

WHAT WE ARE GOING TO DISCUSS

Most engineers are pretty familiar with the general concepts of vibration analysis but maybe just need a few pointers on the fundamentals and perhaps some suggested readings. This white paper provides the simulation engineer with all the necessary background to perform the following analysis sequences with Femap and NX Nastran.

- Normal Modes Analysis
 - ° Natural Frequencies and Mode Shapes (why they are only shapes and not magnitudes)
 - ° Utility and Significance of Mass Participation
 - ° Mode Strain Energy
- Modal Frequency Analysis
 - ° Theory of Modal Frequency (all loads are sinusoidal)
 - [°] Shaper Table Sine Sweep Analysis
 - [°] Expanding Complex Results
- PSD Analysis
 - ° Modal Frequency Analysis using Statistics
 - ° One Sigma versus Three Sigma von Mises Results
 - ° Fatigue Analysis with PSD
- Direct Transient Analysis (it is easier than you think)



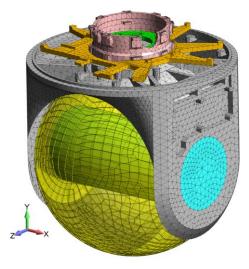
1. INTRODUCTION TO LINEAR DYNAMICS (NX NASTRAN)

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



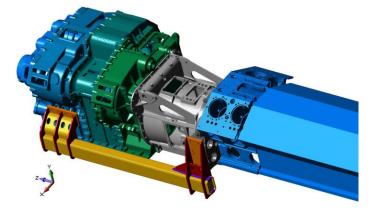
Linear FEA Model (courtesy Predictive Engineering)



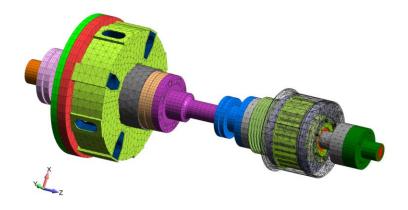


1.1 SOME EXAMPLES OF LINEAR VIBRATION ANALYSIS (PREDICTIVE)

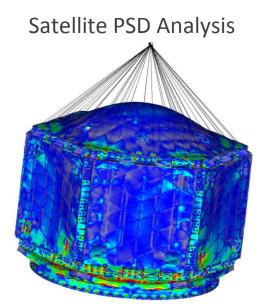
3,000 HP Transmission

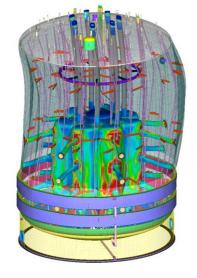


Drive Train Coupling



Seismic Analysis of Large Vessels







Applied CAx / Predictive Engineering White Paper – Please share with your Friends

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: $[k - \omega^2 m] \{u_0\} = 0$
For non-trivial solutions (i.e., $u_0 \neq 0$):
 $[k - \omega^2 m] = 0$ Giving us the well-known frequency relationship: $\omega = \sqrt{\frac{k}{m}}$



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)
- o 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:

$$Ku = F$$



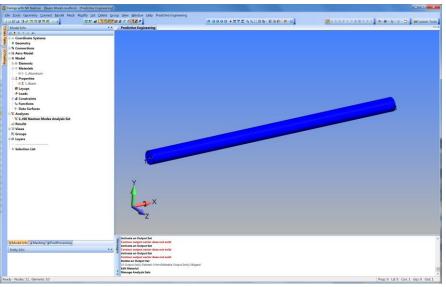
2. FOUNDATION OF FREQUENCY ANALYSIS

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.

Elastic Properties and Mass Density (Snails)

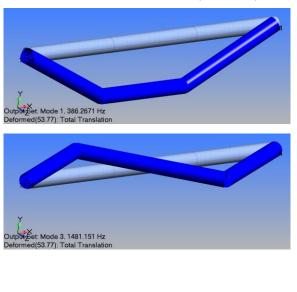
ionoral Function Deferrer	one Manlinear Dhy/Dand Feilum	Croop Electrical/O	Layer 1 Type	10
Stiffness	ces Nonlinear Ply/Bond Failure	Limit Stress	pucal Phase	Data Table
Youngs Modulus, <u>E</u>	10300000.	-	0.	E Charle
		Tension		<u>د</u>
Shear Modulus, <u>G</u>	0.	Compression	0.	
Poisson's Ratio, n <u>u</u>	0.33	Shear	0.	
Thermal				
Expansion Coeff, <u>a</u>	0.			
Conductivity, <u>k</u>	0.	Mass De <u>n</u> sity Da <u>m</u> ping, 2C/Co	2.64243E-4	
Specific <u>H</u> eat, Cp	0.		0.	
		Reference Temp	0.	
Heat Generation Factor	0.			
				Re
				<u></u>





2.1.1 HERE'S YOUR MODAL ANALYSIS CHECKLIST

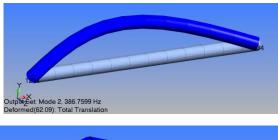
- o Elastic and mass properties are in consistent units
 - [°] The weight of your structure can be checked by summing the mass of the model and multiplying it by gravity (for US units of lbf, inch and seconds, it would be 386 in/s²)
- FEA model with a sufficient mesh density to capture the frequencies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

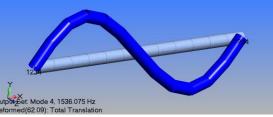


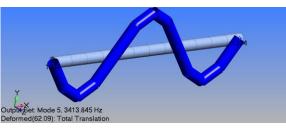
Doesn't Exist

Three Element Mesh (Coarse)

Twelve Element Mesh (Fine)











2.1.2 SETTING UP THE MODEL FOR NORMAL MODES WITH MASS PARTICIPATION

<u>T</u> itle NX Nastran	Modes Analysis Set	
Analysis <u>P</u> rogram	36NX Nastran	-
<u>A</u> nalysis Type	2Normal Modes/E	igenvalue 🔻
	Run Analysis Usi	ng VisQ
Next	<u>O</u> K	Cancel
nalysis Set Manager (Active	e: 1NX Nastran Modes /	Analysi
Analysis Set : 1NX Nastran I Solver : NX Nastran	Modes Analysis Set	Analyze
- Type : Normal Modes/Eige - Analyze : Local	nvalue	Analyze Multiple
		Export
Options Executive/Solution Bulk Data GEOMCHECK		Active
Options Executive/Solution Bulk Data GEOMCHECK MODELCHECK MODELCHECK Modal/Buckling		
Options Executive/Solution Bulk Data GEOMCHECK MODELCHECK MODELCHECK Modal/Buckling DAM Rotor Dynamics		Active
Options Executive/Solution Bulk Data GEOMCHECK MODELCHECK Modal/Buckling DDAM Rotor Dynamics Modal XYPIot Not Defined		Active Preview Input MultiSet Copy
Options Detions Detions Bulk Data GEOMCHECK MODELCHECK MOdal/Buckling DDAM Rotor Dynamics Modal XYPIn -Not Defined Response Spectrum App External Superelement F	Reference	Active Preview Input MultiSet
Options Detions Detions Bulk Data GEOMCHECK MODELCHECK MOdal/Buckling DDAM Rotor Dynamics Modal XYPIn -Not Defined Response Spectrum App External Superelement F	Reference	Active Preview Input MultiSet Copy
Options Executive/Solution Eulk Data GEOMCHECK MODELCHECK Modal/Buckling DDAM Rotor Dynamics Not Defined Response Spectrum App External Superelement f Master Requests and Cond	Reference	Active Preview Input MultiSet Copy
Options Executive/Solution Eulk Data GEOMCHECK MODELCHECK MODALCHECK Modal/Buckling DDAM Rotor Dynamics Modal/YPlot Not Defined Response Spectrum App External Superelement F Master Requests and Cond	Reference	Active Preview Input MultiSet Copy Delete

			- X		
ASTRAN Modal Analysis					
Real Solution Methods	Range of Interest	Method <u>I</u> D	1		
Lanczos <u>Auto (HOU/MHOU)</u>		Real	Imaginary		
© S <u>u</u> bspace	From (Hz)	0.	0.		
Legacy Real Solution Methods	To (Hz)	0.	0.		
Modified Givens	Eigenvalues and Eig	jenvectors	0		
 Inverse Power Inverse Power/Sturm 	Number Desired				
<u>H</u> ouseholder Modified Householder	Normalization Meth	od	Mass		
Complex Solution Methods Hessenberg Complex Inverse Power	 Mass Node I Max Point 	0	 Default Lumped Coupled 		
Complex Lanczos	Complex Solution Options				
Solution Type	Convergence Region Width		0.		
◯ Mo <u>d</u> al	Overall Damping (0.		
Prev Ne <u>x</u> t		<u>O</u> K	Cancel		
NASTRAN XY Output	t for Modal Ana				

<u>O</u>K

Cancel

Reference Node 0

✓ Modal Effective Mass Fraction

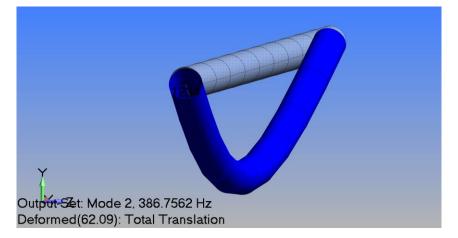


2.1.3 INTERPRETING RESULTS BASED ON ORTHOGONALITY AND MASS PARTICIPATION

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.

Frist Frequency 386.8 Hz

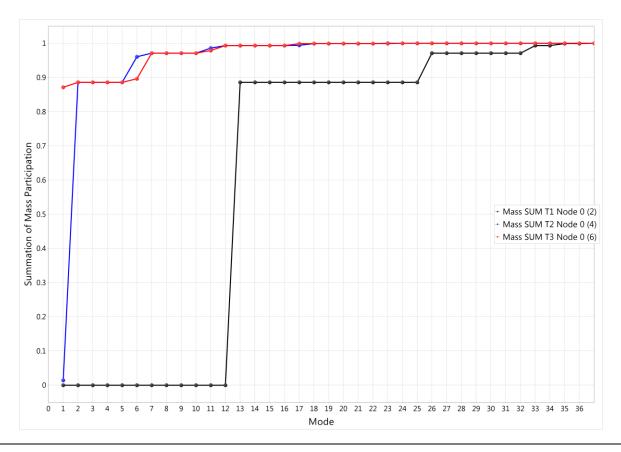
Second Frequency 386.8 Hz





FINITE ELEMENT ANALYSIS Predictive Engineering

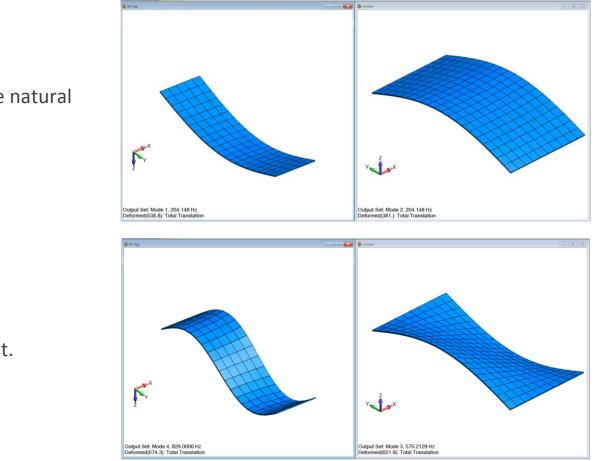
Mass participation tells you which modes have the "umph" and how many frequencies you need (modes) to accurately capture the dynamic response of the structure. On this later subject, a modal frequency analysis (e.g., PSD) formulates its response based on the number of modes chosen for the analysis. To 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.



At the first mode, we have the same natural frequency.

At higher frequencies, things get lost.



2.1.5 SIGNIFICANCE OF STRAIN ENERGY FOR FREQUENCY ANALYSIS

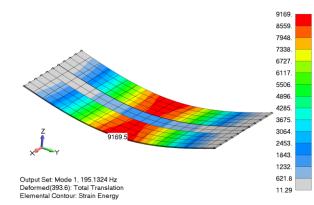
If one wants to move your natural frequencies up or down, sometimes intuition is good enough but it never hurts to have a quantitative tool. When a structure flexes or vibrates, there will be regions within the structure that are deforming more and other regions less. Since a natural frequency analysis 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 sound 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.

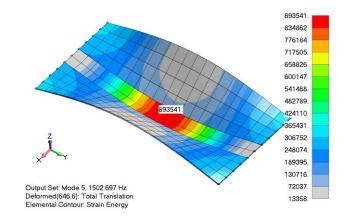
The Default Setup

(i) Magnitude/Phase	Real/Imagina	ry.		Can	cel
Output Modes (a,b,c THRU d)		Ec	Echo Model		<
			2PostProcess Only	•	
Z Element Corner Re	sults	Re	esults Destination	Prev	lus.
Customization					
			Fluid Pressure	0Full Model	
			Energy Loss	0Full Model	*
Iemperature	0Full Model	*	Kinetic Energy	0Full Model	*
Kinetic Energy	0Full Model		Temperature	0Full Model	
Acceleration	0Full Model		Enthalpy Rate	0Full Model	
Velocity	0Full Model	Ψ.	Epthalpy	0Full Model	*
Force Balance	0Full Model		Heat Flug	0Full Model	
Equation Force	0Full Model	*	Strain Energy	0Full Model	•
Constraint Force	0Full Model	•	Strain Strain	0Full Model	٣
Applied Load	0Full Model		Stress 2	0Full Model	
✓ Displacement	0Full Model	•	Eorce	0Full Model	
Nodal			Elemental		

Follow the Red – Increase the thickness of the outer strips



To Increase the Fifth Mode – Increase the thickness of the middle section





3. STANDARD NORMAL MODES ANALYIS

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 with NX Nastran - (PCB Analysis from Modal to MF to PSD to DT.mo		Define Material - ISOTROPIC	×
Elle Tools Geometry Connect Model Mesh Modely List Delete Gro		Define Material - 130 NOFIC	
00000000000000000000000000000000000000			
	PCB Analysis from Mo AFR-0814-01 Vibration		
通過 別 ため 通		ID 1 Itle FR4 PCB Color 55 Palette	Layer 1 Type
# A Coordinate Systems			
Geometry Gonnections			
# & Connections		General Function References Nonlinear Ply/Bond Failure Creep Electrical/Optica	al Phase
H Model			
I Elements	ALDELT	Stiffness	
🖂 4 Materials			
一回ル 1.FR4 PCB		Youngs Modulus, E 21000. Tension 0.	
-iii 4 2.Big Dog			
WA 3.Little Dog		Shear Modulus, G 0. Compression 0.	
I Properties		Shear Modulus, <u>G</u> 0. Compression 0.	
-#11.5404 PCB -#12.55g Dog			
I 3.Little Dog		Poisson's Ratio, nu 0.15 Shear 0.	
# Layups		. Shear	
e & Loads			
🗐 🕭 1Y-Axis PSD Driver Load		Thermal	
⊕ *= Load Definitions			
3+ Body Loads		Expansion Coeff, a 0.	
ef Other Loads e 4 2.Y-Axis Transient Shock Load		Mass Density 3.	.13559E-9
Constraints	Y 123456	Conductivity, k 0.	
E 2 1. Universal All-Purpose Constraint Set	123456		
W Functions			•
STATES AND A STATE		Specific Heat, Cp 0.	
4y 2.PSD Driver Function over Frequency		Reference Temp	
4 3.PSD Modal Frequency Critical Damping		Heat Generation Factor 0.	
4y 4. Transient Shock (Pothole) from Table 30, GMW3172 2012			
t" Data Surfaces			
S Anatyses 1.XXX-0814-01 Normal Modes Analysis	Z		
S 2.XXX-0814-01 Y-Axis PSD Analysis			
\$ 3XXX-0814-01 Y-Axis Transient Shock (Pothole) Analysis			
al Results			
III III Views			
計測; Groups			
# # Layers	Todate 64 Pents).		
A Selection List	Full OpenGL Hardware Appeleration (Double Buffered)		
	Vicinity Supports OpenIX, 4.3 © Performance Constript is enabled		
	2 Edit Load Definition		
	Edit Laad Definition		
ti Model Info a Meshing a PostProcessing	1 Node(s) Selected.		
	1 Loadiji Creanal.		
Entity Info **	2 Analysis Set(s) Deleted, 0 NonDeletable Analysis Set(s) Skipped.		
Node 1 Ccord(0) = -9.219982, 21.94487, -8.128E-5	Renumber Analysis Sets Rebuilt Stockel		
DefCS = 0 OutCS = 0	Beginning Update of Database		
Construction of August 1991	Distance Update Completed. No Every.		OK Caraci
	Shrip Datase	fxy Load Save Copy	OK Cancel
ady - Nodes: 7442, Bements: 6249	Prop. 0 Ld.2 Conc.1 Gept 1 Out 115		

Femap v11.1.2A Model

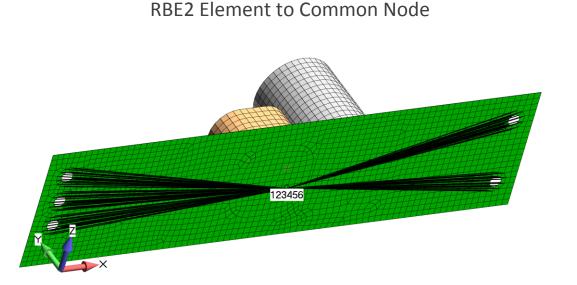
Units: N, mm, Tonne, s





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".



RBE2 with 3-DOF Dependent Nodes

Define RIGID Element - Enter Nodes or Select with C	Cursor
ID 10427 <u>Color</u> 24576 <u>Palette</u> Layer 1 RBE1 RBE2 RBE3 (Interpolation)	1 Property Type
Notes 483 DOF Nodes 483 484 485 487 487 486 487 487 487 489 7 TY RY 3-487 489 491 492 492	Independent Node 1 New Node At Center
Thermal Expansion Coefficient 0. Material	Single RBE2 OK Cancel





3.2 NATURAL FREQUENCY RESULTS AND INTERPRETATION

A	nalysis S	Set	X
	Title	XXX-0814-01	Normal Modes Analysis
	Analysis	Program	36NX Nastran 👻
	Analysis	Туре	2Normal Modes/Eigenvalue 🔹
			Run Analysis Using VisQ
	Nex	ĸt	OK Cancel

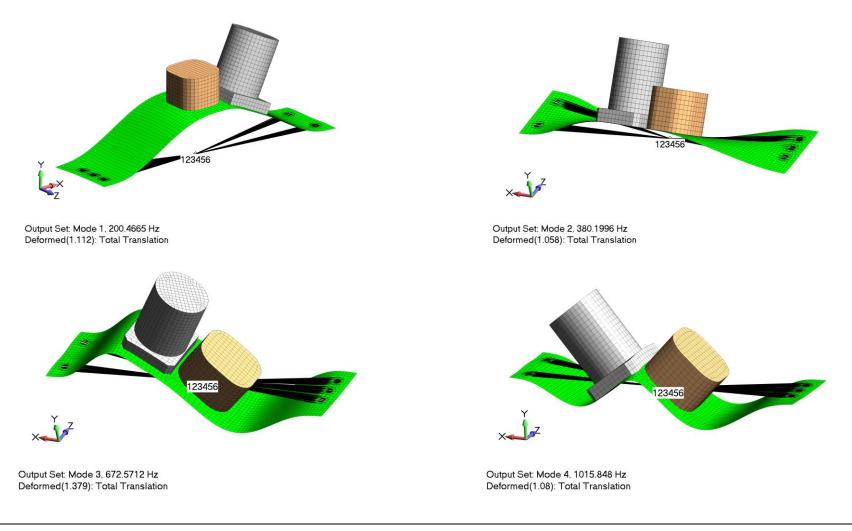
A normal modes / Eigenvalue analysis is the starting point for all linear dynamics work. It is simple to setup but difficult to interpret the results.

Boundary Conditions	×	NASTRAN Modal Analysis	Conditions .	X	Nastran Output Req	uests			-	
Primary Sets Constraints	1Universal All-Purpose Con: -	Skip EIGx Real Solution Methods	Range of Interest	Method ID 1	Nodal			emental		
Loads Temperatures	0None	Lanczos Auto (HOU/MHOU) Subspace	From (Hz) 0.	Real Imaginary	Displacement	0Full Model		Force Stress	0Full Model	
Initial Conditions	0None	Legacy Real Solution Methods	To (Hz) 0. Eigenvalues and Eigenve	0.	Constraint Force	0Full Model		Strain Strain Energy	0Full Model 0Full Model	_
Constraint Equations	0From Constraint Set 🔹	Modified Givens	Number Estimated	0	Force Balance	0Full Model	•	Heat Flux	0Full Model	
Bolt Preloads	0From Load Set	 Inverse Power/Sturm Householder 	Number Desired	10	Velocity	0Full Model		Enthalpy	0Full Model	
Other DOF Sets Master (ASET)	0None	Modified Householder	Normalization Method	Mass Default	Acceleration	0Full Model		Enthalpy Rate	0Full Model	
Kinematic (SUPORT)	0None 👻	Complex Solution Methods Hessenberg Complex Inverse Power	Max DOF O	C Lumped	Kinetic Energy	0Full Model		Temperature Kinetic Energy	0Full Model	
SUPORT1	0None	Complex Inverse Power	Complex Solution Option	0.				Energy Loss	0Full Model	
OMIT	0None	Solution Type	Convergence Region Width	0.	Customization			Fluid Pressure	0Full Model	
CSET	0None ▼	Modal	Overall Damping (G)	0.	Element Corner F		Results	Destination	Prev	/
BSET	0None 👻	Prev Next	ОК	Cancel	Output Modes (a,b,4	: THRU d)	2Po	ostProcess Only odel	▼ 0	к
Prev Next	OK Cancel				Magnitude/Phase	🔘 Real/Imaginar			Can	
					Relative Enforced	Motion Results				



FINITE ELEMENT ANALYSIS Predictive Engineering

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.



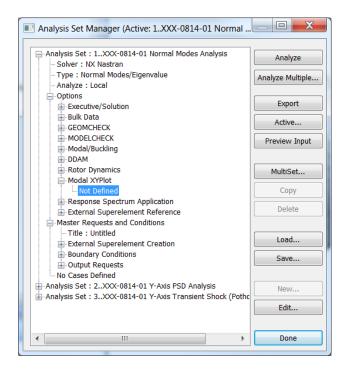


Applied CAx / Predictive Engineering White Paper – Please share with your Friends

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).

Analysis Set Manager / Normal Modes

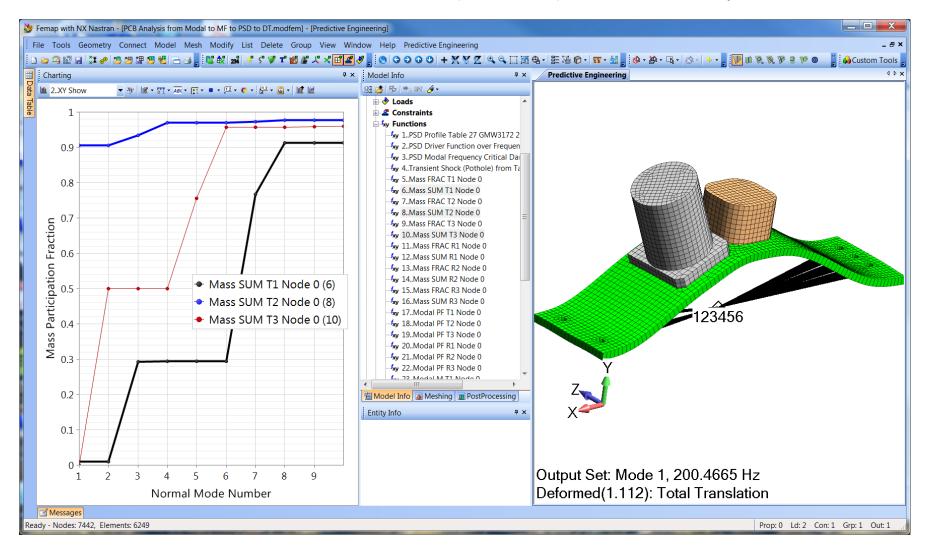


With the Not Defined item highlighted, hit the Edit button and the NASTRAN XY Output for Modal Analysis dialog box will appear. This box requests the mass particpation factors.

NASTRAN XY Output f	or Modal Analysis
Output Requests	
Summary	
Modal Participatio	n Factors
Modal Effective M	ass
Modal Effective W	/eight
Modal Effective M	ass Fraction
Reference Node	0
	OK Cancel



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.



4.1 RUNNING A MODAL FREQUENCY ANALYSIS IN FEMAP AND NX NASTRAN

Analysis S	et		X
<u>T</u> itle	XXX-0814-01	Modal Frequency A	Analysis
Analysis	<u>P</u> rogram	36NX Nastran	-
<u>A</u> nalysis	Туре	4Frequency/Harr	monic Respons 🔻
		Run Analysis Us	ing VisQ
Nex	t	<u>O</u> K	Cancel

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 Ydirection. 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.

Function Definition	
ID 3 Title PSD Modal Frequency Critical Damping	Type 7Critical Damp vs. Freq
X - Frequency Y - Frac Crit	X Axis Log Scale
10. 0.02 Frac Crit 20000. 0.02 04 .03 - 0.3 .025 - 02 .015 - 015 .015 - - .005 - - .03341.667 - -	7 6673.333 10005 13336.67 16668.33 20000. Frequency
Data Entry Gingle Yalue Edit Phase (X) Linear gamp Edit Maggitude (Y) Delta X 1 Eguation Periodic X Variable X X Yariable X Yariable X <lix< li=""> X<td>Add Copy Function Get Data Series Data Update Load from Library Paste from Clipboard Save to Library Copy to Clipboard</td></lix<>	Add Copy Function Get Data Series Data Update Load from Library Paste from Clipboard Save to Library Copy to Clipboard
X 10. Y 0.02 To X To Y	Reset QK Cancel

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 this document:

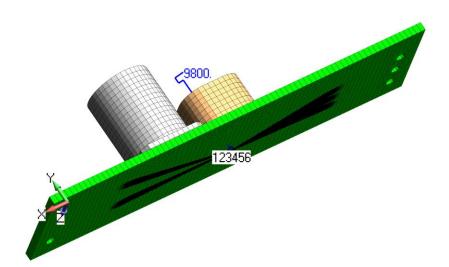
NX Nastran Dynamic Analysis.pdf





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.

Editing Load Definition			×
Load Set 1 Motor Fre	quency		
Title Acceleration on Node in	n mm/sec^2	Coord S <u>y</u> s	0Basic Rectangular 🔹
Color 10 Palette	Layer 1		
Acceleration	Direction Components Vector Along Curve Normal to Plane Normal to Surface	Specify	Method Constant Variable Data Surface Advanced
	Load <u>V</u> alue	Time/Freq Dependence	Data Surface
	AX 🔲 0.	0None 🔻 🖡	xy 0None 💌 🧮
	AY 🔽 9800		0None 💌 🧮
	AZ 🔲 0.		0None 💌 🧮
	Phase 0.	0None 🗸	W
		(OK Cancel





FINITE ELEMENT ANALYSIS Predictive Engineering

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).

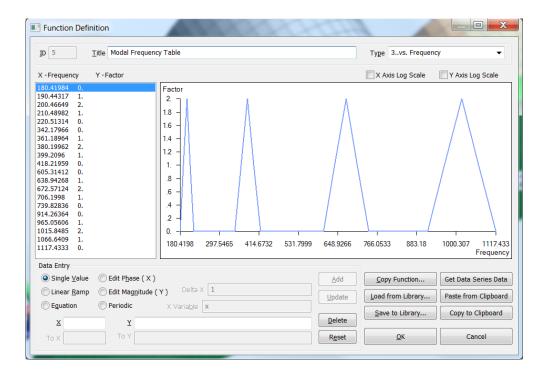
NASTRAN Modal Analysis			X
Skip EIGx		Method <u>I</u> D	1
Real Solution Methods	Range of Interest		
Lanczos		Real	Imaginary
<u>Auto (HOU/MHOU)</u>		-	
Subspace	F <u>r</u> om (Hz)	0.	0.
Legacy Real Solution Methods	<u>T</u> o (Hz)	0.	0.
O Givens	Eigenvalues and Eig	genvectors	
O Modified Givens	Number Catingated	-	0
O Inverse Power	Number Estimated	0	
O Inverse Power/Sturm	Num <u>b</u> er Desired		10
O <u>H</u> ouseholder	Normalization Meth	od	Mass
O Modified Householder	Macc		Default
Complex Solution Methods	Ma <u>s</u> Node II	0	C Lumped
Hessenberg	O Point	0	Coupled
O Complex Inverse Power	<u> </u>		Coupied
Complex Lanczos	Complex Solution C	ptions	
Solution Type	Convergence		0.
O Dire <u>c</u> t	Region Width		0.
Modal	Overall Damping (O	G)	0.
Prev Next		<u>o</u> k	Cancel

Dynamic Control Options		100 M	X		
Use Load Set Options					
Options for Dynamic Analysis Advanced	Options				
Equivalent Viscous Damping					
Overall Structural Damping Coeff (G)	Number of Modes	0			
Modal Damping 3PSD Modal F	Frequer 🔻 🖍	Lowest Freq (Hz)	0.		
As Structural (KDAMP)	_	Highest Freq (Hz)	0.		
Equivalent Viscous Damping Conversion	Transient Time Step Intervals				
Convert using Solution Freq (WMC	Number of	0			
Rigid Body Zero Modes(FZERO)	Time per	0.			
Freq for System Damping (W3 - Hz)	Output Interval	0			
Freq for Element Damping (W4 -	0.				
Frequency Response	Response/Shock Spe	ectrum			
Frequencies	Damping/Freq Corr	elation			
Modal Freq		0None	▼ f _{xy}		
Prev Next		<u>O</u> K	Cancel		



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.

Frequency Table From Modal Results						
Modal Results						
<u>F</u> irst Freq 1Mode 1, 200.4665 Hz ▼						
Last Freq 4Mode 4, 1015.848 Hz ▼						
Additional Solution Frequency Points						
Number of Points per Existing Mode 5						
Frequency Band <u>Spread (+/-)</u> 10. %						
OK Cancel						





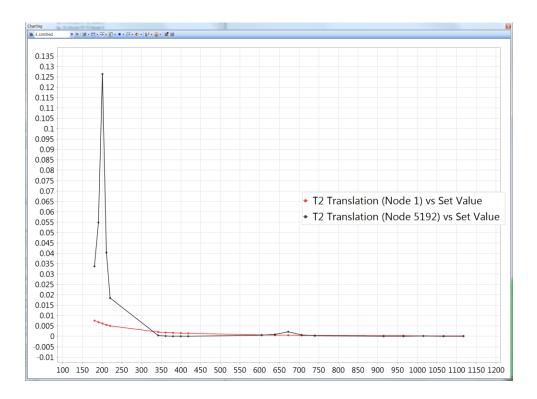
Then, one applies the boundary conditions and then lastly, one sets the output requests.

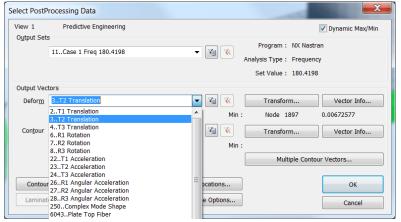
Boundary Conditions	X
Primary Sets	
<u>C</u> onstraints	1Universal All-Purpose Con: 🔻
<u>L</u> oads	1Motor Frequency
Temperatures	0From Load Set 👻
Initial Conditions	0None
Constraint <u>E</u> quations	0From Constraint Set 👻
Bolt Preloads	0From Load Set 🔹
Other DOF Sets	
M <u>a</u> ster (ASET)	0None 🔻
Kinematic (<u>S</u> UPORT)	0None 👻
SUPORT1	0None 👻
0 <u>м</u> іт	0None 👻
QSET	0None 👻
CSE <u>T</u>	0None 👻
<u>B</u> SET	0None 🔻
Prev Next	<u>O</u> K Cancel

Nodal			Elemental	
Displacement	0Full Model	•	Eorce	0Full Model
Applied Load	0Full Model	Ŧ	Stress	0Full Model 🗸
Constraint Force	0Full Model	Ŧ	St <u>r</u> ain	0Full Model
Equation Force	0Full Model	•	Strain Energy	0Full Model
Force <u>B</u> alance	0Full Model	Ŧ	Heat Flux	0Full Model
<u>V</u> elocity	0Full Model	•	E <u>n</u> thalpy	0Full Model
Acceleration	0Full Model	•	Enthalpy Rate	0Full Model
Kinetic Energy	0Full Model	Ŧ	Temperature	0Full Model
<u>T</u> emperature	0Full Model	Ŧ	Kinetic Energy	0Full Model
			Energy Loss	0Full Model
			Fluid Pressure	0Full Model
Customization				
🔽 E <u>l</u> ement Corner Re	sults	Re	sults Destination	Prev
Output Modes (a,b,c	THRU d)	_	2PostProcess Only	•
		Ec	ho <u>M</u> odel	ОК
Magnitude/Phase	Real/Imaginar	¥		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 (u_o) and phase angles (Θ). Hence, to get the time varying nature, you need to expand the complex results.







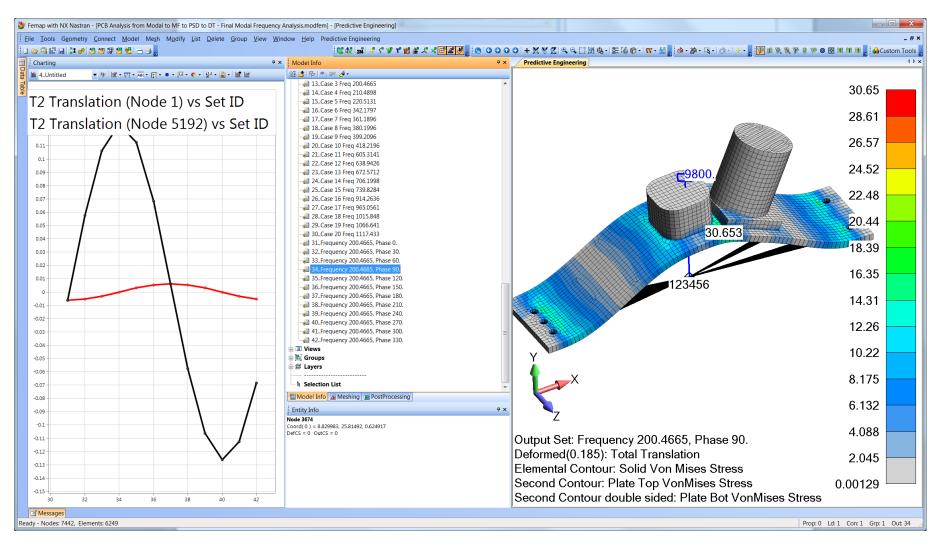
To obtain the time varying response from a Modal Frequency, one goes to Model / Output / Expand Complex and pick your solution of interest. For this structure it is the maximum response at 200 Hz and then we'll request that it is expanded into 12 solutions.

t Sets	Output Vectors
F. (2) (2) (3) (3)	Image: Constraint of the second sec
Mode 1, 200.4665 Hz	🙀 👫 📴 🏹 🐼 Ouick Filter 🛛None - Ignore 🔻 🗸
Mode 2, 380.1996 Hz	
Mode 3, 672.5712 Hz	2T1 Translation 70012SolidC1 Z Normal Stress 71011SolidC6 Y Normal Str
Mode 4, 1015.848 Hz	3T2 Translation 70013SolidC1 XY Shear Stress 71012SolidC6 Z Normal Str
Mode 5, 2022.406 Hz	4T3 Translation 70014SolidC1 YZ Shear Stress 71013SolidC6 XY Shear Str
Mode 6, 2625.677 Hz	6R1 Rotation 70015SolidC1 ZX Shear Stress 71014SolidC6 YZ Shear Str
Mode 7, 4156.374 Hz	7.R2 Rotation 70210SolidC2 X Normal Stress 71015SolidC6 ZX Shear Str
Mode 8, 4366.092 Hz	8R3 Rotation 70211SolidC2 Y Normal Stress 71210SolidC7 X Normal Str
Mode 9, 5601.611 Hz	22T1 Acceleration 70212SolidC2 Z Normal Stress 71211SolidC7 Y Normal Str
Mode 10, 6685.72 Hz	23T2 Acceleration 70213SolidC2 XY Shear Stress 71212SolidC7 Z Normal Str
Case 1 Freq 180.4198	24T3 Acceleration 70214SolidC2 YZ Shear Stress 71213SolidC7 XY Shear Str
Case 2 Freq 190.4432	26R1 Angular Acceleration 70215SolidC2 ZX Shear Stress 71214SolidC7 YZ Shear Str
.Case 3 Freq 200.4665	27R2 Angular Acceleration 70410SolidC3 X Normal Stress 71215SolidC7 ZX Shear Str
Case 4 Freq 210.4898	28R3 Angular Acceleration 70411SolidC3 Y Normal Stress 71410SolidC8 X Normal Str
Case 5 Freq 220.5131	250Complex Mode Shape 70412SolidC3 Z Normal Stress 71411SolidC8 Y Normal Str
Case 6 Freg 342.1797	6043Plate Top Fiber 70413SolidC3 XY Shear Stress 71412SolidC8 Z Normal Str
Case 7 Freg 361.1896	6044Plate Bottom Fiber 70414SolidC3 YZ Shear Stress 71413SolidC8 XY Shear Str
Case 8 Freg 380.1996	7020Plate Top X Normal Stress 70415SolidC3 ZX Shear Stress 71414SolidC8 YZ Shear Str
Case 9 Freg 399.2096	7021Plate Top Y Normal Stress 70610SolidC4 X Normal Stress 71415SolidC8 ZX Shear Str
Case 10 Freq 418.2196	7023Plate Top XY Shear Stress 70611SolidC4 Y Normal Stress 100007PltC1 Top Fiber
.Case 11 Freq 605.3141	7420Plate Bot X Normal Stress 70612SolidC4 Z Normal Stress 100008PltC1 Bottom Fiber
.Case 12 Freq 638.9426	7421Plate Bot Y Normal Stress 70613SolidC4 XY Shear Stress 100220PltC1 Top X Normal
.Case 13 Freq 672.5712	7423Plate Bot XY Shear Stress 70614SolidC4 YZ Shear Stress 100221PltC1 Top Y Normal
.Case 14 Freq 706.1998	60010Solid X Normal Stress 70615SolidC4 ZX Shear Stress 100223PltC1 Top XY Shear
.Case 15 Freq 739.8284	60011Solid Y Normal Stress 70810SolidC5 X Normal Stress 100620PltC1 Bot X Normal
.Case 16 Freq 914.2636	60012Solid Z Normal Stress 70811SolidC5 Y Normal Stress 100621PltC1 Bot Y Normal
.Case 17 Freq 965.0561	60013Solid XY Shear Stress 70812SolidC5 Z Normal Stress 100623PltC1 Bot XY Shear
.Case 18 Freq 1015.848	60014Solid YZ Shear Stress 70813SolidC5 XY Shear Stress 150007PltC2 Top Fiber
.Case 19 Freq 1066.641	60015Solid ZX Shear Stress 70814SolidC5 YZ Shear Stress 150008PltC2 Bottom Fiber
.Case 20 Freq 1117.433	70010SolidC1 X Normal Stress 70815SolidC5 ZX Shear Stress 150220PltC2 Top X Normal
	70011SolidC1 Y Normal Stress 71010SolidC6 X Normal Stress 150221PltC2 Top Y Normal

Expand For Single Phase Phase Range Increment 30. OK Cancel	Expand Complex Output	Data	X
Single Phase Last Phase 360. Phase Range Increment 30.	Expand For		
Phase Range Increment 30.		<u>F</u> irst Phase	0.
Increment 30.		<u>L</u> ast Phase	360.
OK Cancel	Phase Range	Increment	30.
		ОК	Cancel



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. PSD ANALYSIS (MODAL FREQUENCY WITH STATISTICS)

A PSD analysis is just a sophisticated and extremely useful form of the modal frequency analysis. Instead of having to interrogate multiple results sets, the PSD approach excites the structure using a broadspectrum acceleration load and then nicely sums up the solutions into one single-valued result. One can also think of it as a white noise or broad-band excitation where the structure is excited at all frequencies simultaneously. The approach is statistical and the displacement and stress results (and anything else) are termed 1- σ responses. The sigma (σ) refers to a Gaussian distribution where 1- σ to 3- σ refers to 68, 95 and 99.7% ranges. At 1- σ you have a 68% chance that the stresses are within this value. For many applications, one is required to use a 3- σ interpolation and thus your results are multiplied by 3x.

The actual theory behind the PSD approach is beyond this simple note but if one just considers that it is based on the response of the natural frequencies of the structure in the form of a modal frequency analysis and that the results are statistical quantities, you'll be in good shape for doing most basic PSD work.

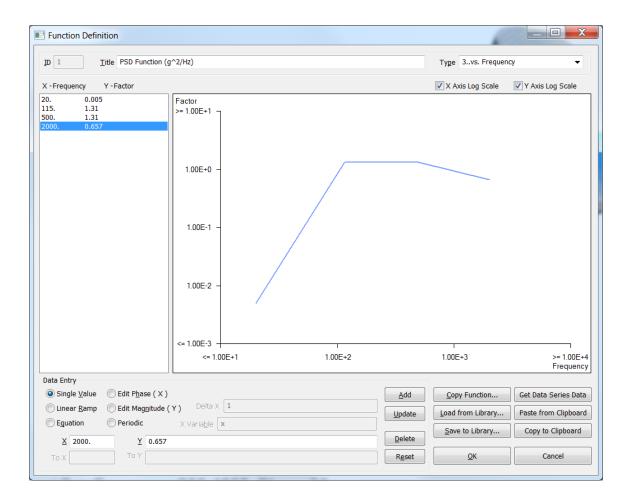
If you are interested in a tutorial just covering PSD analysis, then please take a read on our white paper:

PSD Random Vibration Tutorial for Femap and NX Nastran.pdf





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 g^2/Hz . In the center of the spectrum from 115 to 500 Hz, the PSD input is 1.31 g^2/Hz and then tapers.

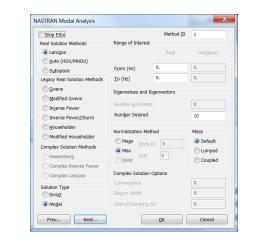




The PSD procedure is almost identical to the modal frequency analysis. There are some new screens but the only critical one is where you apply your PSD spectrum (lower-right-hand-corner). Otherwise, it is identical to that of the prior modal frequency analysis.

Analysis	Set		X
<u>T</u> itle	XXX-0814-01	Y-Axis PSD Analysi	s
Analysis	s <u>P</u> rogram	36NX Nastran	•
<u>A</u> nalysis	s Т у ре	6Random Respo	nse 🔻
		Run Analysis Us	ing VisQ
Ne	xt	<u>о</u> к	Cancel

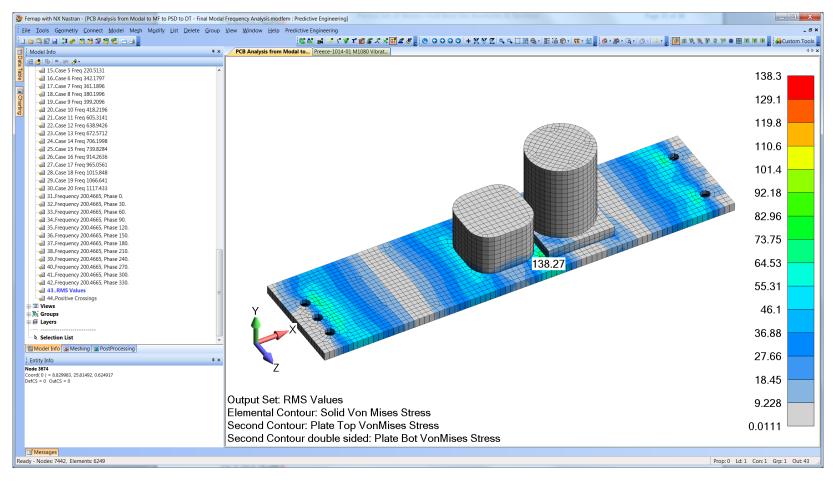
otions for Dynamic Analysis Advar	nced Options		
quivalent Viscous Damping		Limit Response Base	d on Modes
Overall Structural Damping Coeff (G) 0.	Number of Modes	
Modal Damping 3PSD Mod	lal Frequer 💌 🖍	Lowest Freq (Hz)	0.
As Structural (KDAMP)		Highest Freq (Hz)	0.
quivalent Viscous Damping Conver	sion	Transient Time Step	Intervals
Convert using Solution Freq (V	/MODAL)	Number of	0
Rigid Body Zero Modes(FZERO) 1.E-4	Time per	0.
Freq for System Damping (W3 - Ha	2) 0.	Output Interval	0
Freq for Element Damping (W4 -	0.		
requency Response		Response/Shock Spe	ctrum
Frequencies 5Modal Frequency	Table 🔻 🖍	Damping/Freq Corn	elation
Modal Freg		0None	- f _{xv}



NASTRAN P	ower Sp	ectra	I Density Factor	rs			X
Correlation ⁻	Table						
Master=>M	aster 1	.(1)	:Int1=0	Excited		Master	
				Load Set:	1	Notor Frequency	
				Applied		Master	
				Load Set:	1	Notor Frequency	
Edit Correla	ition Tabl	e					
	Factor		PSD Fund	tion		PSD Interpolation	
Real	1.	x	1PSD Function	(g^2/Hz)	•	0Log Log	•
Imaginary	1.	x	0None		•	0Log Log	-
	Ар	ply					
Prev	Next.			<u>о</u> к		Cancel	



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.





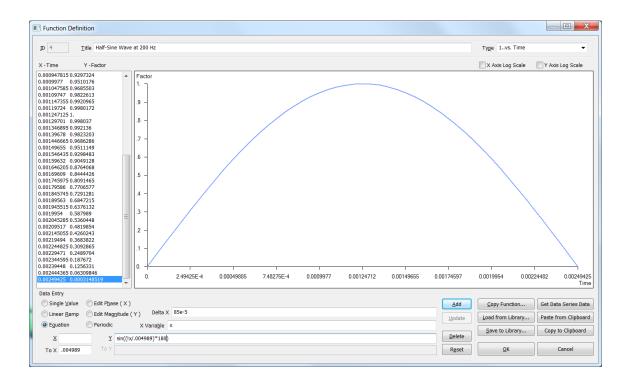


6. DIRECT TRANSIENT ANALYSIS

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:

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

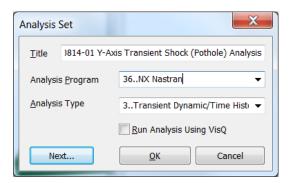
Our equation is developed in Femap using a sin((!x/0.004988)*) to180 create a 200.4 Hz half-sine wave:

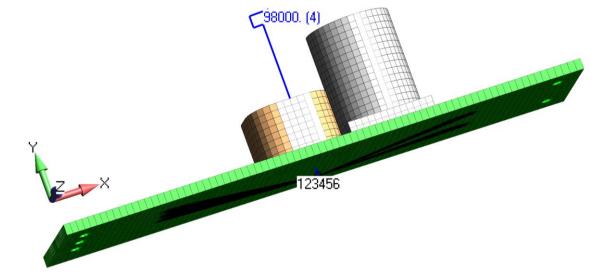






For our work, we are just going to use half the sin wave to give the system a shock pulse. The load for this analysis is 100 g (98,000 mm/s²) with our half-sine function at 200.4 Hz.









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

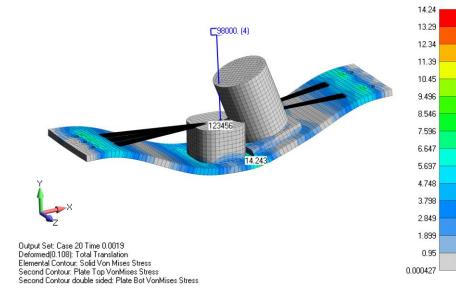
NASTRAN Modal Analysis			×
Skip EIGx		Method <u>I</u> D	1
Real Solution Methods	Range of Interest		
Lanczos		Real	Imaginary
<u>Auto (HOU/MHOU)</u>			
Subspace	F <u>r</u> om (Hz)	0.	0.
Legacy Real Solution Methods	<u>T</u> o (Hz)	0.	0.
© <u>G</u> ivens	Eigenvalues and Ei	genvectors	
Modified Givens	Number Estimated		0
Inverse Power			
Inverse Power/Sturm	Num <u>b</u> er Desired		10
O <u>H</u> ouseholder	Normalization Meth	bod	Mass
Modified Householder	Macc		Default
Complex Solution Methods	Mass Node I	DO	C Lumped
Hessenberg	DOF	0	<u> </u>
Complex Inverse Power	O Point –		Coupled
Complex Lanczos	Complex Solution (Options	
- ·	Convergence		0.
Solution Type	Region Width		0.
 Modal 	Overall Damping (G)	0.
Prev Next		<u>о</u> к	Cancel

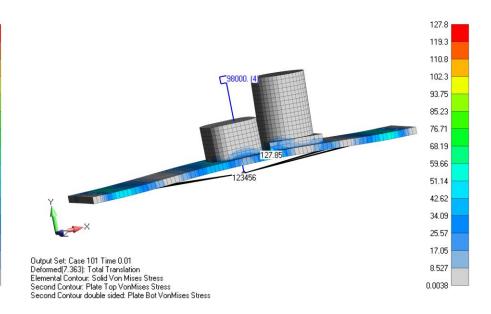
Dynamic Control Options		X	
Use Load Set Options			
Options for Dynamic Analysis Advanced Options			
Equivalent Viscous Damping	Limit Response Base	d on Modes	
Overall Structural Damping Coeff (G) 0.	Number of Modes	0	
Modal Damping 3PSD Modal Frequer 👻 🖡	Lowest Freq (Hz)	0.	
As Structural (KDAMP)	Highest Freq (Hz)	0.	
Equivalent Viscous Damping Conversion	Transient Time Step Intervals		
Convert using Solution Freq (WMODAL)	Number of	1000	
Rigid Body Zero Modes(FZERO)	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	ectrum	
Frequencies 0None	Damping/Freq Correlation		
Modal Freq	0None	▼ f _{xy}	
Prev Next	<u>O</u> K	Cancel	





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









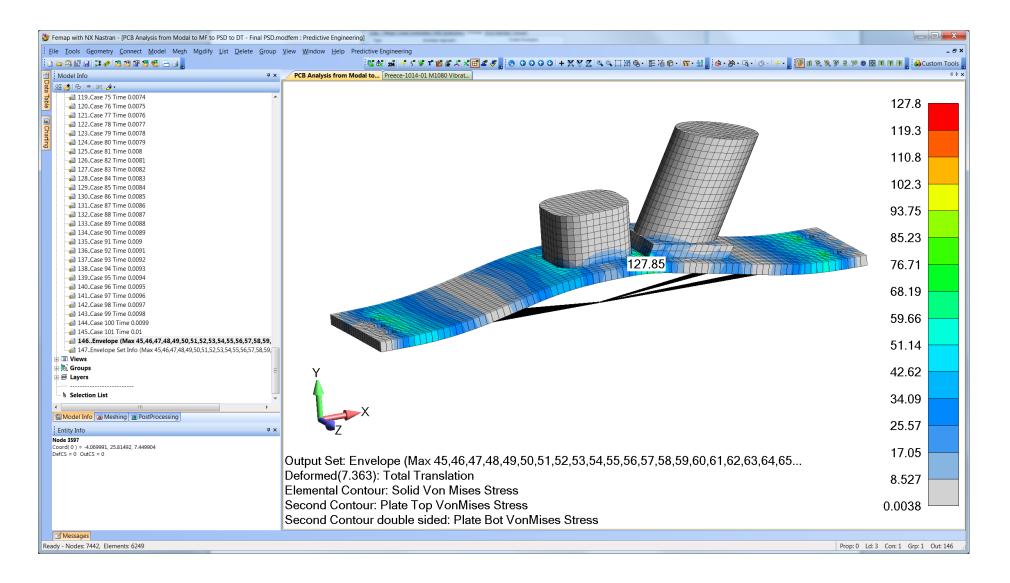
A much simpler way to process transient results is to use the Femap envelope function and then select all the output sets with "time" in the title:

Filter Filter Title Contains time OK Cancel							
Process Output Data							
What to Process							
Complete Output Sets One or More Selected Output Vectors							
Processing Operations							
Copy Merge Linear Combination RSS Combination Envelope Error Estimate Convert							
Type Envelope Approach Create Envelopes							
O Max Value O Envelope All Selected Vectors Within Output Sets							
Omin Value Omin Envelope All Locations For Each Vector Image: Control of the second							
Max Absolute Value							
Envelope Every Vector across Multiple Output Sets							
Select Output To Process Store Output in Set 0New Output Set							
Operations That Will Be Processed - Review Before Pressing OK							
Max 45Case 1 Time 0> New Set Envelope Each Vector Across Sets with SetInfo Max 46Case 2 Time 1.E-4	Reset						
Max 47Case 3 Time 0.0002 Max 48Case 4 Time 0.0003	Delete						
Max 49Case 5 Time 0.0004	E Delete						
Max 50Case 6 Time 0.0005 Max 51Case 7 Time 0.0006							
Max 52Case 8 Time 0.0007 Max 53Case 9 Time 0.0008							
Max 54Case 10 Time 0.0009							
Max 55Case 11 Time 0.001 Max 56Case 12 Time 0.0011							
Max 57Case 13 Time 0.0012 Max 58Case 14 Time 0.0013							
Max 59Case 15 Time 0.0014							
Max 60Case 16 Time 0.0015 Max 61case 17 Time 0.0016							
Max 62Case 18 Time 0.0017 Max 63Case 19 Time 0.0018							
Max 64Case 20 Time 0.0019							
Max 65Case 21 Time 0.002 Max 66Case 22 Time 0.0021							
Max 67Case 23 Time 0.0022 Max 68Case 24 Time 0.0023	<u>О</u> К						
Max 69Case 25 Time 0.0024							
Max 71. Cose 26 Time 0.0025	- Cancel						





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





7. 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.

Question: (We'll add additional material here from questions asked during the Seminar)



8. BEING AN EXPERT: VIBRATION IS ABOUT MASS AND CONSTRAINTS

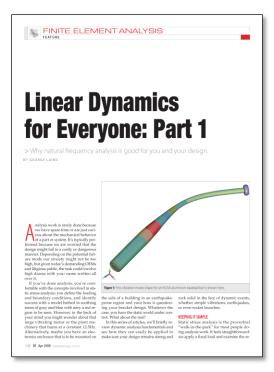
8.1 CHECK F06 FOR MASS SUMMATION AND KNOW WHAT YOU KNOW

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 "OUPUT FROM GRID POINT WEIGHT GENERATOR"?
- o Error and Warning Messages?

Vibration Analysis White Paper



For more information see our Seminar: Normal Modes Analysis



8.2 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 Check			
DOF SET	G	🖻 <u>F</u>	DOF SET	<u>G</u>	E	
	<u>N</u>	<u>A</u>		<u>N</u>	<u>A</u>	
	N+AUTOSPC	<u>∎</u> ⊻	N+AUTOSPC			
				F	Print Forces Abov	e
\Box <u>C</u> GI (Center of Gravity)		DATAREC		10	%	
Re <u>f</u> Node	0		Ref Nod <u>e</u>		0	
Units	0W	/eight 👻	Max Strajn E	Energy	0.	
Pre	v Next.			<u>)</u> K	Cancel	

For more information see our Seminar: What is Groundcheck?



9. ADDITIONAL READING

- Linear Dynamics for Everyone.pdf
- Femap and NX Nastran Technical Seminar on Vibration Analysis for Engineers.pdf
- PSD Random Vibration Tutorial for Femap and NX Nastran.pdf
- NX Nastran Dynamic Analysis.pdf
- Vibration Analysis for Electronic Equipment, Dave S. Steinberg



10. TRAINING OPPORTUNITIES

Femap & NX Nastran Training

Foundation | Advanced | Customization



When: October 20-24, 2014 (Monday-Friday)

Where: Portland, Oregon

Cost: Foundation and Advanced training (Mon-Thurs) is \$2,100 per student with the optional Customization/API training on Friday for an additional \$525.

What's Included: Course manual and workshop videos saved to a USB stick. A lunch and social event are provided to encourage class interaction with fellow users.

Registration: Early registration is encouraged since space is limited to 18 students and it is expected that the class will fill. *To register please send email to:*

> Training@PredictiveEngineering.com Attn: George Laird, PhD, PE

About Predictive Engineering

Based in Portland, Oregon, Predictive has more than 18 years of experience with Femap, Nastran and LS-DYNA and is well known as the "go-to-company" for Femap training. References can be found on our website: www.PredictiveEngineering.com





Welcome Femap and NX Nastran Colleague,

This week-long course taught by **Predictive Engineering** will take the new user from ground floor through FEA best practices to advanced subjects dealing with manifold and non-manifold surface modeling, detailed plate meshing and tet versus hex meshing. The final day will finish with a focus on customization and automation using Excel and Femap's own API interface. The course will be fast paced and follow a workshop format with theory, practice and Q&A sessions.

Course Outline

Foundation of FEA Modeling with Femap + NX Nastran (Two Days)

- I. FEA theoretical background w.r.t Beam, Isoparametric and special elements
- II. Tour of Fernap interface: Preferences, Panes, Toolboxes, Help and Tips & Tricks
- III. Femap modeling workflow for Beam, Plate and Solid (BPS) elements
- IV. Static stress analysis and results interpretation of BPS elements
- V. Introduction to Plate and Solid modeling with surface and solid geometry and Mesh Toolbox
- VI. Introduction to Assembly Modeling: Glued, Contact and Rigid element Usage

Advanced Femap + NX Nastran (Two Days)

- I. Surface modeling using Manifold and Non-Manifold geometries
- II. Advanced surface preparation for high-accuracy Plate modeling
- III.
 Meshing toolbox tips and tricks with Jacobian optimization

 IV.
 Building efficient assemblies via efficient Solid modeling (tet & hex elements) and Linear Contact
- V. Introduction to linear dynamics (modal analysis tips & tricks)
- VI. Non-linear analysis: geometric versus material non-linearity

Customization & Automation of Femap (One Day)

- Automation of results processing via Excel
- Introduction to Femap's macro capability
- III. Introduction to Femap's API via Custom Tools
- IV. Programming Femap's API

and best practices

03-206-5571 | Training@PredictiveEngineering.com | www.PredictiveEngineering.com

11

