

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
- o PSD Analysis
 - ° Modal Frequency Analysis using Statistics
 - ° One Sigma versus Three Sigma von Mises Results
 - ° Fatigue Analysis with PSD
- o Direct Transient Analysis (it is easier than you think)





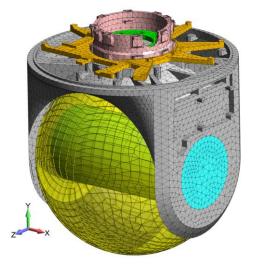
1. INTRODUCTION TO LINEAR DYNAMICS (NX NASTRAN)

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



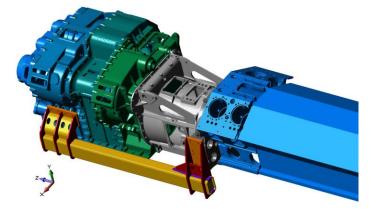
Linear FEA Model (courtesy Predictive Engineering)



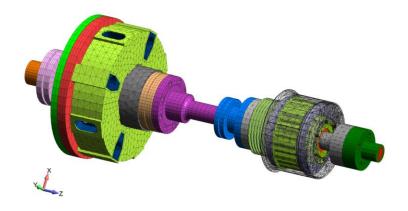


1.1 SOME EXAMPLES OF LINEAR VIBRATION ANALYSIS (PREDICTIVE)

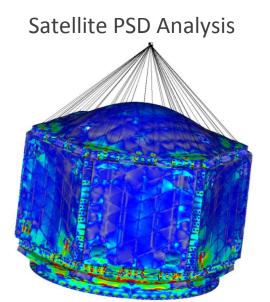
3,000 HP Transmission

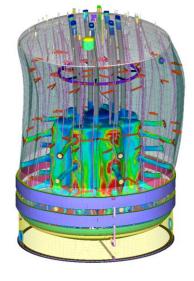


Drive Train Coupling



Seismic Analysis of Large Vessels



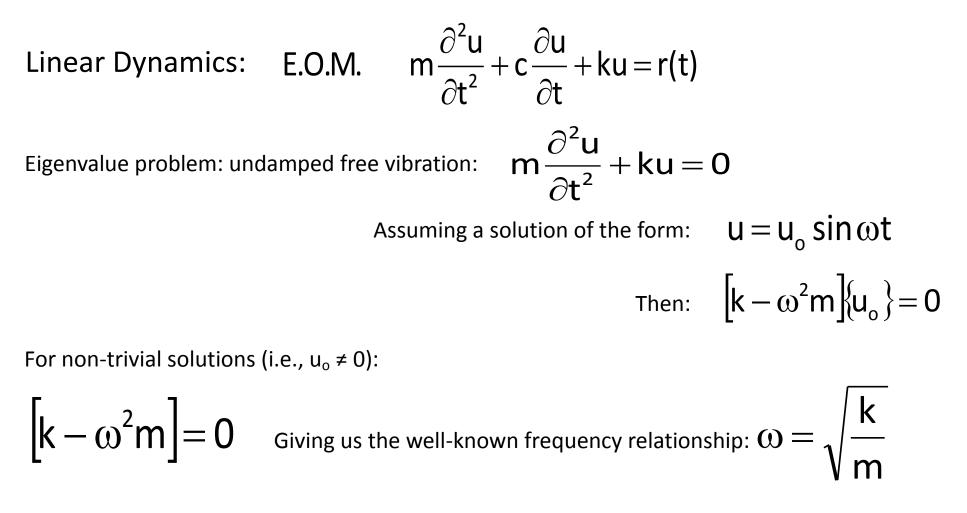




Applied CAx / Predictive Engineering White Paper – Please share with your Friends

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:





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:

$$Ku = F$$



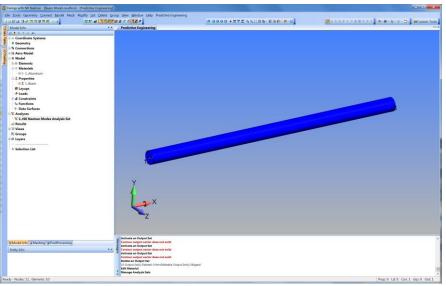
2. FOUNDATION OF FREQUENCY ANALYSIS

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)

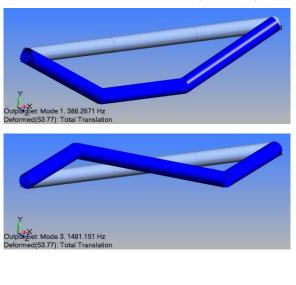
Youngs Modulus, E 10300000. Tension 0. Shear Modulus, G 0. Compression 0. Poisson's Ratio, nu 0.33 Shear 0.	Youngs Modulus, E 10300000. Shear Modulus, G 0. Poisson's Ratio, nu 0.33 Thermal Shear Expansion Coeff, a 0. Conductivity, k 0. Specific Heat, Cp 0.		nces Nonlinear Ply/Bond Failure		ptical Phase
Shear Modulus, G 0. Poisson's Ratio, nu 0.33 Compression 0. Shear 0. Thermal 0. Expansion Coeff, a 0. Conductivity, k 0. Specific Heat, Cp 0. Reference Temp 0.	Shear Modulus, G 0. Compression 0. Poisson's Ratio, nu 0.33 Shear 0. Thermal Expansion Coeff, a 0. Mass Density 2.64243E-4 Conductivity, k 0. Damping, 2C/Co 0. Specific Heat, Cp 0. Reference Temp 0.	Stiffness		Limit Stress	
Poisson's Ratio, nu 0.33 Shear 0. Thermal Expansion Coeff, a 0. Mass Density 2.64243E-4 Conductivity, k 0. Damping, 2C/Co 0. Specific Heat, Cp 0. Reference Temp 0.	Poisson's Ratio, nu 0.33 Shear 0. Thermal Expansion Coeff, a 0. Mass Density 2.64243E-4 Conductivity, k 0. Damping, 2C/Co 0. Specific Heat, Cp 0. Reference Temp 0.	Youngs Modulus, <u>E</u>	10300000.	Tension	0.
Thermal Expansion Coeff, a 0. Conductivity, k 0. Specific Heat, Cp 0. Reference Temp 0.	Thermal Expansion Coeff, a 0. Conductivity, k 0. Specific Heat, Cp 0. Reference Temp 0.	Shear Modulus, <u>G</u>	0.	Compression	0.
Expansion Coeff, a 0. Mass Density 2.64243E-4 Conductivity, k 0. Damping, 2C/Co 0. Specific Heat, Cp 0. Reference Temp 0.	Expansion Coeff, a 0. Mass Density 2.64243E-4 Conductivity, k 0. Damping, 2C/Co 0. Specific Heat, Cp 0. Reference Temp 0.	Poisson's Ratio, n <u>u</u>	0.33	Shear	0.
Conductivity, k 0. Specific Heat, Cp 0. Mass Density 2.64243E-4 Damping, 2C/Co 0. Reference Temp 0.	Conductivity, k 0. 2.64243E-4 Specific Heat, Cp 0. 0. Reference Temp 0.	Thermal			
Conductivity, k 0. Specific Heat, Cp 0. Reference Temp 0.	Conductivity, k 0. Specific Heat, Cp 0. Bamping, 2C/Co 0. Reference Temp 0.	Expansion Coeff, <u>a</u>	0.	Mara Daraika	2 64243E-4
Specific Heat, Cp 0. Reference Temp 0.	Specific Heat, Cp 0.	Conductivity, <u>k</u>	0.		
		Specific <u>H</u> eat, Cp	0.		
		Heat Generation Factor	0.	Reference Temp	0.





2.1.1 HERE'S YOUR MODAL ANALYSIS CHECKLIST

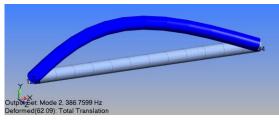
- o Elastic and mass properties are in consistent units
 - [°] The weight of your structure can be checked by summing the mass of the model and multiplying it by gravity (for US units of lbf, inch and seconds, it would be 386 in/s²)
- o FEA model with a sufficient mesh density to capture the frequencies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

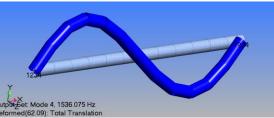


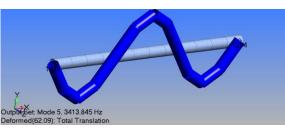
Doesn't Exist

Three Element Mesh (Coarse)

Twelve Element Mesh (Fine)











2.1.2 SETTING UP THE MODEL FOR NORMAL MODES WITH MASS PARTICIPATION

Title NX Nastran	Modes Analysis Set		
Analysis <u>P</u> rogram	36NX Nastran	-	
<u>A</u> nalysis Type	2Normal Modes/Eig	envalue 🔻	
	Run Analysis Using) VisQ	
Next	<u>O</u> K	Cancel	
nalysis Set Manager (Active	e: 1NX Nastran Modes And	alysi	
Analysis Set : 1NX Nastran M	1odes Analysis Set	Analyze	
Solver : NX Nastran Type : Normal Modes/Eige	nvalue	Analyze Multiple	
Analyze : Local			
Executive/Solution		Export	
Bulk Data GEOMCHECK		Ac <u>t</u> ive	
		Active Preview Input	
GEOMCHECK MODELCHECK Modal/Buckling DDAM Rotor Dynamics Modal XYPlot		Preview Input	
GEOMCHECK MODELCHECK DDAM Rotor Dynamics Modal XYPIot Not Defined Response Spectrum App		Preview Input	
GEOMCHECK MODELCHECK Modal/Buckling DDAM Rotor Dynamics Modal XYPIot - Not Defined Response Spectrum App External Superelement R	eference	Preview Input MultiSet Copy	
GEOMCHECK MODELCHECK Dodal/Buckling DDAM Rotor Dynamics Modal XYPIot Not Defined Response Spectrum App	eference	Preview Input MultiSet Copy	
GEOMCHECK MODELCHECK Modal/Buckling DDAM Rotor Dynamics Not Defined Response Spectrum App External Superelement R Master Requests and Condi	eference	Preview Input MultiSet Copy	
GEOMCHECK MODELCHECK Modal/Buckling DDAM Rotor Dynamics Not Defined Response Spectrum App External Superelement R Master Requests and Condi	eference	Preview Input MultiSet Copy Delete Save	
GEOMCHECK MODELCHECK Modal/Buckling DDAM Rotor Dynamics Not Defined Response Spectrum App External Superelement R Master Requests and Condi	eference	Preview Input MultiSet Copy Delete	
GEOMCHECK MODELCHECK Modal/Buckling DDAM Rotor Dynamics Not Defined Response Spectrum App External Superelement R Master Requests and Condi	eference	Preview Input MultiSet Copy Delete Save	

ASTRAN Modal Analysis			×			
Skip EIGx Real Solution Methods	Range of Interest	Method <u>I</u> D	1			
Lanczos <u>Auto (HOU/MHOU)</u>	From (Hz)	Real	Imaginary			
Subspace Legacy Real Solution Methods Givens	F <u>r</u> om (Hz) To (Hz)	0.	0.			
 Modified Givens Inverse Power Inverse Power/Sturm 	Eigenvalues and Eig Number Estimated Number Desired	-	0			
 <u>H</u>ouseholder Modified Householder 	Normalization Meth		Mass Oefault			
Complex Solution Methods Hessenberg Complex Inverse Power 	Max Point	0	Coupled			
Complex Lanczos Solution Type Direct Modal	Complex Solution C Convergence Region Width Overall Damping (0.			
Prev Ne <u>x</u> t		<u>O</u> K	Cancel			
NASTRAN XY Output for Modal Analysis Output Requests Summary Modal Participation Factors Modal Effective Mass						
Modal Effective	-					
Modal Effective	Mass Fraction					

Reference <u>N</u>ode

0

<u>O</u>K

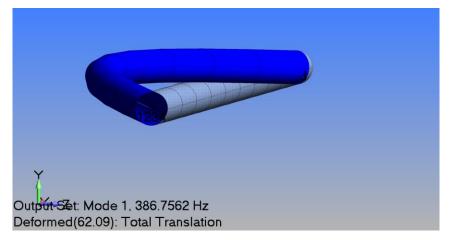
Cancel



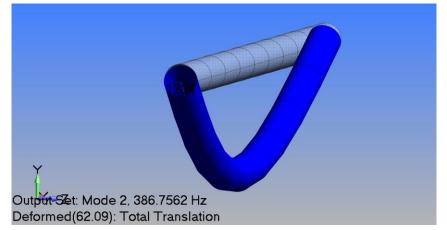
2.1.3 INTERPRETING RESULTS BASED ON ORTHOGONALITY AND MASS PARTICIPATION

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



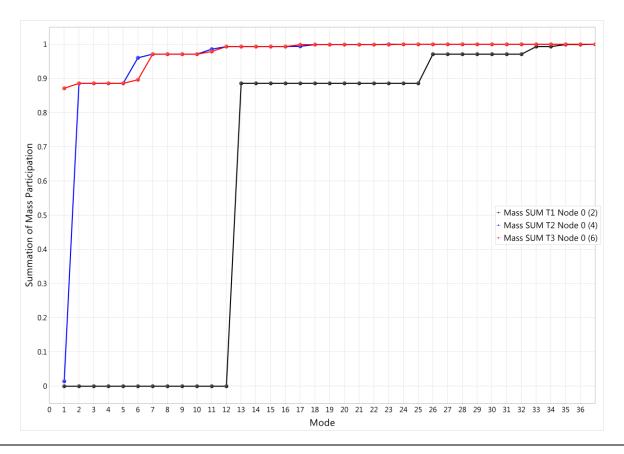
Second Frequency 386.8 Hz







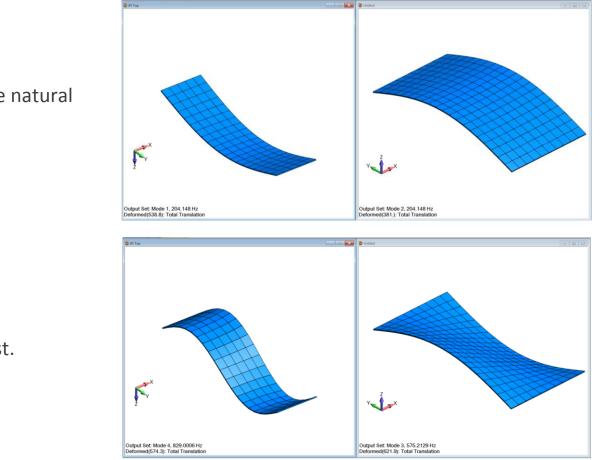
Mass participation tells you which modes have the "umph" and how many frequencies you need (modes) to accurately capture the dynamic response of the structure. On this later subject, a modal frequency analysis (e.g., PSD) formulates its response based on the number of modes chosen for the analysis. To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

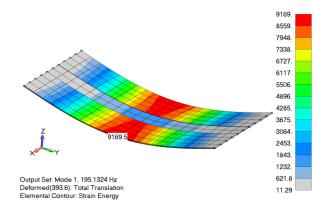
If one wants to move your natural frequencies up or down, sometimes intuition is good enough but it never hurts to have a quantitative tool. When a structure flexes or vibrates, there will be regions within the structure that are deforming more and other regions less. Since a natural frequency analysis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

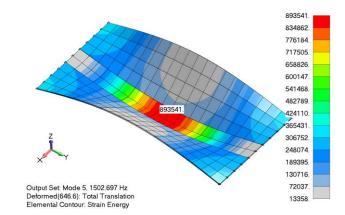
The Default Setup

Nodal			Elemental		
Displacement 0Full Model		•	Eorce	0Full Model	
Applied Load	0Full Model		Stress	0Full Model	
Constraint Force	0Full Model	-	Strain	0Full Model	
Equation Force	0Full Model		Strain Energy	0Full Model	•
Force Balance	0Full Model		Heat Flug	0Full Model	
Velocity	0Full Model		Epthalpy	0Full Model	
Acceleration	0Full Model		Enthalpy Rate	0Full Model	. v
Kinetic Energy	0Full Model		Temperature	0Full Model	
Iemperature	0Full Model	٠	Kinetic Energy	0Full Model	Y
			Energy Loss	0Full Model	
			Fluid Pressure	0Full Model	
Customization					
Element Corner Re	sults	Re	esults Destination		Prev
Output Modes (a,b,c	THRU d)		2PostProcess Only	-	
			ho Model		QK
Magnitude/Phase	Real/Imagina	iχ			Cancel

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).

Fernap with NX Nastran - (PCB Analysis from Modal to MF to PSD to DT.mo Elle Tools Geometry Connect Model Megh Mgdify List Delete Geo		Define Materia	- ISOTROPIC		X
	변화가 (편화) 0 0 0 0 + XYZ 5 5 126 + 분유한 명·권 변화 5 7 17 4 분위한 명·권 · 현·관· 이 · · · · · · · · · · · · · · · · ·				
	COX				
a a to the attention of	PLD Analysis from No. APR 0514 02 VERSION.	ID 1	Title FR4 PCB	Color 55 D.L.U.	
A Coordinate Systems		<u>I</u> O I	Title FR4 PCB	Color 55 Palette.	Layer 1 Type
8 + 0 Geometry					
Connections			1		
Arro Model		General Funct	ion References Nonlinea	ar Ply/Bond Failure Creep Electrical	Optical Phase
Model					
Elements	ATTACT.	Stiffness		Limit Stress	
B 4 Materials					
一回ル 1.FR4 PCB		Youngs Mo	dulus, <u>E</u> 21000.	Tension	0.
·III 4 2.Big Dog		roungs no	21000.	Tension	0.
21 4. 3_Little Dog					
E Properties		Shear Modu	ilus, <u>G</u> 0.	Compression	0.
-WII 1.FR4 PCB				Compression	·
-∉1 2.8g Dog		Poisson's R	atio, n <u>u</u> 0.15	C	0.
-WI 3.Little Dog		POISSOITS IN	0.15	Shear	0.
📾 Layups					
🕀 🕈 Loads		Thermal			
⇒ ♦ 1.Y-Axis PSD Driver Load ⇒ *= Load Definitions		Therma			
ter 1⊂ Load Definitions					
e Other Loads		Expansion (Coeff, a 0.		
E & 2Y-Axis Transient Shock Load				Mass Density	3.13559E-9
Constraints	122456	Conductivit	v.k 0.	Huss Density	
# # 1Universal All-Purpose Constraint Set	Y 123456	Conductivit	y, <u>b</u> 0.		0.
⊖ & Functions				Da <u>m</u> ping, 2C/C	0.
Sector 1. PSD Profile Table 27 GMW3172 2012		Specific <u>H</u> ea	at, Cp 0.		
4y 2.PSD Driver Function over Frequency				Reference Tem	n 0.
4 3.PSD Modal Frequency Critical Damping		Heat Gener	ation Factor 0.	Rejerence rem	
- 4 A.Transient Shock (Pothole) from Table 30, GMW3172 2012		Theat Gener	0.		
-tr Data Surfaces					
⊖ S Analyses	Z				
S 1.XXX-0814-01 Normal Modes Analysis					
\$2,XXX-0814-01 Y-Axis PSD Analysis \$3,XXX-0814-01 Y-Axis Transient Shock (Pothole) Analysis					
3XXX-0814-01 T-Axis Transient Shock (Pothole) Analysis					
I Views					
R Groups					
III E Layers					
	Loading 54 Partis.				
A Selection List	Vorur and specific Neinskein Konstentiering (Daulie Buffereit)				
	Performance Graphics is enabled				
	²⁶ Eff Load Definition Eff Load Definition				
	Cati Load Definition				
Mi Model Info a Meshing a PostProcessing	1 Nodejs) Selected.				
	L Loadiji Creand. Delete an Analysis Set				
Entity Info **	2 Analysis Set(s) Deleted. 0 NonDeletable Analysis Set(s) Skipped.				
Node 1 Coord(0) = -9.219982 21.94487 -8.128E-5	Renumber Analysis Sees Rebuilt Model				
DefCS = 0 OutCS = 0	Requiring Update of Database.				
CANAD (2008) (27 (2017) (247)	Database Update Completed. No Empre.				
	Sinning Discharger	fxy L0	ad Save	Copy	OK Cancel
eady - Nodes: 7442, Elements: 6249	Prop.0 Ldt.2 Com 1 Grp 1 Out 115				

Femap v11.1.2A Model

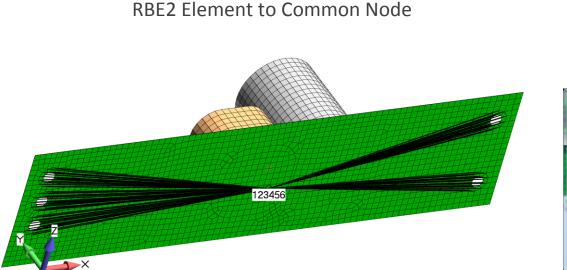
Units: N, mm, Tonne, s





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 Cursor ID 10427 Color 24576 Palette Layer 1 Prop	erty Type
DOF 484 ✓ TX RX ✓ TY RY ▲ 485 ✓ TY RY ▲ 488 489 ✓ TZ RZ Delete 491	Independent Node 1 New Node At Center
Thermal Expansion Coefficient 0. Material Single F Com	OK Cancel





3.2 NATURAL FREQUENCY RESULTS AND INTERPRETATION

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

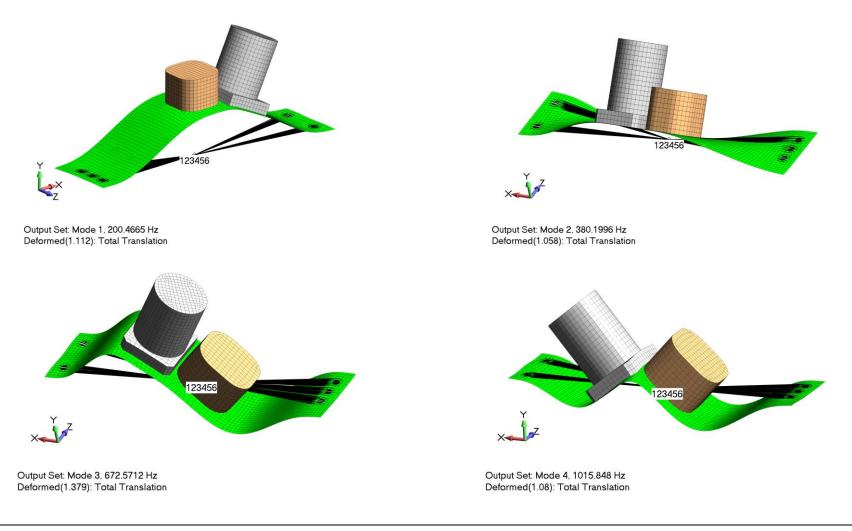
A normal modes / Eigenvalue analysis is the starting point for all linear dynamics work. It is simple to setup but difficult to interpret the results.

Boundary Conditions	X	NASTRAN Modal Analysis	Cardinana -		X	Nastran Output Requ	ests	-		-	
Primary Sets Constraints	1Universal All-Purpose Con: 🔻	Real Solution Methods	Range of Interest	Method ID	1	Nodal			Elemental		
Loads	0None	Lanczos Auto (HOU/MHOU)	_	Real	Imaginary	Displacement	0Full Model	• •	Force	0Full Model 0Full Model	
Temperatures Initial Conditions	0From Load Set	Subspace	From (Hz) 0. To (Hz) 0.		0.	Constraint Force	0Full Model	•	Strain	0Full Model	
Constraint Equations	0From Constraint Set	Givens Modified Givens	Eigenvalues and Eigenv		0	Equation Force	0Full Model 0Full Model	• •	Strain Energy	0Full Model 0Full Model	
Bolt Preloads	0From Load Set 🔹	Inverse Power Inverse Power/Sturm Householder	Number Desired		10	Velocity	0Full Model	•	Enthalpy	0Full Model	
Other DOF Sets Master (ASET)	0None	Modified Householder	Normalization Method	0	Mass Default	Kinetic Energy	0Full Model 0Full Model	*	Enthalpy Rate	0Full Model 0Full Model	
Kinematic (SUPORT)	0None 👻	Complex Inverse Power	Max DOF Point	0	Coupled		0Full Model	Ŧ	Kinetic Energy	0Full Model	
SUPORT1	0None	Complex Lanczos	Complex Solution Option		0.				Fluid Pressure	0Full Model 0Full Model	
QSET	0None 👻	Direct Modal	Region Width Overall Damping (G)		0.	Customization	sults	Re	sults Destination	Pr	ev
CSET	0None	Prev Next		ок	Cancel	Output Modes (a,b,c		_	2PostProcess Only	• _	
Prev Next	OK Cancel					Magnitude/Phase	Real/Imaginary		no Model		OK ancel
						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.





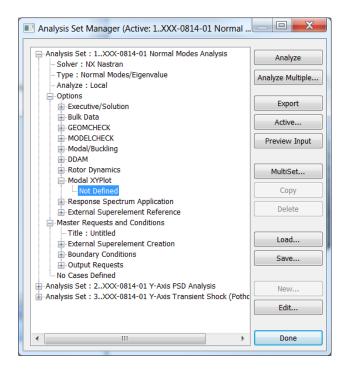
Applied CAx / Predictive Engineering White Paper – Please share with your Friends



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



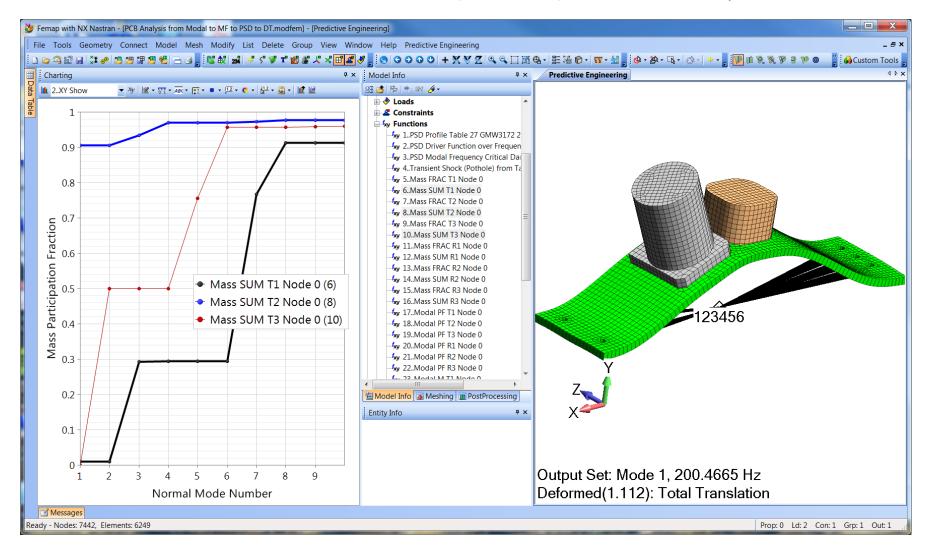
With the Not Defined item highlighted, hit the Edit button and the NASTRAN XY Output for Modal Analysis dialog box will appear. This box requests the mass participation factors.

NASTRAN XY Outp	ut for Modal Analysis
Output Requests	
Summary	
Modal Partici	pation Factors
Modal Effectiv	ve Mass
Modal Effectiv	/e Weight
Modal Effectiv	e Mass Fraction
Reference Node	0
	OK Cancel



FINITE ELEMENT ANALYSIS Predictive Engineering

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	Set	X
<u>T</u> itle	XXX-0814-01	Modal Frequency Analysis
Analysis	s <u>P</u> rogram	36NX Nastran 👻
<u>A</u> nalysis	s Туре	4Frequency/Harmonic Respons 🔻
		Run Analysis Using VisQ
Ne	xt	OK 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.

E Function Definition		1000	
ID 3 Title PSD Modal Frequency Critical Dampin	Ig	Type 7Critical Dam	p vs. Freq 💌
X - Frequency Y - Frac Crit		X Axis Log Scale	Y Axis Log Scale
10. 0.02 Frac Crit 20000. 0.02 0.4 0.35 0.25 0.2 0.15 0.11 0.05 0.1 10.05 0.1 10. 334: 10. 334:	.667 6673.333	10005. 13336.67	16668.33 20000. Frequency
Data Entry ● Edit Phase (X) ● Linear Bamp ● Edit Magnitude (Y) ● Equation ● Periodic X 10. Y 0.02	<u>A</u> dd Update Delete	<u>C</u> opy Function Load from Library Save to Library	Get Data Series Data Paste from Clipboard Copy to Clipboard
To X To Y	Reset	<u>Q</u> K	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:

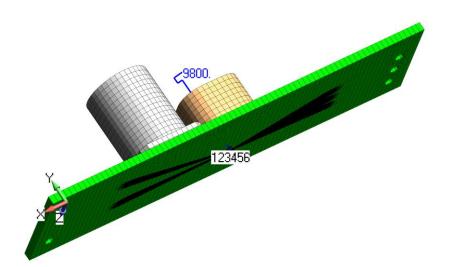
NX Nastran Dynamic Analysis.pdf





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
	<u>V</u> alue	Time/Freq Dependence	Data Surface
	AX 🔲 0.	0None 🔻 🖍	V ONone 💌 🧾
	AY 📝 9800		0None 💌 🧮
	AZ 🔲 0.		0None 💌 🔳
	Phase 0.	0None 🔻 🔩	
		[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).

ASTRAN 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 Eigenvalues	envectors	
O Modified Givens			
Inverse Power	Number Estimated		0
Inverse Power/Sturm	Num <u>b</u> er Desired		10
O Householder			
Modified Householder	Normalization Meth	od	Mass
Complex Solution Methods	() Ma <u>s</u> s Node I	0	Oefault
Hessenberg	Max DOF	0	C Lumped
Complex Inverse Power	O Point		Coupled
	Complex Solution (options	
Complex Lanczos	Convergence		0.
Solution Type	Region Width		0.
) Dire <u>c</u> t	2		
Modal	Overall Damping (G)	0.
Prev Next		<u>o</u> k	Cancel

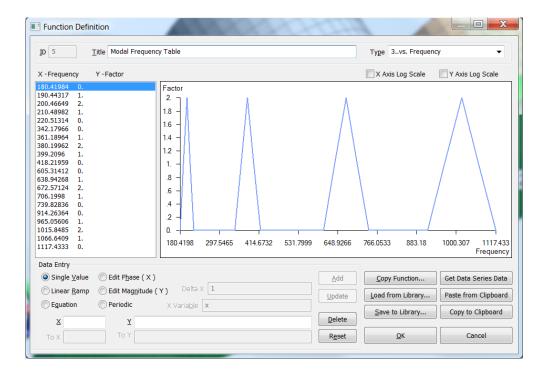
Dynamic Control Options			X
Use Load Set Options			
Options for Dynamic Analysis Advanced	Options		
Equivalent Viscous Damping		Limit Response Base	d on Modes
Overall Structural Damping Coeff (G)	0.	Number of Modes	0
Modal Damping 3PSD Modal F	Frequer 🔻 fxy	Lowest Freq (Hz)	0.
As Structural (KDAMP)		Highest Freq (Hz)	0.
Equivalent Viscous Damping Conversion	Equivalent Viscous Damping Conversion		
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 - Hz)	Freq for System Damping (W3 - Hz)		0
Freq for Element Damping (W4 -	0.		
Frequency Response		Response/Shock Spe	ctrum
Frequencies	▼ f _{xy}	Damping/Freq Correlation	
Modal Freq		0None	▼ f _{xy}
Prev Next		<u>O</u> K	Cancel



FINITE ELEMENT ANALYSIS Predictive Engineering

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 Fr	rom Modal Results		x	
Modal Results				
<u>F</u> irst Freq	1Mode 1, 200.4665 Hz		•	
<u>L</u> ast Freq	4Mode 4, 1015.848 Hz		•	
Additional Solution Frequency Points				
Number of Points per Existing Mode 5				
Frequency Band Spread (+/-) 10. %			%	
OK Cancel				







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

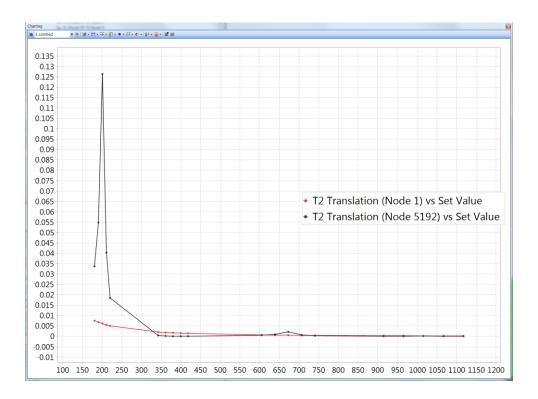
Boundary Conditions	×		
Primary Sets			
<u>C</u> onstraints	1Universal All-Purpose Con: 🔻		
<u>L</u> oads	1Motor Frequency		
Tem <u>p</u> eratures	0From Load Set		
Initial Conditions	0None		
Constraint <u>E</u> quations	0From Constraint Set 👻		
Bolt Preloads	0From Load Set 🔹		
Other DOF Sets			
M <u>a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

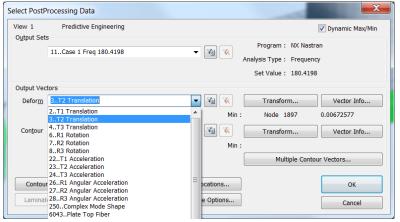
Nodal			Elemental	
Displacement	0Full Model	•	Eorce	0Full Model
Applied Load	0Full Model	-	Stress	0Full Model 🗸
Constraint Force	0Full Model	-	St <u>r</u> ain	0Full Model
Equation Force	0Full Model	Ŧ	Strain Energy	0Full Model
Force <u>B</u> alance	0Full Model	Ŧ	Heat Flux	0Full Model
Velocity	0Full Model	Ŧ	Enthalpy	0Full Model
Acceleration	0Full Model	•	Enthalpy Rate	0Full Model
Kinetic Energy	0Full Model	•	Temperature	0Full Model
	0Full Model	-	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	•
Echo Model			ОК	
Magnitude/Phase	🔘 Real/Imagina	ry		Cancel



FINITE ELEMENT ANALYSIS Predictive Engineering

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.









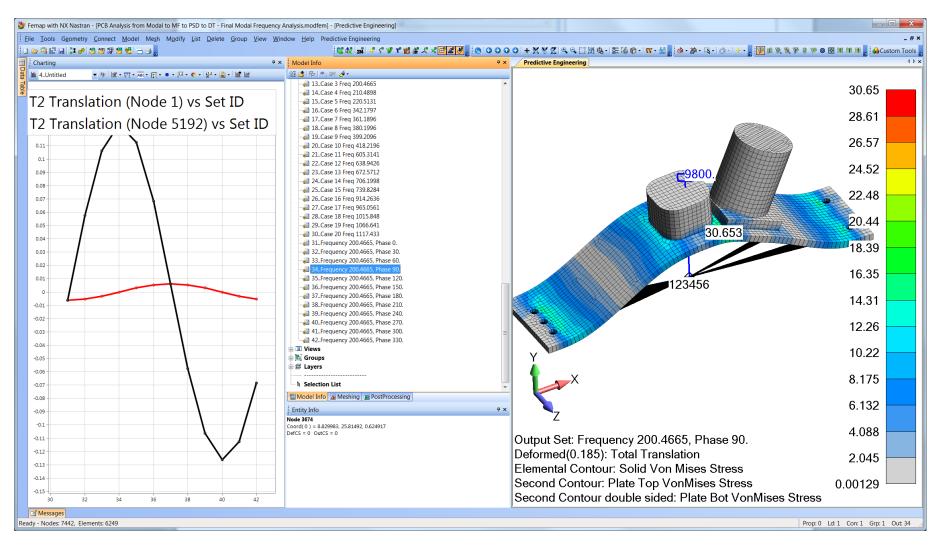
To obtain the time varying response from a Modal Frequency, one goes to Model / Output / Expand Complex and pick your solution of interest. For this structure it is the maximum response at 200 Hz and then we'll request that it is expanded into 12 solutions.

utput Sets	Output Vectors		
5 F. 5 F. 10 X	All Output Vectors From Output	Set 11Case 1 Freq 180.4198	•
1Mode 1, 200.4665 Hz		Quick Filter 0None - Ignore	•
2Mode 2, 380.1996 Hz			
3Mode 3, 672.5712 Hz	2T1 Translation	70012SolidC1 Z Normal Stress	71011SolidC6 Y Normal Str
4Mode 4, 1015.848 Hz	3T2 Translation	70013SolidC1 XY Shear Stress	71012SolidC6 Z Normal Str
5Mode 5, 2022.406 Hz	4T3 Translation	70014SolidC1 YZ Shear Stress	71013SolidC6 XY Shear Str
6Mode 6, 2625.677 Hz	6R1 Rotation	70015SolidC1 ZX Shear Stress	71014SolidC6 YZ Shear Str
7Mode 7, 4156.374 Hz	7R2 Rotation	70210SolidC2 X Normal Stress	71015SolidC6 ZX Shear Str
8Mode 8, 4366.092 Hz	8R3 Rotation	70211SolidC2 Y Normal Stress	71210SolidC7 X Normal Str
9Mode 9, 5601.611 Hz	22T1 Acceleration	70212SolidC2 Z Normal Stress	71211SolidC7 Y Normal Str
10Mode 10, 6685.72 Hz	23T2 Acceleration	70213SolidC2 XY Shear Stress	71212SolidC7 Z Normal Str
11Case 1 Freq 180.4198	24T3 Acceleration	70214SolidC2 YZ Shear Stress	71213SolidC7 XY Shear Str
12Case 2 Freq 190.4432	26R1 Angular Acceleration	70215SolidC2 ZX Shear Stress	71214SolidC7 YZ Shear Str
/ 13Case 3 Freg 200.4665	27R2 Angular Acceleration	70410SolidC3 X Normal Stress	71215SolidC7 ZX Shear Str
14Case 4 Freq 210.4898	28R3 Angular Acceleration	70411SolidC3 Y Normal Stress	71410SolidC8 X Normal Str
15Case 5 Freg 220.5131	250Complex Mode Shape	70412SolidC3 Z Normal Stress	71411SolidC8 Y Normal Str
16Case 6 Freg 342.1797	6043Plate Top Fiber	70413SolidC3 XY Shear Stress	71412SolidC8 Z Normal Str
17Case 7 Freg 361.1896	6044Plate Bottom Fiber	70414SolidC3 YZ Shear Stress	71413SolidC8 XY Shear Str
18Case 8 Freg 380.1996	7020Plate Top X Normal Stress	70415SolidC3 ZX Shear Stress	71414SolidC8 YZ Shear Str
19Case 9 Freq 399.2096	7021Plate Top Y Normal Stress	70610SolidC4 X Normal Stress	71415SolidC8 ZX Shear Str
20Case 10 Freg 418.2196	7023Plate Top XY Shear Stress	70611SolidC4 Y Normal Stress	100007PltC1 Top Fiber
21Case 11 Freq 605.3141	7420Plate Bot X Normal Stress	70612SolidC4 Z Normal Stress	100008PltC1 Bottom Fiber
22Case 12 Freq 638.9426	7421Plate Bot Y Normal Stress	70613SolidC4 XY Shear Stress	100220PltC1 Top X Normal
23Case 13 Freg 672.5712	7423Plate Bot XY Shear Stress	70614SolidC4 YZ Shear Stress	100221PltC1 Top Y Normal
24Case 14 Freq 706.1998	60010Solid X Normal Stress	70615SolidC4 ZX Shear Stress	100223PltC1 Top XY Shear
25Case 15 Freq 739.8284	60011Solid Y Normal Stress	70810SolidC5 X Normal Stress	100620PltC1 Bot X Normal
26Case 16 Freg 914.2636	60012Solid Z Normal Stress	70811SolidC5 Y Normal Stress	100621PltC1 Bot Y Normal
27Case 17 Freg 965.0561	60013Solid XY Shear Stress	70812SolidC5 Z Normal Stress	100623PltC1 Bot XY Shear
28Case 18 Freq 1015.848	60014Solid YZ Shear Stress	70813SolidC5 XY Shear Stress	150007PltC2 Top Fiber
29Case 19 Freq 1066.641	60015Solid ZX Shear Stress	70814SolidC5 YZ Shear Stress	150008PltC2 Bottom Fiber
30Case 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
	 III 		4

Expand Complex Output	Data	×
Expand For		
	<u>F</u> irst Phase	0.
Single <u>P</u> hase	<u>L</u> ast Phase	360.
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.

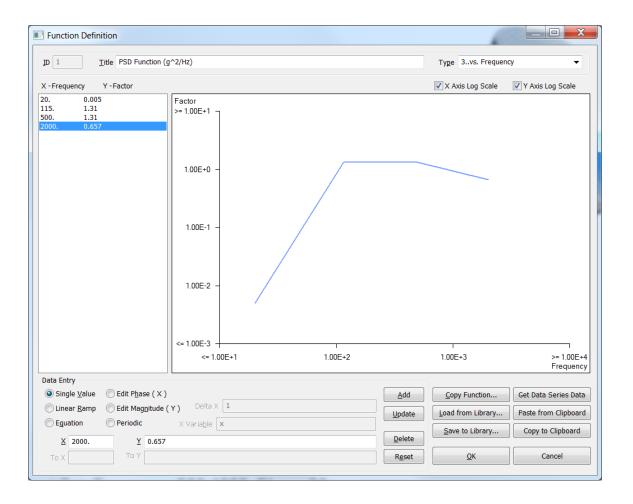
If you are interested in a tutorial just covering PSD analysis, then please take a read on our white paper:

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.



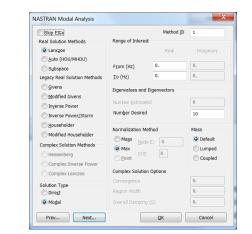




The PSD procedure is almost identical to the modal frequency analysis. There are some new screens but the only critical one is where you apply your PSD spectrum (lower-right-hand-corner). Otherwise, it is identical to that of the prior modal frequency analysis.

Analysis Set			×
<u>T</u> itle XX	X-0814-01	Y-Axis PSD Analysi	s
Analysis <u>P</u> ro	ogram	36NX Nastran	-
<u>A</u> nalysis Ty	pe	6Random Respo	nse 🔻
		Run Analysis Us	ing VisQ
Next		<u></u> K	Cancel

ptions for Dynamic Analysis Advance	d Options		
Equivalent Viscous Damping		Limit Response Base	d on Modes
Overall Structural Damping Coeff (G)	0.	Number of Modes	
Modal Damping 3PSD Modal	Frequer 🔻 🖍	Lowest Freq (Hz)	0.
As Structural (KDAMP)		Highest Freq (Hz)	0.
Equivalent Viscous Damping Conversio	n	Transient Time Step	Intervals
Convert using Solution Freq (WM	ODAL)	Number of	0
Rigid Body Zero Modes(FZERO)	1.E-4	Time per	0.
Freq for System Damping (W3 - Hz)	0.	Output Interval	0
Freq for Element Damping (W4 -	0.		
Frequency Response		Response/Shock Spe	ctrum
Frequencies 5Modal Frequency Ta	ible 🔻 🖍	Damping/Freq Corn	elation
Modal Freq		0None - fxy	

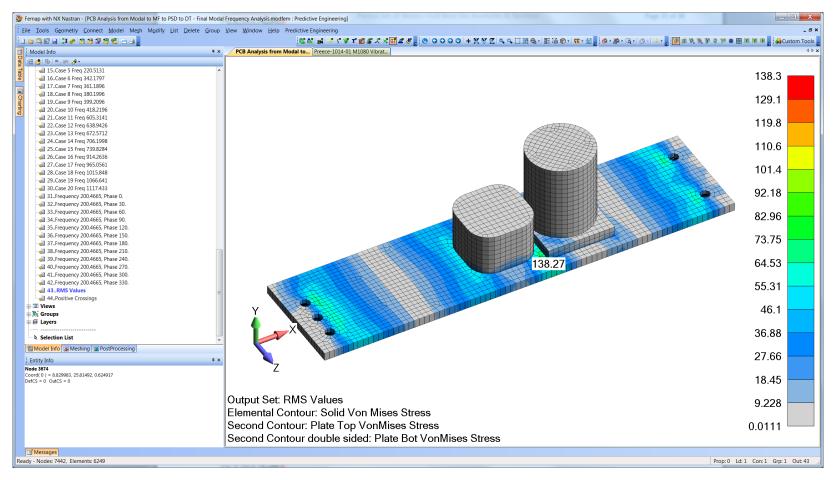


NASTRAN Power Spectral Density Factors					
Correlation Table					
Master=>Master 1.(1) :Int1=0	Excited Master				
	Load Set: 1Motor Frequency				
	Applied Master				
	Load Set: 1Motor Frequency				
Edit Correlation Table					
Factor PSD Fun	ction PSD Interpolation				
Real 1. × 1PSD Function	n (g^2/Hz) ▼ 0Log Log ▼				
Imaginary 1. × 0None	▼ 0Log Log ▼				
Apply					
Prev Next	QK 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.





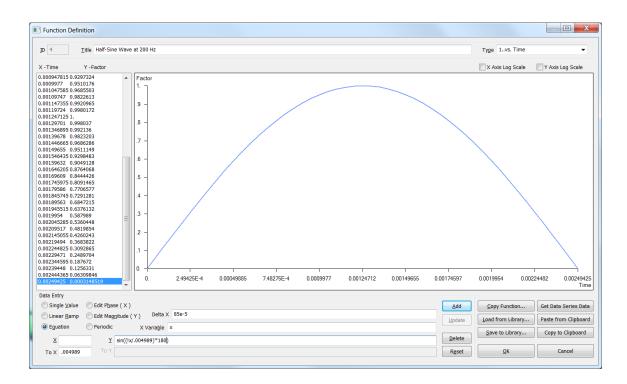


6. DIRECT TRANSIENT ANALYSIS

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:

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

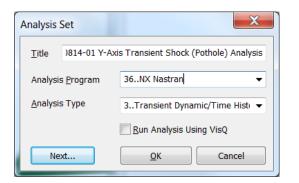
Our equation is developed in Femap using a sin((!x/0.004988)*) to180 create a 200.4 Hz half-sine wave:

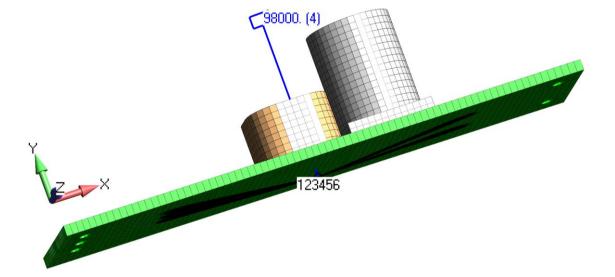






For our work, we are just going to use half the sin wave to give the system a shock pulse. The load for this analysis is 100 g (98,000 mm/s²) with our half-sine function at 200.4 Hz.









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

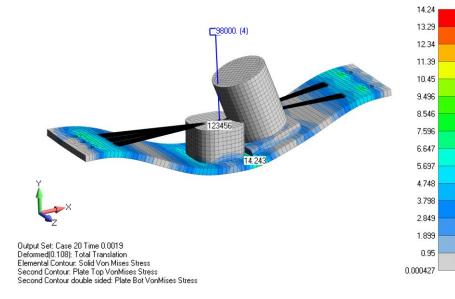
NASTRAN Modal Analysis			X
Skip EIGx		Method <u>I</u> D	1
Real Solution Methods	Range of Interest		
Lanczos		Real	Imaginary
<u>Auto (HOU/MHOU)</u>		-	
Subspace	F <u>r</u> om (Hz)	0.	0.
Legacy Real Solution Methods	<u>T</u> o (Hz)	0.	0.
© <u>G</u> ivens © Modified Givens	Eigenvalues and Eigenvalues	genvectors	
Inverse Power	Number <u>E</u> stimated		0
Inverse Power/Sturm	Num <u>b</u> er Desired		10
O Householder	Normalization Meth	od	Mass
Modified Householder			
Complex Solution Methods	Ma <u>s</u> s Max	0	Default Umped
Hessenberg	DOF	0	
Complex Inverse Power	O Point –		Coupled
Complex Lanczos	Complex Solution C	Options	
- ·	Convergence		0.
Solution Type	Region Width		0.
 Modal 	Overall Damping (G)	0.
Prev Next		<u>O</u> K	Cancel

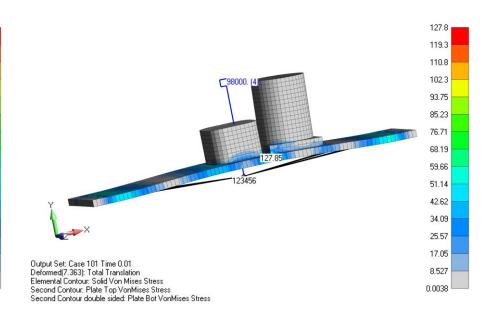
Dynamic Control Options		X		
Use Load Set Options				
Options for Dynamic Analysis Advanced Options				
Equivalent Viscous Damping				
Overall Structural Damping Coeff (G) 0.	Number of Modes	0		
Modal Damping 3PSD Modal Frequer 💌 🔩	Lowest Freq (Hz)	0.		
As Structural (KDAMP)	Highest Freq (Hz)	0.		
Equivalent Viscous Damping Conversion	Transient Time Step Intervals			
Convert using Solution Freq (WMODAL)	Number of	1000		
Rigid Body Zero Modes(FZERO) 1.E-4	Time per	1.E-5		
Freq for System Damping (W3 - Hz) 0.	Output Interval	10		
Freq for Element Damping (W4 - 0.				
Frequency Response	Response/Shock Spe	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 Title Contains time OK Cancel	
Process Output Data	
What to Process	
@ Complete Output Sets	
Processing Operations	
Processing Operations Copy Merge Linear Combination RSS Combination Envelope Error Estimate Convert	
Type Envelope Approach Create Envelopes	
Max Value O Envelope All Selected Vectors Within Output Sets	
Min Value Envelope All Locations For Each Vector	
Max Absolute Value Envelope Each Vector Independently Store Set/Location Info	
Envelope Every Vector across Multiple Output Sets	
Select Output To Process Store Output in Set 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 Max 47Case 3 Time 0.0002	Reset
Max 48Case 4 Time 0.0003 Max 49Case 5 Time 0.0004	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 56. Case 12 Time 0.0011	
Max 57. Case 13 Time 0.0012 Max 58. Case 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	<u>о</u> к
Max 68Case 24 Time 0.0023 Max 69Case 25 Time 0.0024	
Max 70Case 26 Time 0.0025	- Cancel





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

Image: Comparison of the comparison	Tools Geometry Connect Model Mesh Modify List Delete Group	View Window Help. Predictive Engineering	
Side Control Control Control 127.40 127.4			A III III III A Custon
1) 1) 2.0.02 1) 2.0.02			The last as a group custom
113 Code 91 Time 0014 127.8 12 Code 91 Time 0015 110.000 13 Code 91 Time 0015 110.000 <tr< th=""><th></th><th>Tech Analysis from modal to Prece-1014-01 m1000 vitulat</th><th></th></tr<>		Tech Analysis from modal to Prece-1014-01 m1000 vitulat	
11 Dick of Pitme 0075 127, 48 12 Dick of Pitme 0075 119, 3 12 Dick of Pitme 0075 119, 3 12 Dick of Pitme 0075 100, 3 12 Dick of Pitme 0075 100, 3 12 Dick of Pitme 0075 100, 3 13 Dick of Pitme 0075 100, 3 <			
12 12 cot of Time 0007 111 cot of Time 0007 112 cot of Time 0007 <			127.8 🗖
12 J.C. et al Time 007 1119.3 12 J.C. et al Time 007 110.4 12 J.C. et al Time 007 101.4 12 J.C. et al Time 007 101.4 12 J.C. et al Time 007 101.4 13 J.C. et al Time 007 101.4 14 J.C. et al Time 007 101.4 <			
12 J.C. des 11 me 6007 110.0 12 J.C. des 11 me 6008 100.0 13 J.C. des 11 me 6008 100.0 14 J.C. des 11 me 6008 100.0 15 J.C. des 11 me 6008 100.0 14 J.C. des 11 me 6008 100.0 <t< td=""><td>🗃 122Case 78 Time 0.0077</td><td></td><td>440.0</td></t<>	🗃 122Case 78 Time 0.0077		440.0
12 32 Code 21 Time 0008 1101.0008 12 32 Code 21 Time 0008 1002.3 13 32 Code 31 Time 0008 1002.3 13 32 Code 31 Time 0008 93.75 13 32 Code 31 Time 0008 85.203 13 32 Code 31 Time 0008 93.75 13 32 Code 31 Time 0008 85.203 13 32 Code 31 Time 0009 95.601 13 42 Code 31 Time 0009 95.601 13 42 Code 31 Time 0009 95.601 14 42 Code 31	-		119.3
12 D C cet 2 Time 0002 110.8 12 D C cet 2 Time 0002 100.2 13 D C cet 2 Time 0002 93.75 13 D C cet 2 Time 0002 93.75 13 D C cet 2 Time 0002 85.203 13 D C cet 2 Time 0002 93.75 13 D C cet 2 Time 0002 85.203 14 D C cet 2 Time 0002 85.203 15 D C cet 2 Time 0002 95.255.			
137 Loc de 31 me 00031 102.4 de 31 me 00031 102.3 de 11 me 00031 102.4 de 31 me 00031			110.8
12 JAC 48 5 Tree 00081 102.3 13 JAC 48 5 Tree 00081 93.75 13 JAC 48 5 Tree 00081 85.22 13 JAC 48 5 Tree 00081 95.66 14 JAC 48 5 Tree 00081 95.66 <t< td=""><td></td><td></td><td>110.0</td></t<>			110.0
13 Laces 97 Ime 0.0056 13 Laces 97 Ime 0.0056 93.75 13 Laces 97 Ime 0.0056 85.203 13 Laces 97 Ime 0.0056 12.2.4.85 Time 0.0056 13 Laces 97 Ime 0.0056 12.2.4.85 Time 0.0056 14 Laces 97 Time 0.0056 12.2.4.85 Time 0.0066 Time 0.0066 Time 0.0066 Time 0.0066 Time 0.0066 Ti			400.0
131.2.c.287 Time 00087 93.75 131.2.c.288 Time 00087 85.23 132.c.288 Time 00087 95.66 132.c.288 Time 00087 95.66 <t< td=""><td>al 129Case 85 Time 0.0084</td><td></td><td>102.3 -</td></t<>	al 129Case 85 Time 0.0084		102.3 -
132.4ce 85 Time 0.088 93.75 133.4ce 85 Time 0.088 85.23 135.4ce 85 Time 0.088 85.23 135.4ce 85 Time 0.089 127.85 134.4ce 80 Time 0.089 59.66 135.4ce 80 Time 0.089 59.66 135.4ce 80 Time 0.089 59.66 136.4ce 8	-		
313.4.ces 91 me 0099 135.4.ces 91 me 0099 85.2.9 135.4.ces 91 me 0099 127.8.9 76.71 135.4.ces 91 me 0093 127.8.9 76.71 135.4.ces 91 me 0093 68.19 135.4.ces 91 me 0003 59.66 145.4.ces 91 me 0003 59.66 146.4.fes 91 me 0003 59.66 157.7 50.56,57,58	-		93 75 🗕
13 Lace 29 Time 0.009 135.cce 20 Time 0.009 135.cce 20 Time 0.009 127.85 76.71 13 Lace 28 Time 0.009 127.85 76.71 68.19 14 Lace 28 Time 0.009 59.66 59.66 15 Lace 10 Time 0.009 59.66 59.66 16 Lace 28 Time 0.009 59.66 59.67 16 Lace 28 Time 0.009 59.66 59.65 16 Lace 28 Time 0.009 59.66 59.66 16 Lace 28 Time 0.010 Contour: 10 Lace 28 Time 0.010 Con			00.70
131 Acces 27 Ime 0.0091 1127.85 76.71 131 Acces 26 Time 0.0092 1127.85 68.19 131 Acces 26 Time 0.0093 69.19 69.19 131 Acces 26 Time 0.0093 69.19 59.666 131 Acces 26 Time 0.0093 59.666 59.666 131 Acces 26 Time 0.0093 59.666 59.666 131 Acces 26 Time 0.0093 59.666 59.66 131 Acces 26 Time 0.0093 59.666 59.66 131 Acces 26 Time 0.0093 59.666 59.66 141 Acces 27 Time 0.019 59.66 59.66 151 Acces 26 Time 0.019 59.66 59.67 152 Acces 26 Time 0.019 59.66 59.67 154 Acces 27 Time 0.019 59.66 59.67 154 Acces 27 Time 0.019 59.66 59.57 154 Acces 27 Time 0.019 59.66 59.56 154 Acces 27 Time 0.010 59.66 59.56 154 Acces 27 Time 0.010 59.56 57.58			05.00
13/2.6ce 93 Time 0.0092 127.85 76.71 13/2.6ce 93 Time 0.0093 127.85 76.71 13/2.6ce 93 Time 0.0093 68.19 13/2.6ce 93 Time 0.0093 59.66 13/2.6ce 93 Time 0.0093 59.57 13/2.6ce 93 Time 0.00038 59.56			85.23
1 Jack 2004 95 Time 0.0005 1 Jack 2005 97 Time 0.0005 1 Jack 2005 97 Time 0.0007 1	-		
13 Jack 26 95 Time 0.0094 68.19 14 Jack 26 95 Time 0.0095 59.66 14 Jack 26 95 Time 0.0096 59.66 14 Jack 26 95 Time 0.0096 59.66 14 Jack 26 95 Time 0.0096 51.14 14 Jack 26 95 Time 0.0097 51.14 14 Jack 26 95 Time 0.0096 51.14 14 Jack 26 95 Time 0.0097 51.14 14 Jack 26 95 Time 0.0098 51.14 14 Jack 26 95 Time 0.0097 51.14 14 Jack 26 95 Time 0.0098 52.555.57.38.59 5etclin list 34.09 10 - a 0.00991, 25.1542, 7.44004 25.57 587 25.57 10 - a 0.00991, 25.1542, 7.44004 25.57 10 - a 0.00991, 25.1542, 7.44004 25.57 10 - a 0.00991, 25.1542, 7.44004 50.151, 52, 53, 54, 55, 56, 57, 58, 59, 60, 61, 62, 63, 64, 65 Deformed(7, 363): Total Translation 8.527 Deformed(7, 363): Total Translation 8.527 Second Contour: Plate Top VonMises Stress 0.0038 Second Contour double sided: Plate Bot VonMises Stress 0.0038	-	127.85	76 71 🗕
1 = 10.42.88 9 Time 00095 68.19 1 = 14.12.88 9 Time 00097 59.66 1 = 14.2.88 9 Time 00098 59.165 1 = 14.2.88 9 Time 00098 51.114 1 = 14.2.88 9 Time 00098 52.57 5 = 116 0 al Mething ig PostPocessing 25.57 1 = 0 oct 5 = 0 0 utput Set: Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65 17.05 0 oct 5 = 0 0 utput Set: Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65 17.05 0 oct 5 = 0 0 utput Set: Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65 17.05 0 oct 5 = 0 0 utput Set: Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65 17.05 0 oct 5 = 0 0 utput Set: Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65 17.05			70.71
14/L2as 97 Time 00096 68.19 14/L2as 97 Time 00097 59.66 14/L2as 90 Time 00098 59.66 14/L2as 90 Time 00097 51.14 14/L2as 90 Time 00097 52.55 14/L2as 90 Time 00097 51.14 14/L2as 90 Time 00097 52.57 56ection List 34.09 10/L adoes 10 Time 001 52.57 587 25.57 587 25.57 587 25.57 597 25.57 597 25.57 597 25.57 597 25.57 597 25.57 597 25.57 597 25.57 597 25.57 597 25.57 597 25.57 597 25.5			00.40
1 14. Gase 00 Time 0.009 1 45. Gase 101 Time 0.01 1 45. Gase 101 Tim			68.19
144.Case 100 Time 0.0099 39.66 145.Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65) 51.14 145.Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65) 51.14 Versus 42.62 Selection List 34.09 Voidel Info (a) Mething (a) Costrocesing 25.57 160 - More Selection List 34.09 Voidel Info (a) Mething (a) Costrocesing 25.57 160 - More Selection List 34.09 Distribution (a) Costrocesing 25.57 160 - More Selection List 34.09 Distribution (a) Costrocesing 25.57 Selection List 35.27 Distribution (C) Costroce (C) Co			
 I Hacker IDI Time 0039 I Hacker IDI Time 0019 I Hacker IDI Time	-		59 66
a) 147.Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65) 147.Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65) 142.62 142.62 142.62 142.62 142.62 142.62 142.62 142.62 142.62 150 160 160 160 160 160 160 160 16			00.00
sel tar. Levelope Set Info (Max 45,46,47,48,495,051,52,533,5455,556,57,58,59) i verses i selection List into a x selection List into a x selection List into a x second Contour: Solid Von Mises Stress Second Contour: Plate Top VonMises Stress Second Contour double sided: Plate Bot VonMises Stress			F4 44
i Groups 42.62 selection List 34.09 indeel Info (a Meshing) @ PostProcessing 25.57 isyn 25.57 0 Output Set: Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65) 17.05 Deformed(7.363): Total Translation 8.527 Elemental Contour: Solid Von Mises Stress 0.0038 Second Contour double sided: Plate Bot VonMises Stress 0.0038			51.14
Layers 42.62 Selection List 34.09 iv Info ax 25.57 10 = -4.069901, 258,1492, 7.449904 = 0 Outcs = 0 Output Set: Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65) Deformed(7.363): Total Translation Elemental Contour: Solid Von Mises Stress Second Contour: Plate Top VonMises Stress Second Contour double sided: Plate Bot VonMises Stress 0.0038			
Selection List 34.09 Import Import inty Info Import 3897 25.57 (0) = -4.06991, 25.81492, 7.449904 17.05 (inty of the construction		Y	42.62
Image: Control of the second Contour: Plate Top VonMises Stress 34.09 Addel Info @ Meshing @ PostProcessing 25.57 ity Info * x 25.57 25.57 0 utput Set: Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65) 17.05 Deformed(7.363): Total Translation 8.527 Elemental Contour: Solid Von Mises Stress 0.0038 Second Contour double sided: Plate Bot VonMises Stress 0.0038	Layers		
Image: Control of the desting in PostProcessing PostProcessing idy Info * * 387 25.57 10) = -4.000001, 25.81492, 7.449904 17.05 10) = -4.000001, 25.81492, 7.449904 0utput Set: Envelope (Max 45, 46, 47, 48, 49, 50, 51, 52, 53, 54, 55, 56, 57, 58, 59, 60, 61, 62, 63, 64, 65 17.05 Deformed(7.363): Total Translation 8.527 Elemental Contour: Solid Von Mises Stress 0.0038 Second Contour: Plate Top VonMises Stress 0.0038	Selection List		24.00
inty Info ** Z 25.57 ssy V(1) = -405991, 25,81492, 7,449904 17.05 Output Set: Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65 17.05 Deformed(7.363): Total Translation 8.527 Elemental Contour: Solid Von Mises Stress 0.0038 Second Contour: Plate Top VonMises Stress 0.0038			34.09
inty Info ** Z 25.57 ssy V(1) = -405991, 25,81492, 7,449904 17.05 Output Set: Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65 17.05 Deformed(7.363): Total Translation 8.527 Elemental Contour: Solid Von Mises Stress 0.0038 Second Contour: Plate Top VonMises Stress 0.0038	lodel Info 👩 Meshing 🔳 PostProcessing		
387 17.05 (0) = -4.069991, 25.81492, 7.449904 0utput Set: Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65 17.05 Deformed(7.363): Total Translation 8.527 Elemental Contour: Solid Von Mises Stress 0.0038 Second Contour: Plate Top VonMises Stress 0.0038			25.57 📥
••• ourcs •• Output Set: Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65 17.05 Deformed(7.363): Total Translation 8.527 Elemental Contour: Solid Von Mises Stress 0.0038 Second Contour: Plate Top VonMises Stress 0.0038	3597		
Output Set: Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65 Deformed(7.363): Total Translation Elemental Contour: Solid Von Mises Stress Second Contour: Plate Top VonMises Stress Second Contour double sided: Plate Bot VonMises Stress	i(0) = -4.069991, 25.81492, 7.449904		17.05
Deformed(7.363): Total Translation 8.527 Elemental Contour: Solid Von Mises Stress 0.0038 Second Contour: Plate Top VonMises Stress 0.0038 Second Contour double sided: Plate Bot VonMises Stress 0.0038	= 0 Out(S = 0	Output Set: Envelope (Max 45,46,47,48,49,50,51,52,53,54,55,56,57,58,59,60,61,62,63,64,65,	17.05
Elemental Contour: Solid Von Mises Stress 0.0027 Second Contour: Plate Top VonMises Stress 0.0038 Second Contour double sided: Plate Bot VonMises Stress 0.0038			
Second Contour: Plate Top VonMises Stress 0.0038 Second Contour double sided: Plate Bot VonMises Stress 0.0038			8.527 🛏
Second Contour double sided: Plate Bot VonMises Stress		Elemental Contour: Solid Von Mises Stress	
Second Contour double sided: Plate Bot VonMises Stress		Second Contour: Plate Ton VonMises Stress	0.0038
			0.0030
tessages		Second Contour double sided: Plate Bot VonMises Stress	
	lessages		



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

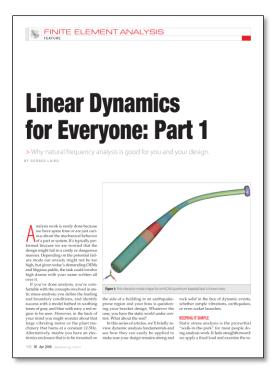
8.1 CHECK F06 FOR MASS SUMMATION AND KNOW WHAT YOU KNOW

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 Che	ck		Ground Check	
DOF SET		■ <u>F</u>	DOF SET	<u>E</u>
	<u>N</u>	<u>A</u>	<u>N</u>	<u>A</u>
□ N+A <u>U</u> TOSPC □ <u>V</u>		N+AUTOSPC		
				Print Forces Above
\Box <u>C</u> GI (Center of Gravity)		DATAREC	10 %	
Re <u>f</u> Node	0		Ref Nod <u>e</u>	0
<u>U</u> nits	0V	/eight 👻	Max Strain Energy	0.
Pre	ev 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 Femap 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

