</u> stant
	◯ Vector		◯ Variable
	O Along Curve		O Data Surface
	○ Normal to Plane		
	Normal to Surface	<u>S</u> pecify	<u>A</u> dvanced
	Load	Time (Free Descendence	Data Surface
	AX 🔽 0.	Time/Freq Dependence	f <sub>av</sub> 0None
	AY 🖂 386.		0None 🗸 👘
	AZ 🔽 0.		0None 🗡 🏥
	Phase 0.	0None ~	ĥy
			<u>Q</u> K Cancel



# 4.8 CONSTRAINING THE MODEL

Since we are using the *Direct Acceleration Method*, only one constraint set is required.

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 Ydirection (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 Layer 1								
	X Symmetry X AntiSym							
	Y Symmetry Y AntiSym							
RX RY RZ Free No Rotation	Z Symmetry Z AntiSym							

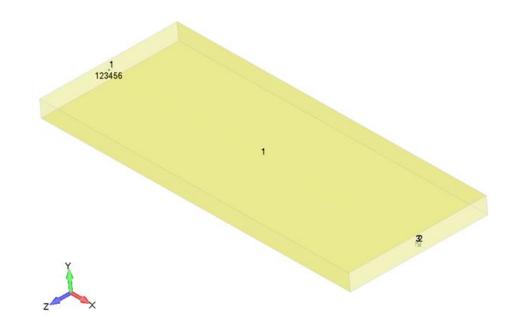




# 4.9 SPECIFYING GROUPS FOR NODAL AND ELEMENTAL OUTPUT

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.







#### 4.10 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 X							
Title PSD Analysis							
Analysis Program 36NX Nastran ~							
Analysis Type 6Random Response ~							
Solve Using							
<ul> <li>Integrated Solver</li> </ul>							
$\bigcirc$ Linked Solver							
Solver is undefine	ed. Go to File Prefer	ences Solvers.					
⊖⊻isQ							
Next <u>O</u> K Cancel							





Keep pressing **Next...** until you arrive at the 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	Dance of Internet	Method ID	1		
Real Solution Methods	Range of Interest				
Lanczos		Real	Imaginary		
○ Auto (HOU/MHOU)		0.	0.		
○ S <u>u</u> bspace	F <u>r</u> om (Hz)				
Legacy Real Solution Methods	<u>T</u> o (Hz)	2000.	0.		
◯ <u>G</u> ivens	Eigenvalues and Eig	envectors			
O Modified Givens	-				
◯ Inverse Power	Number <u>E</u> stimated	0			
○ Inverse Power/Sturm	Number Desired	0			
○ <u>H</u> ouseholder	Normalization Meth	Mass			
O Modified Householder	Macc		Default		
Complex Solution Methods	O Max	D 0			
OHessenberg	DOF	0			
O Complex Inverse Power	<u>P</u> oint				
O Complex Lanczos	Complex Solution O	ptions			
	Convergence		0.		
Solution Type	Region Width		0.		
	5	-	0		
● Mo <u>d</u> al	Overall Damping (	3)	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 Response bounds (# 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.

NASTRAN Dynamic Analysis			×		
Use Load Set Options		Frequency ID	2		
Options for Dynamic Analysis Solution	Frequencies				
Equivalent Viscous Damping	Limit Response Based	I on Modes			
Overall Structural Damping Coeff (G)	0.	Number of Modes	0		
Modal Damping 2Damping	~ f <sub>xy</sub>	Lowest Freq (Hz)	0.		
As Structural ( KDAMP )		Highest Freq (Hz)	2000.		
Equivalent Viscous Damping Conversion		Transient Time Step 1	Intervais		
Convert using Solution Freq ( WMC	DAL)	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 Tal	ble 🗸 🛵	Damping/Freq Corre	elation		
Modal Freq		0None	√ f <sub>y</sub>		
Prev Ne <u>x</u> t		<u>O</u> K	Cancel		
	5 14 115 1				
	e From Modal Result	ts ×			
Modal Results	Hz				
	Eirst Freq 1Mode 1, 110.4497				
Last Freq	Last Freq 1Mode 1, 110.4497 H				
Additional Soluti	on Frequency Points				
Number of Po	ints per Existing Mode	5			
Frequency Ba	nd <u>S</u> pread (+/-)	10. %			

<u>0</u>K

Cancel





In the XY Output 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 *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 Analysis $X$							
Output Requests	Output Requests						
Summary							
Modal Participation F	actors						
Modal Effective Mass	5						
Modal Effective Weig	iht						
Modal Effective Mass	s Eraction						
Reference Node 0							
Prev Next	<u>O</u> K	Cancel					
NASTRAN Output for Ra	ndom Analysis	×					
Nodal Output Requests							
None							
O Power Spectral Densit	y Functions						
O Autocorrelation Function	ons						
○ Root Mean Square							
Elemental Output Requests	S						
None							
O Power Spectral Density Functions							
O Autocorrelation Functions							
O Root Mean Square	○ Root Mean Square						
Prev Next	<u>0</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-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 4.9. If you select PSDF it will generate a function with the acceleration vs frequency for a group.

NASTRAN Out	put for Randor	n Analysis	×					
Nodal Output Re	Nodal Output Requests							
Displacement Velocity Acceleration		T3     R1       T3     R1       T3     R1       T3     R1       T3     R1	R2     R3       R2     R3       R2     R3					
Elemental Stress	ses							
<u>S</u> prings	Axial							
Rods	🗌 Axial	Torsional						
<u>B</u> ars	Axial							
1	End A Loc 1 End B Loc 1 End A Loc 1		_					
1	End B 🗌 Loc 1		bc 3 🗌 Loc 4					
<u>P</u> lates Bo	ottom 🗌 X Norr Top 🗌 X Norr	_						
Solids	🗌 X Norr	_						
A <u>x</u> isym	Radial	Azim Azim	xial Shear					
Forces								
Summary Da	ata Only	PSDF	AUTO					
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: 1	PSD Load		
	Applied Subcase:	Master		
	Load Set: 1	PSD 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	×
Primary Sets	
<u>C</u> onstraints	1Fixed ~
<u>L</u> oads	1PSD Excitation Node $\vee$
Temperatures	0From Load Set 🗸 🗸
Initial Conditions	0None v
Constraint <u>E</u> quations	0From Constraint Set V
Bolt Preloads	0From Load Set $\lor$
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 ~
<u>B</u> SET	0None v
Prev Ne <u>x</u> t	OK Cancel



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

NASTRAN Output Requests X						
Nodal			Elemental			
✓ Displacement	0Full Model	~	<b>Eorce</b>	0Full I	Model ~	·
Applied Load	0Full Model	$\sim$	✓ Stress	0Full I	Model ~	·
Constraint Force	0Full Model	$\sim$	Strain	0Full I	Model ~	•
Equation Force	0Full Model	$\sim$	Strain Energy	0Full I	Model ~	-
Force <u>B</u> alance	0Full Model	$\sim$	Heat Flux	0Full I	Model ~	•
<u>Velocity</u>	0Full Model	~	Enthalpy	0Full I	Model ~	•
Acceleration	0Full Model	$\sim$	Enthalpy Rate	0Full I	Model ~	•
Kinetic Energy	0Full Model	~	Temperature	0Full I	Model ~	•
Iemperature	0Full Model	~	Kinetic Energy	0Full I	Model ~	·
			Energy Loss	0Full I	Model ~	
			Fluid Pressure	0Full I	Model ~	·
Customization					Prev	
Element Corner Results Res			sults Destination		Flev	
Output Modes ( a,b,c	THRU d )		2PostProcess Only	$\sim$		
		Ech	no <u>M</u> odel		<u>O</u> K	
Magnitude/Phase     O Real/Imaginary					Cancel	
Relative Enforced Motion Results						



# 4.11 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 a duration of excitation.

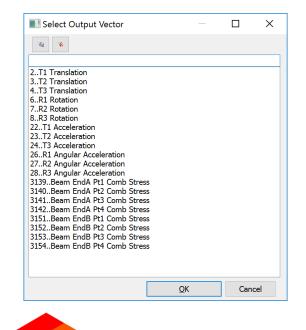
# Model Info

왜 🦽 😼 👘 🗃 🔏 -**± Coordinate Systems** Geometry Connections **±** Sero Model + 
■ Model Analyses Results **Hundred Analysis Study All Results** a 1...Mode 1, 110.4497 Hz 2...Mode 2, 834.8205 Hz 3...Mode 3, 4422.165 Hz 4...RMS Values 5..Positive Crossings + Views **Groups** ± 
 E Layers Selection List

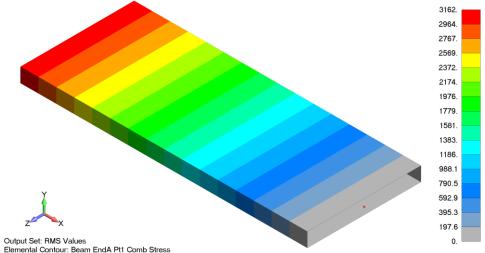


# 4.12 RMS VALUES

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.









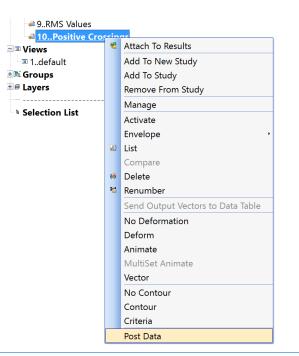
#### 4.13 **POSITIVE CROSSINGS**

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.

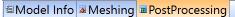


Select PostP	Processing Dat	a		_	×
View 1	default			E	☑ Dynamic Max/Min
O <u>u</u> tput Sets	10Positive Cro	ossings	✓ 😼 « Ar	Program : NX Nastran nalysis Type : Random Set Value : 0.	
Output Vecto	ors				
Defor <u>m</u>	3139Beam Er	ndA Pt1 Comb Stress	<ul><li>✓ A ≪</li></ul>	Transform	Vector Info
	Max : E	lement 1 110.4	1009 Min :	Element 1	110.4009
Contour			× ¥ *	Transform	Vector Info
	Max :		Min :		
	Double-Side	d Planar Contours		Multiple Cont	our Vectors
Contour	r O <u>p</u> tions	Contour Arrows	Trace Locations	Expand Complex	<u>0</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			x
💩 • 📩 🚯 🖏 🛼 • 🖁 • 🗆 • <mark>گ</mark> 🔌			
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	$\sim$	æ
Show As	Filled		$\sim$
🗉 Levels			
🗉 Legend			
Freebody			×
Output Set			
	Contour Deferm and Freehady		4-





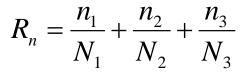


# 4.14 FATIGUE ANALYSIS USING RMS STRESS AND POSITIVE CROSSINGS

We can see that **Beam EndA Pt1 Comb Stress** vector gives a positive crossings frequency of 110.4 Hz. This means that given the white noise PSD input of  $0.2 \text{ G}^2/\text{Hz}$ , 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 $\sigma$  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 4 $\sigma$  level, but this will only happen 0.27% of the time, so we will ignore them.

All three  $\sigma$  level stresses fall into the infinite life 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.





**Predictive Engineering** 

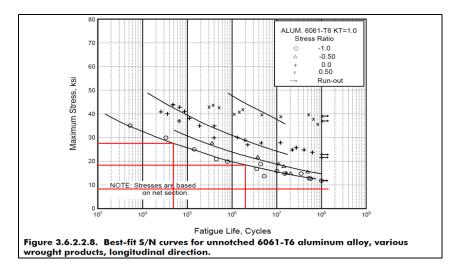


#### 4.15 FATIGUE ANALYSIS – TIME TO FAILURE

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 under 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 _	0.6831·n	0.271·n	0.0433·n
1 -	$\infty$	11.0E5	14.0E4



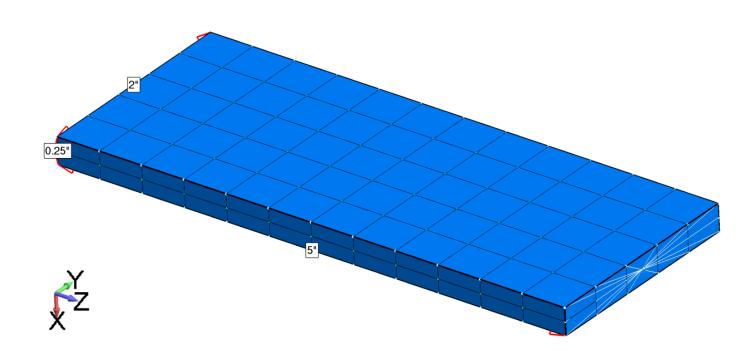


# 5. EXAMPLE 2: SOLID MESHED BEAM

Let's take a look at the same beam geometry modeled with The beam properties are shown below:

solid elements. The beam is massless, with a point mass of 0.5lbf (1.30e-3 snails) attached via RBE2 on the end.

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







# 5.1 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  $Y_{max}$ 

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

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

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

We can then approximate the transmissibility:

$$Q = 2\sqrt{f_n} \approx 21$$

Utilizing Miles Equation to estimate G<sub>rms</sub> we see that G<sub>rms</sub> is approximately 27 Gs:

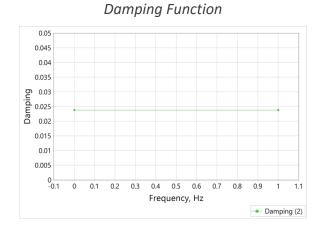
$$G_{outRMS} = \sqrt{\frac{\pi}{2} PSD_{in} f_n Q} = 27G's$$



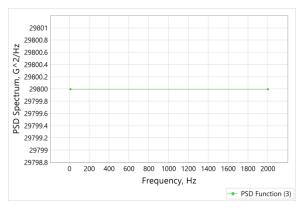


# 5.2 **PSD FUNCTION INPUT**

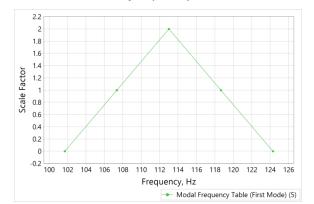
Then we generate the functions necessary for the PSD Analysis. Note the modal frequency table is centered at the first natural frequency with 10% spread in both directions.







Modal frequency table

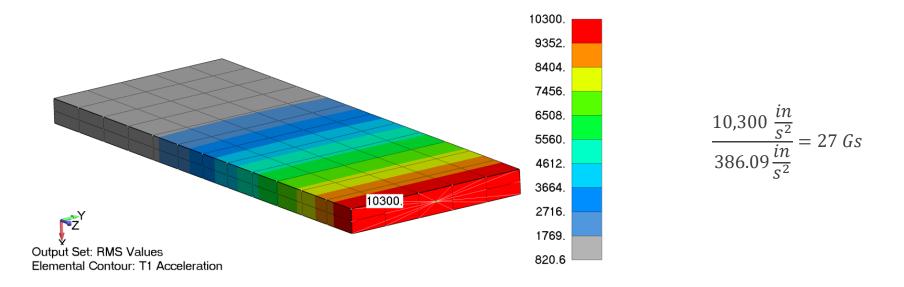






# **5.3 PSD STRESS RESULTS**

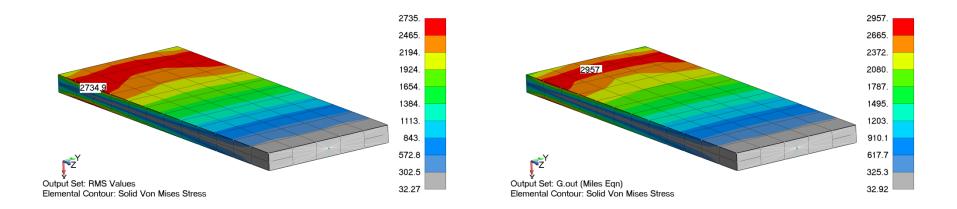
After running the analysis, let's take a look at the results. The PSD results can be validated by checking the resultant acceleration against the Mile's equation prediction. Mile's equation predicted 27 G's for the maximum acceleration. The results show 10,300 in/s<sup>2</sup> acceleration which matches up with the Mile's equation prediction.





# 5.4 COMPARING MILE'S APPROXIMATION AND PSD RESULTS

An additional verification is done by comparing the PSD stress results to the static analysis with the acceleration given by Mile's equation. The below results 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 =  $\frac{Mc}{I} = \frac{(F_{d}L)(\frac{T}{2})}{I_{xx}} = 3,240 \ psi$ 

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





# 6. 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."

