

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)



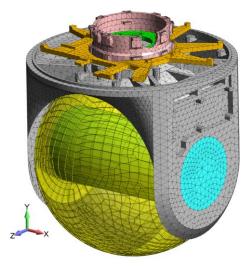
1. INTRODUCTION TO LINEAR DYNAMICS (NX NASTRAN)

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



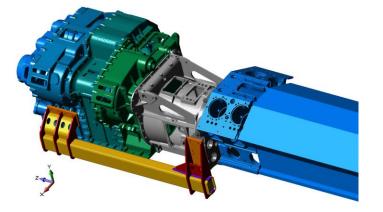
Linear FEA Model (courtesy Predictive Engineering)



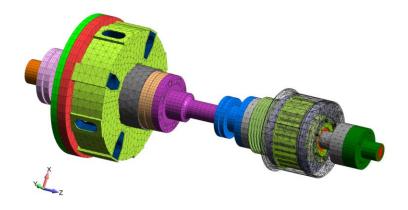


1.1 SOME EXAMPLES OF LINEAR VIBRATION ANALYSIS (PREDICTIVE)

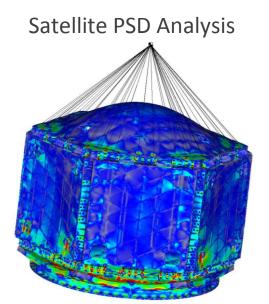
3,000 HP Transmission

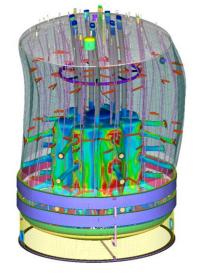


Drive Train Coupling



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:

$$Ku = F$$



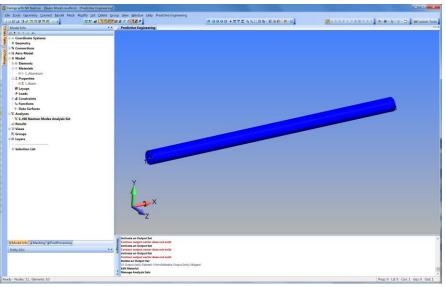
2. FOUNDATION OF FREQUENCY ANALYSIS

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)

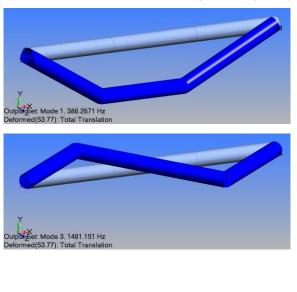
| ionoral Function Deferrer | one Manlinear Dhy/Dand Feilum | Croop Electrical/O | Layer 1 Type | 10 |
|-----------------------------|--------------------------------|--|--------------|------------|
| Stiffness | ces Nonlinear Ply/Bond Failure | Limit Stress | pucal Phase | Data Table |
| Youngs Modulus, <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

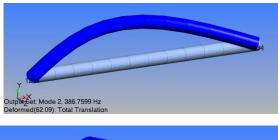
- o Elastic and mass properties are in consistent units
 - [°] The weight of your structure can be checked by summing the mass of the model and multiplying it by gravity (for US units of lbf, inch and seconds, it would be 386 in/s²)
- FEA model with a sufficient mesh density to capture the frequencies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

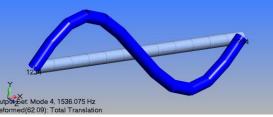


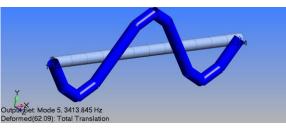
Doesn't Exist

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

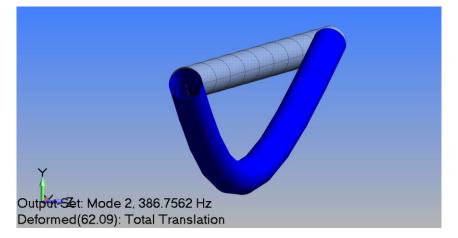


2.1.3 INTERPRETING RESULTS BASED ON ORTHOGONALITY AND MASS PARTICIPATION

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

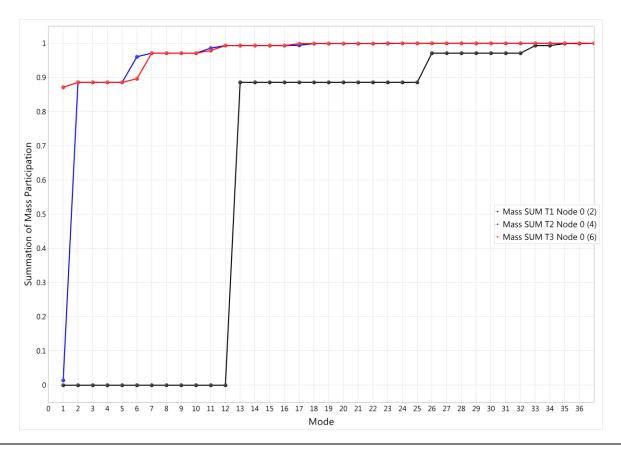
Second Frequency 386.8 Hz





FINITE ELEMENT ANALYSIS Predictive Engineering

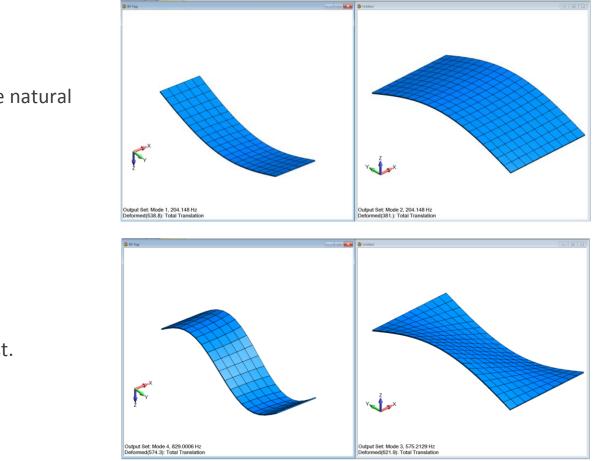
Mass participation tells you which modes have the "umph" and how many frequencies you need (modes) to accurately capture the dynamic response of the structure. On this later subject, a modal frequency analysis (e.g., PSD) formulates its response based on the number of modes chosen for the analysis. To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

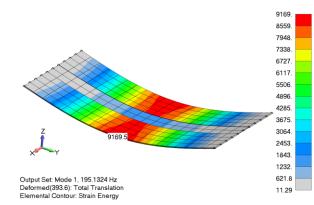
If one wants to move your natural frequencies up or down, sometimes intuition is good enough but it never hurts to have a quantitative tool. When a structure flexes or vibrates, there will be regions within the structure that are deforming more and other regions less. Since a natural frequency analysis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

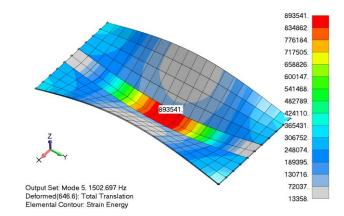
The Default Setup

| (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

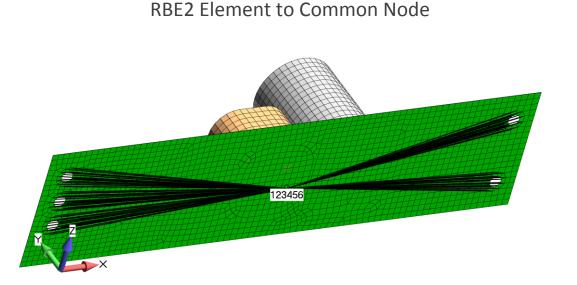
Units: N, mm, Tonne, s





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 |

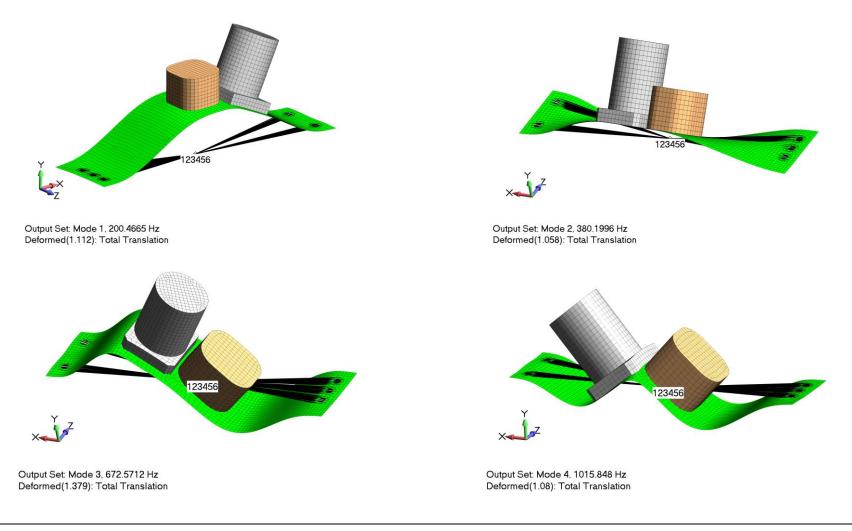
A normal modes / Eigenvalue analysis is the starting point for all linear dynamics work. It is simple to setup but difficult to interpret the results.

| 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.



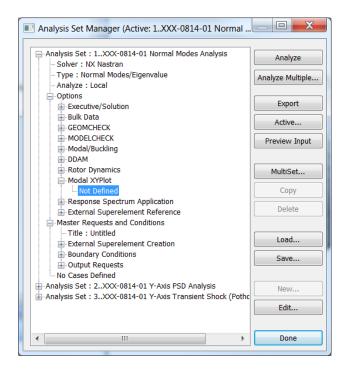


Applied CAx / Predictive Engineering White Paper – Please share with your Friends

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

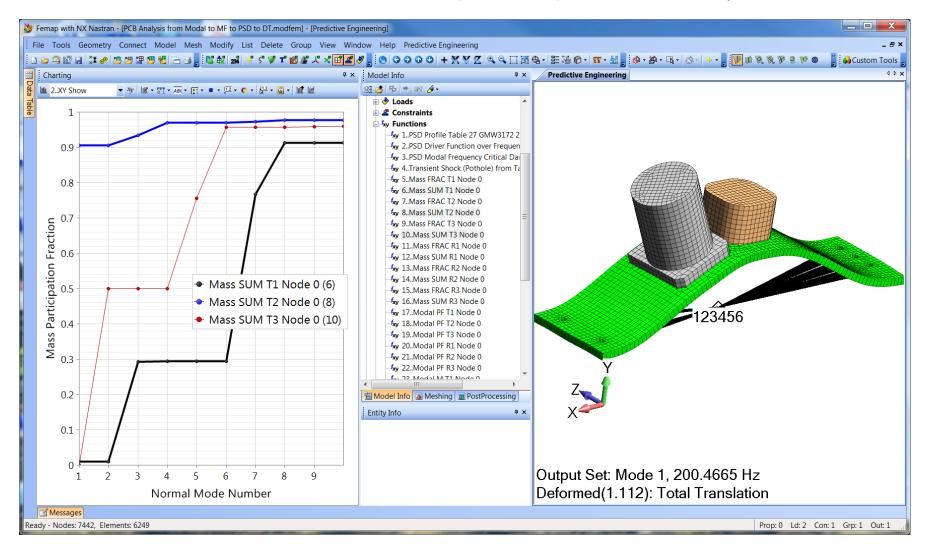


With the Not Defined item highlighted, hit the Edit button and the NASTRAN XY Output for Modal Analysis dialog box will appear. This box requests the mass particpation factors.

| 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:

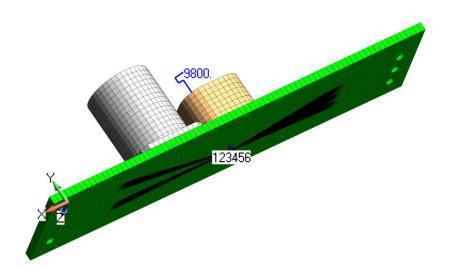
NX Nastran Dynamic Analysis.pdf





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).

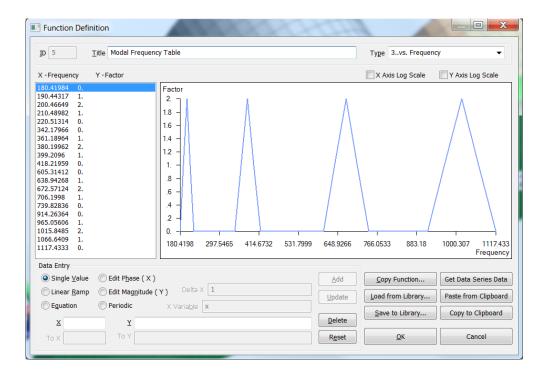
| NASTRAN Modal Analysis | | | X |
|------------------------------|-------------------------|-------------------|-----------|
| Skip EIGx | | Method <u>I</u> D | 1 |
| Real Solution Methods | Range of Interest | | |
| Lanczos | | Real | Imaginary |
| <u>Auto (HOU/MHOU)</u> | | - | |
| Subspace | F <u>r</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 | | | | | | |





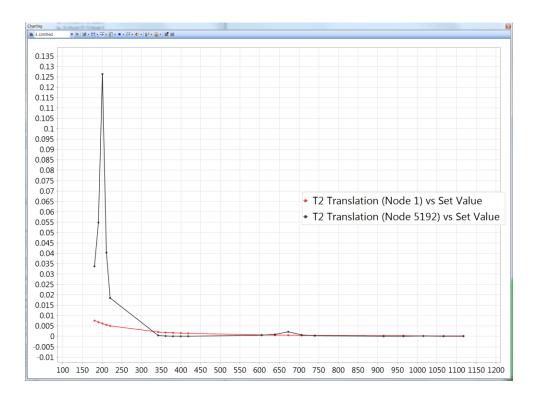
Then, one applies the boundary conditions and then lastly, one sets the output requests.

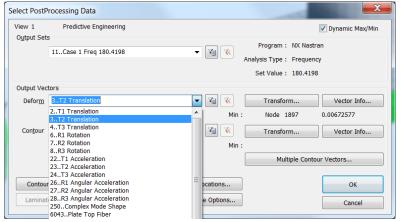
| Boundary Conditions | X |
|------------------------------|-------------------------------|
| Primary Sets | |
| <u>C</u> onstraints | 1Universal All-Purpose Con: 🔻 |
| <u>L</u> oads | 1Motor Frequency |
| Temperatures | 0From Load Set 👻 |
| Initial Conditions | 0None |
| Constraint <u>E</u> quations | 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.







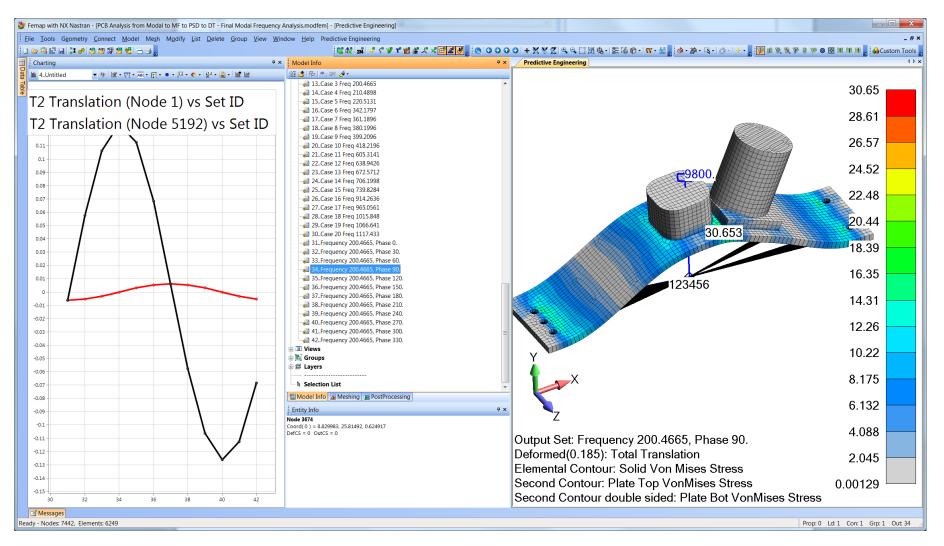
To obtain the time varying response from a Modal Frequency, one goes to Model / Output / Expand Complex and pick your solution of interest. For this structure it is the maximum response at 200 Hz and then we'll request that it is expanded into 12 solutions.

| 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.

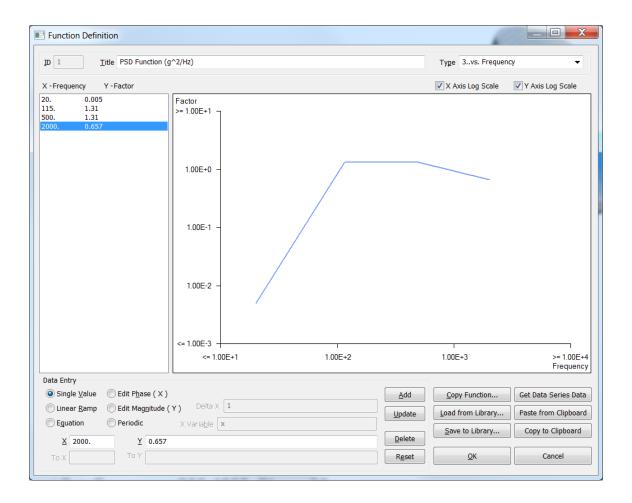
If you are interested in a tutorial just covering PSD analysis, then please take a read on our white paper:

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.

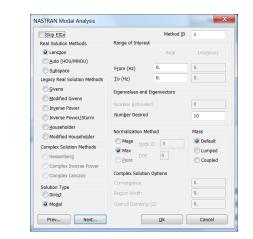




The PSD procedure is almost identical to the modal frequency analysis. There are some new screens but the only critical one is where you apply your PSD spectrum (lower-right-hand-corner). Otherwise, it is identical to that of the prior modal frequency analysis.

| 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 |

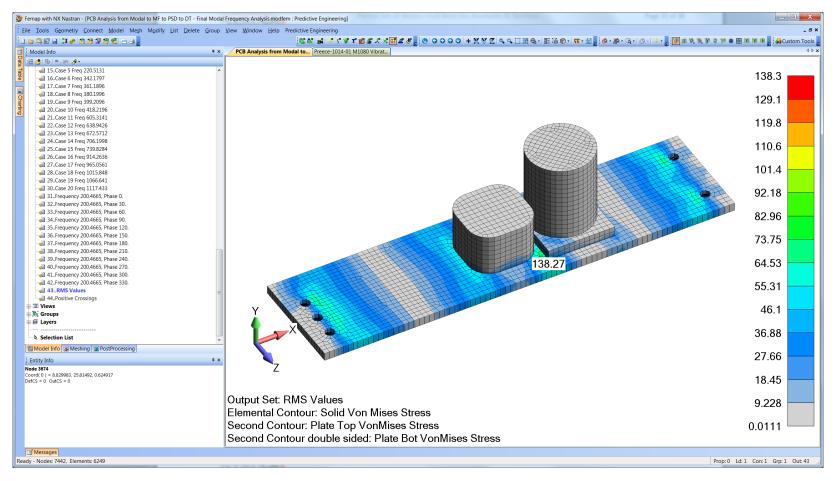
| otions for Dynamic Analysis Advar | nced Options | | |
|------------------------------------|-----------------|---------------------|-------------------|
| quivalent Viscous Damping | | Limit Response Base | d on Modes |
| Overall Structural Damping Coeff (| G) 0. | Number of Modes | |
| Modal Damping 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.





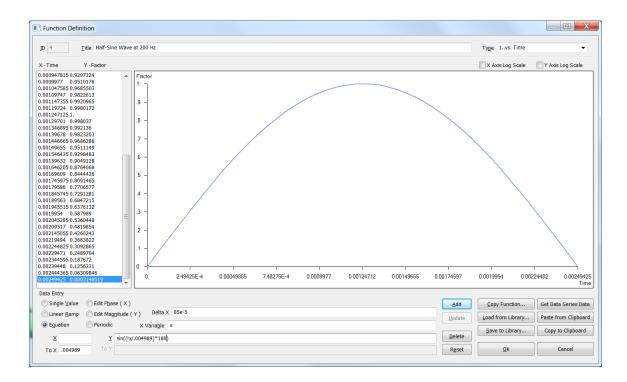


6. DIRECT TRANSIENT ANALYSIS

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:

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

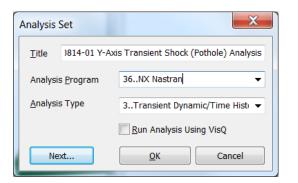
Our equation is developed in Femap using a sin((!x/0.004988)*) to180 create a 200.4 Hz half-sine wave:

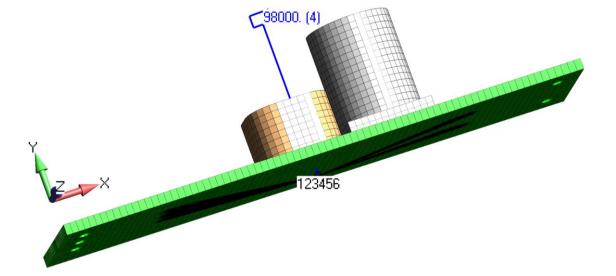






For our work, we are just going to use half the sin wave to give the system a shock pulse. The load for this analysis is 100 g (98,000 mm/s²) with our half-sine function at 200.4 Hz.









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

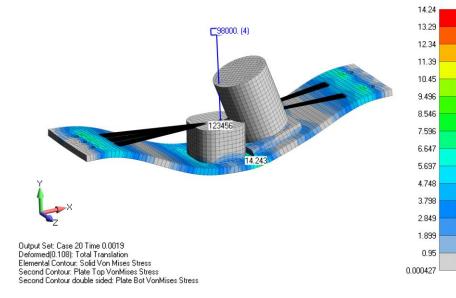
| NASTRAN Modal Analysis | | | × |
|------------------------------|-------------------------|-------------------|-----------|
| Skip EIGx | | Method <u>I</u> D | 1 |
| Real Solution Methods | Range of Interest | | |
| Lanczos | | Real | Imaginary |
| <u>Auto (HOU/MHOU)</u> | | | |
| Subspace | F <u>r</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 |

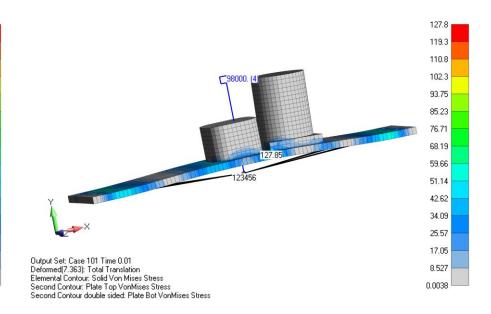
| Dynamic Control Options | | X | |
|---|-------------------------------|-------------------|--|
| Use Load Set Options | | | |
| Options for Dynamic Analysis Advanced Options | | | |
| Equivalent Viscous Damping | Limit Response Base | d on Modes | |
| Overall Structural Damping Coeff (G) 0. | Number 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.









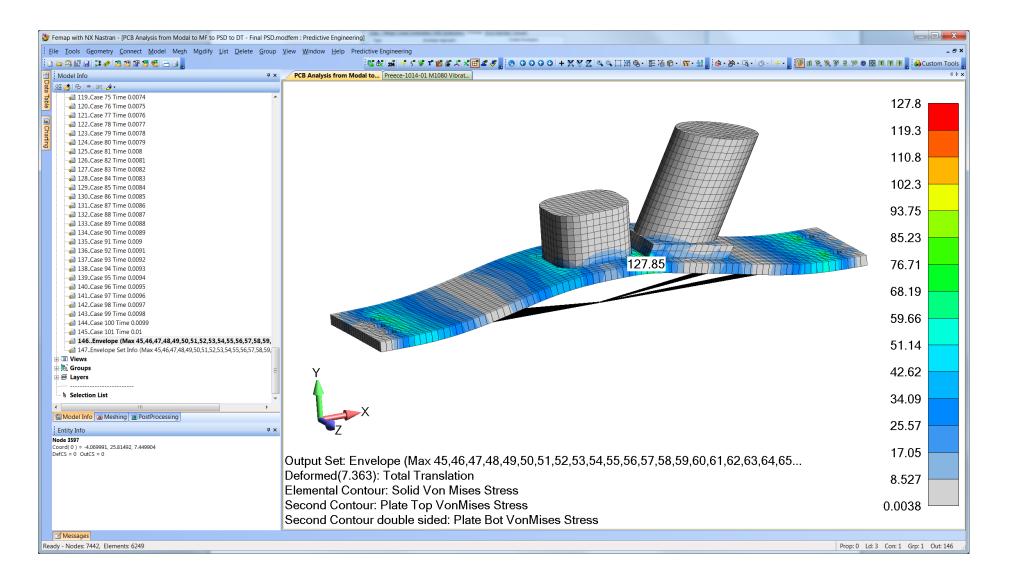
A much simpler way to process transient results is to use the Femap envelope function and then select all the output sets with "time" in the title:

| 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

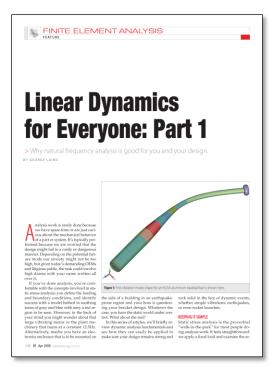
8.1 CHECK F06 FOR MASS SUMMATION AND KNOW WHAT YOU KNOW

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

