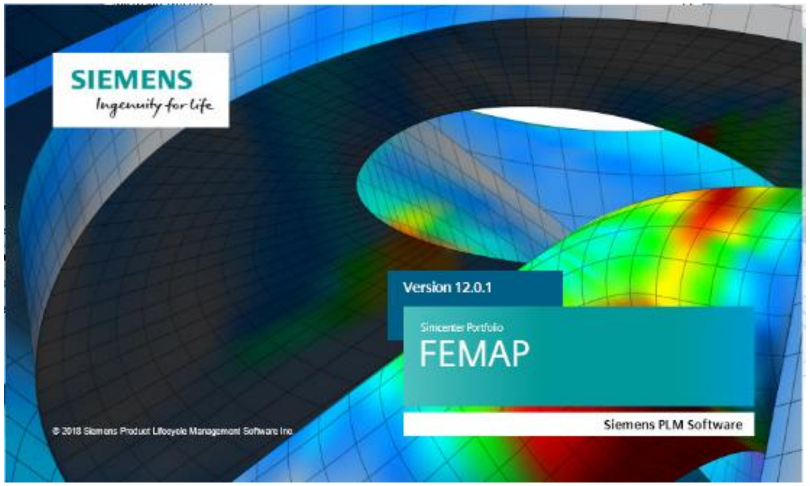
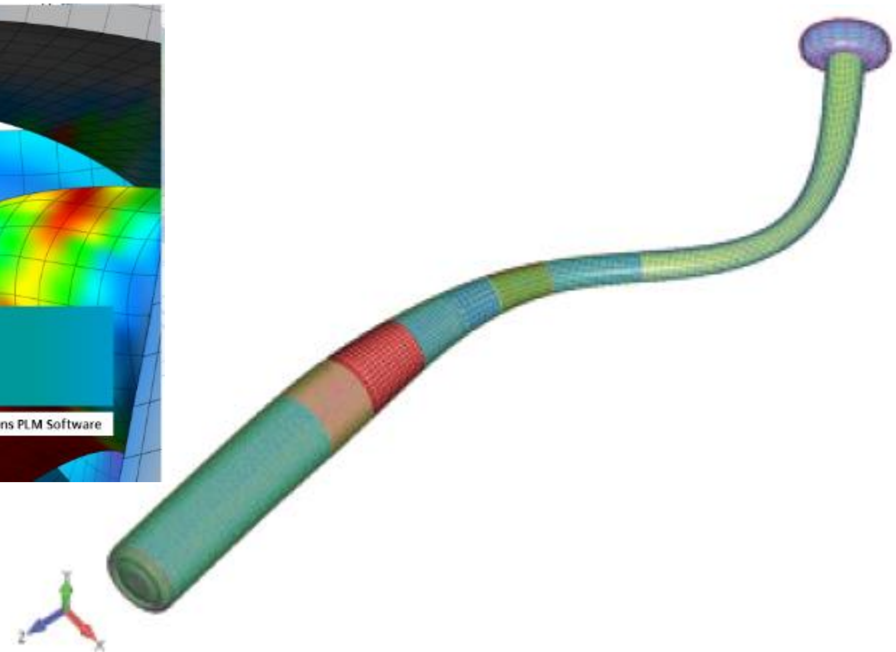




Dynamic Analysis User Guide

A user guide for FEMAP and NX Nastran Users

FINITE ELEMENT ANALYSIS
Predictive Engineering

LS-DYNA Sales, Support & Consulting

www.PredictiveEngineering.com

AppliedCAx

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

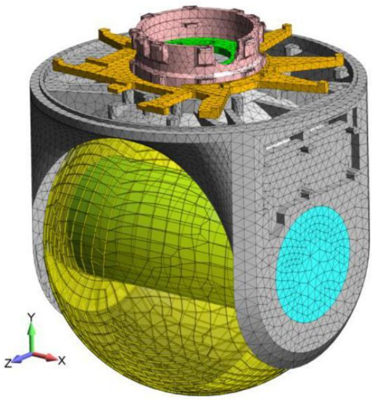
TABLE OF CONTENTS

1. INTRODUCTION TO LINEAR DYNAMICS (NX NASTRAN)

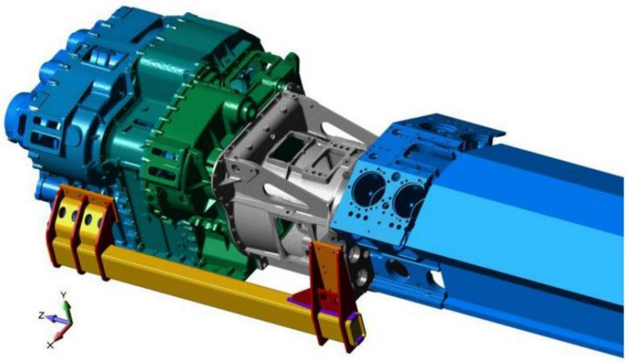
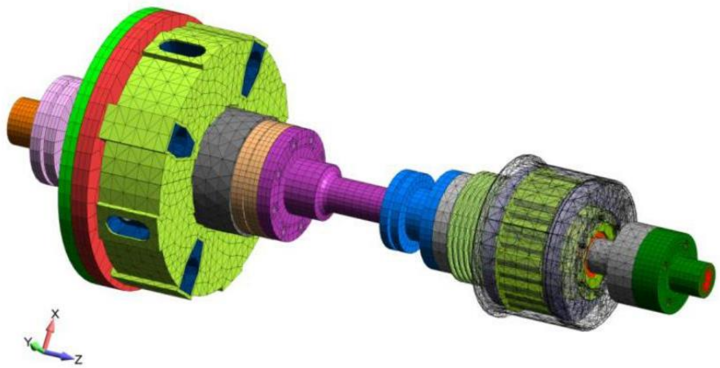
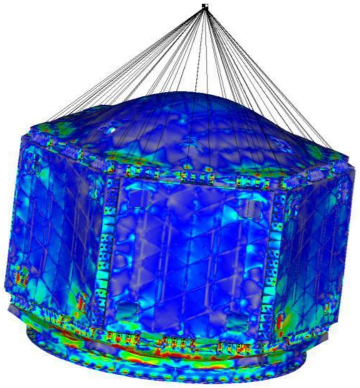
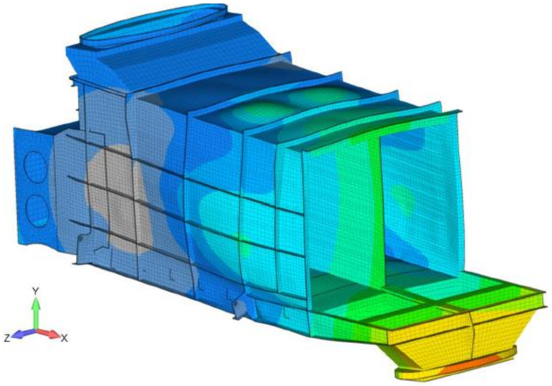
Vibration analysis is a huge topic and is easily the second most common type of FEA analysis after the basic static stress analysis. Within the field of vibration analysis, the most common type of analysis is that based on the linear behavior of the structure or system during its operation. That is, 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
	
Satellite PSD Analysis	**Frequency Response of Shaker Screen**
	

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: $$u = u_o \sin \omega t$$

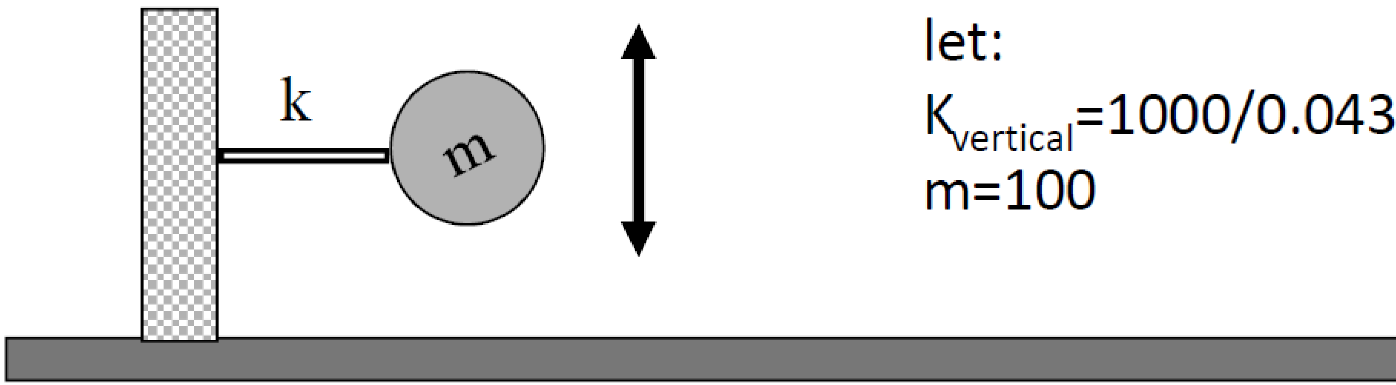
Then: $$\left[k - \omega^2 m\right]\{u_o\} = 0$$

For non-trivial solutions (i.e., $u_o \neq 0$):

$$\left[k - \omega^2 m\right] = 0$$

Giving us the well-known frequency relationship: $$\omega = \sqrt{\frac{k}{m}}$$

Normal Modes / Eigenvalue problem: undamped free vibration



let:
$K_{vertical}$=1000/0.0435
m=100

$$\omega = \sqrt{\frac{k}{m}} = \sqrt{\frac{23{,}000}{100}} = 15.16\,{}^{rad}\!/_{sec}$$

NX Nastran reports frequencies in cycles per second. Hence, 15.16 radians/sec is equal to 2.41 cycles/sec.

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)

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".

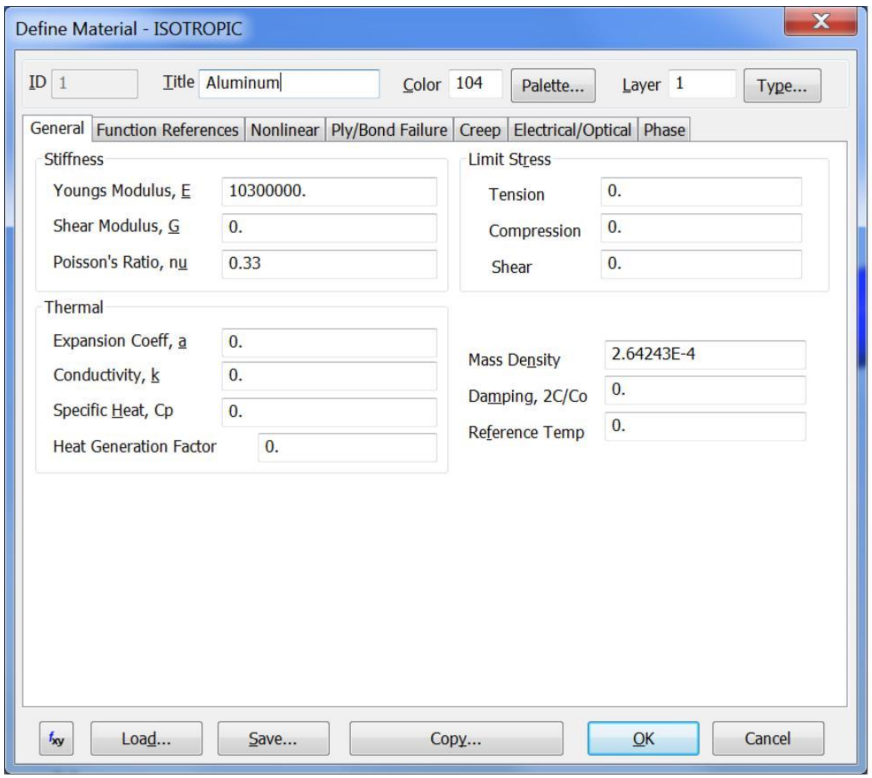
Before we leave this subject, a static stress analysis is nothing more than the above equation with acceleration and velocity at zero and time = zero:

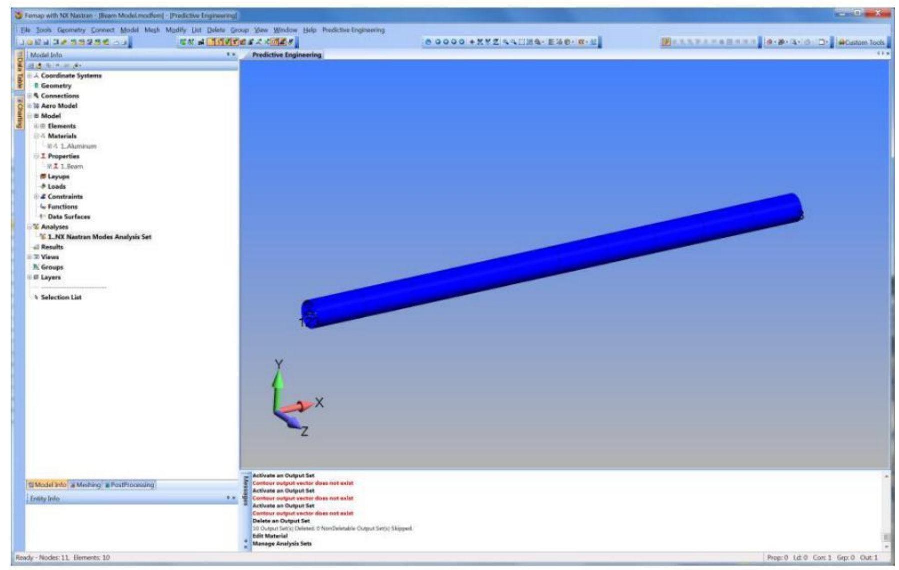
2. FOUNDATIONS OF FREQUENCY ANALYSIS

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)

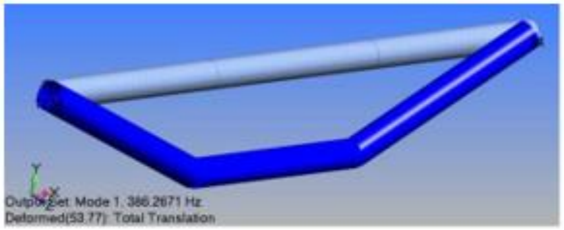


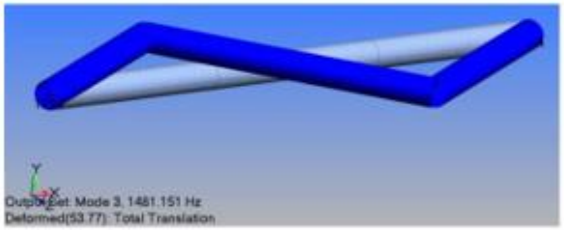


2.1.1 HERE'S YOUR MODAL ANALYSIS CHECKLIST

- 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)
- Constraint set that reflects reality as close as one can with a numerical simulation

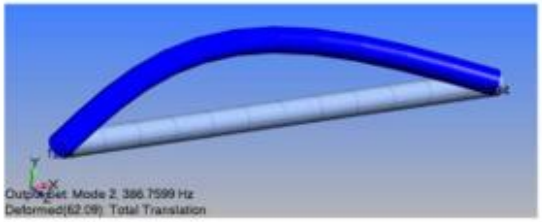
Three Element Mesh (Coarse)

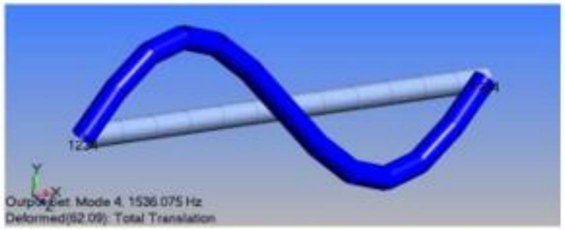


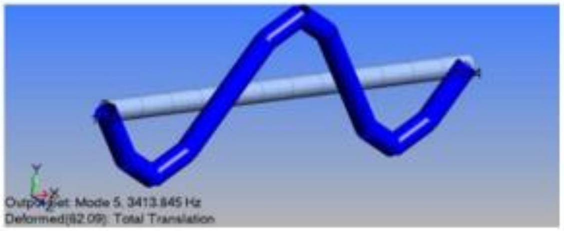


Doesn't Exist

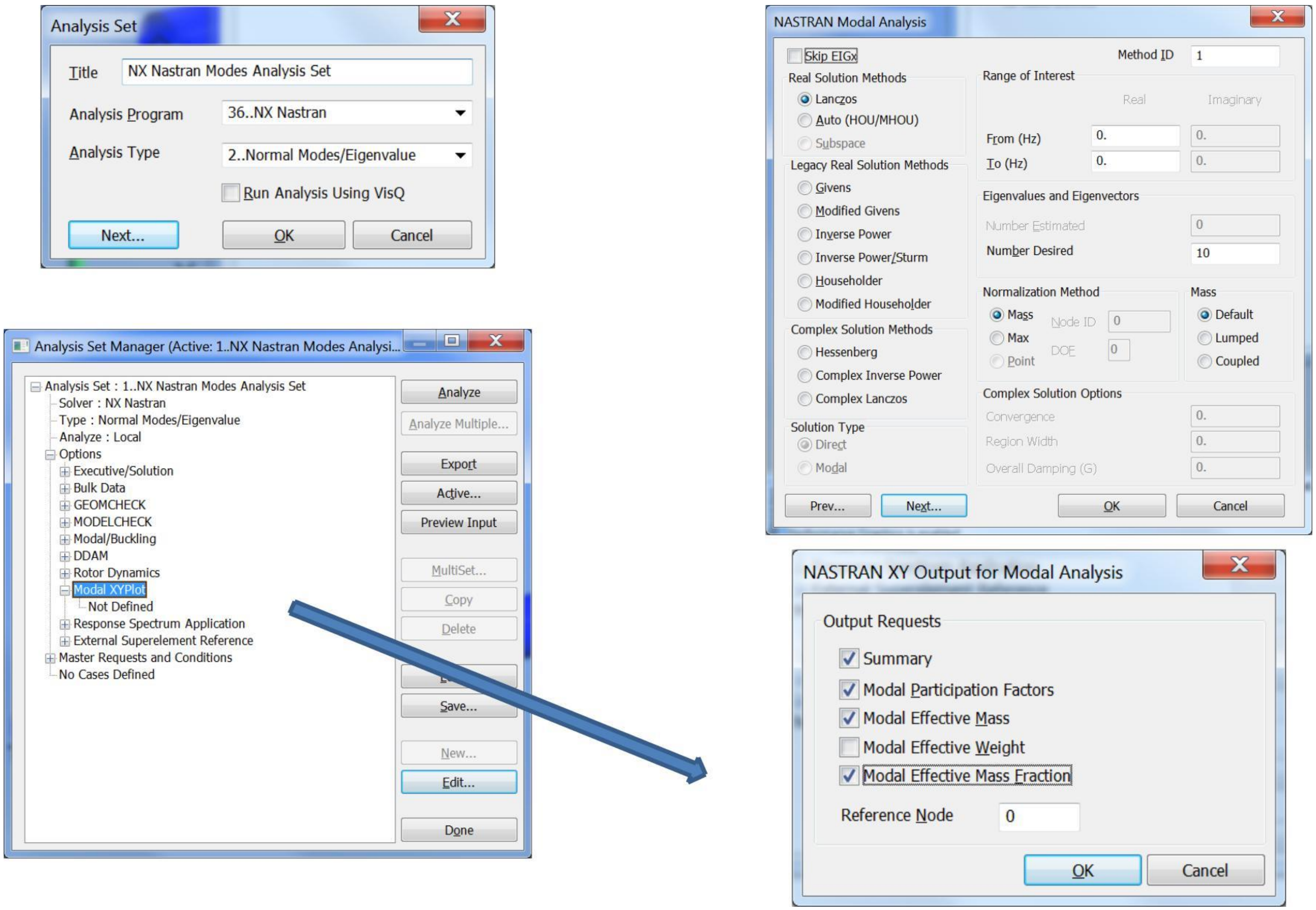
Twelve Element Mesh (Fine)







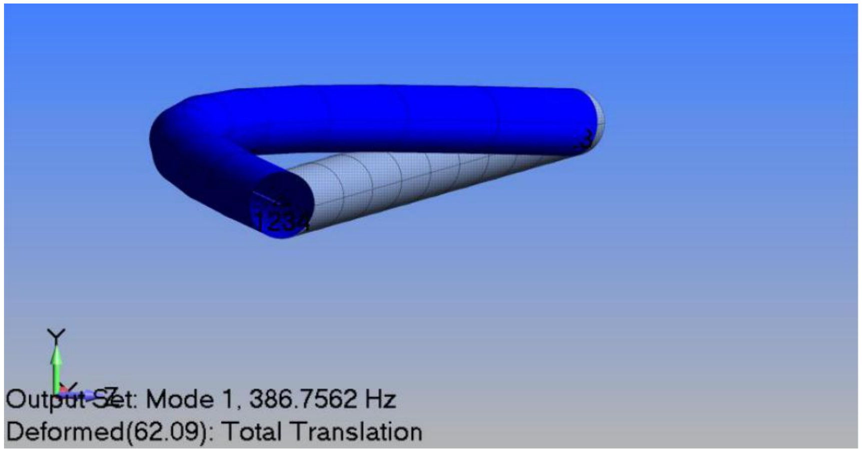
2.1.2 SETTING UP THE MODEL FOR NORMAL MODES WITH MASS PARTICIPATION



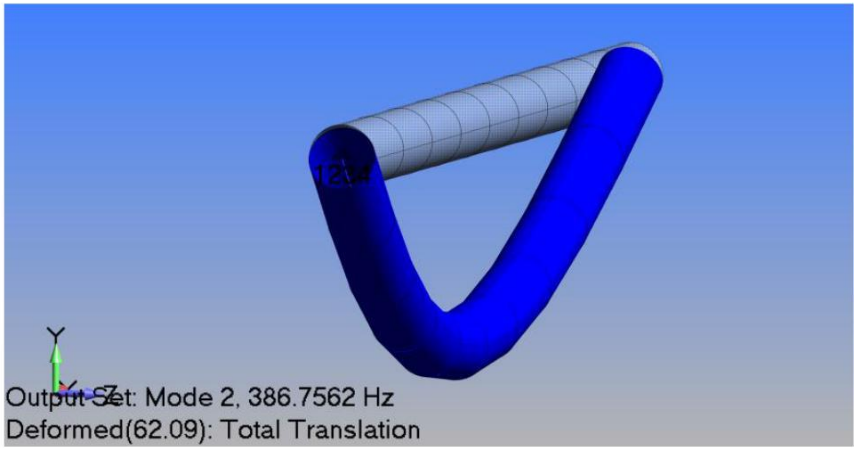
2.1.3 INTERPRETING RESULTS BASED ON ORTHOGONALITY AND MASS PARTICIPATION

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.

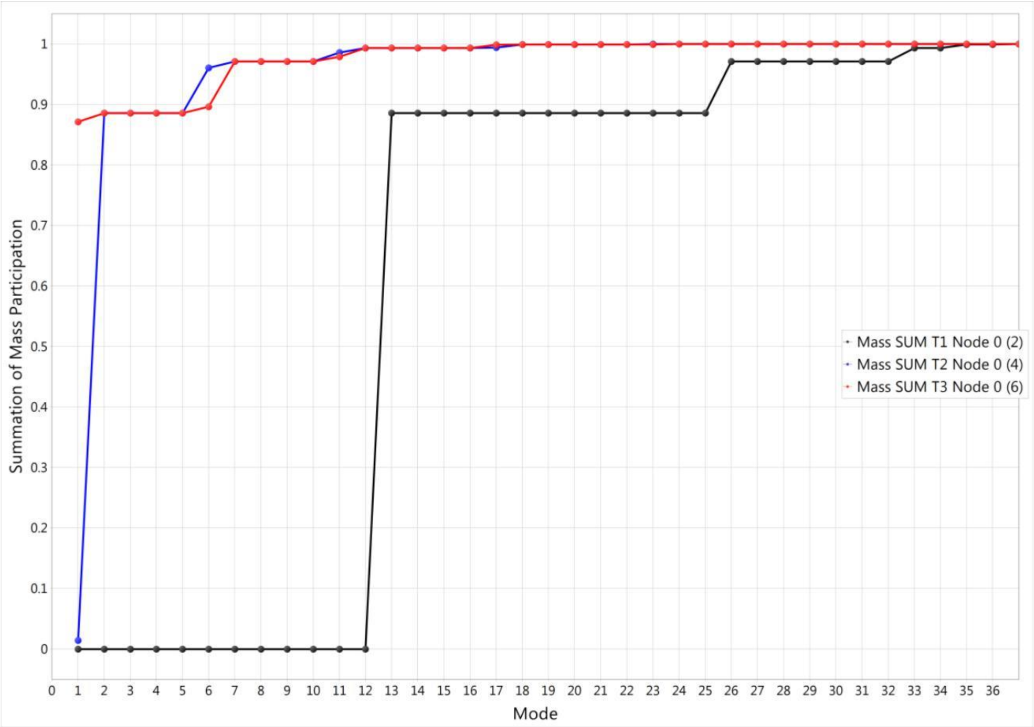
First Frequency 386.8 Hz



Second Frequency 386.8 Hz



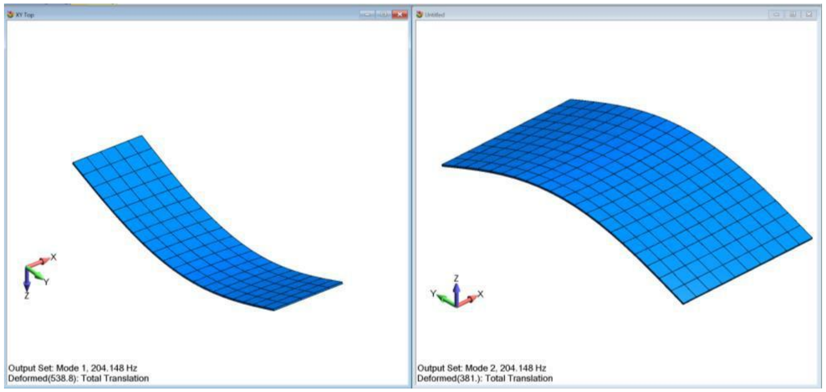
Mass participation tells you which modes have the “umph” and how many frequencies you need (modes) to accurately capture the dynamic response of the structure. On this later subject, a modal frequency analysis (e.g., PSD) formulates its response based on the number of modes chosen for the analysis. To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



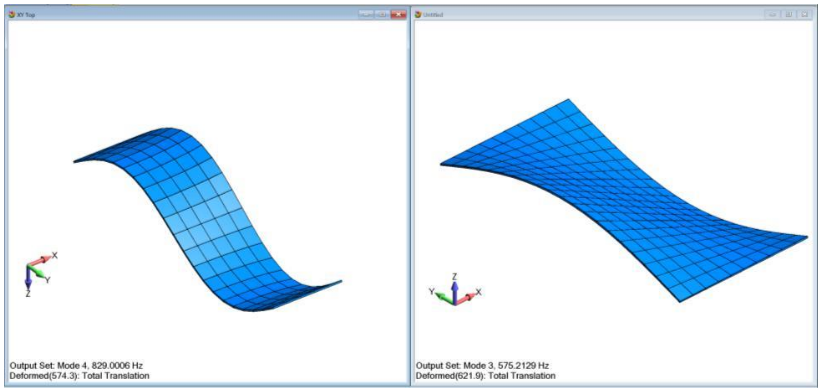
2.1.4 SYMMETRY AND FREQUENCY ANALYSIS

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.

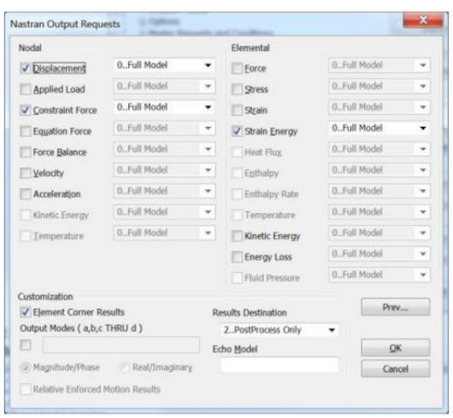


2.1.5 SIGNIFICANCE OF STRAIN ENERGY FOR FREQUENCY ANALYSIS

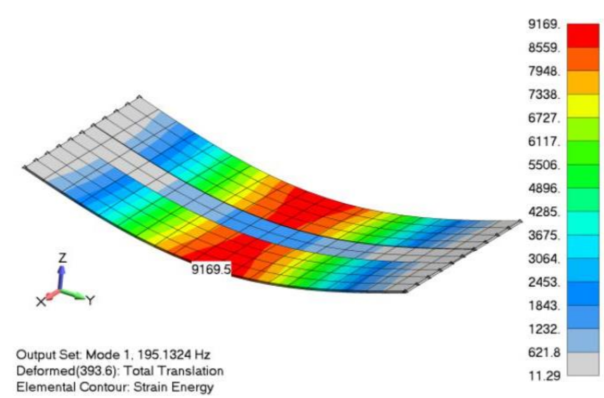
If one wants to move your natural frequencies up or down, sometimes intuition is good enough but it never hurts to have a quantitative tool. When a structure flexes or vibrates, there will be regions within the structure that are deforming more and other regions less. Since a natural frequency analysis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

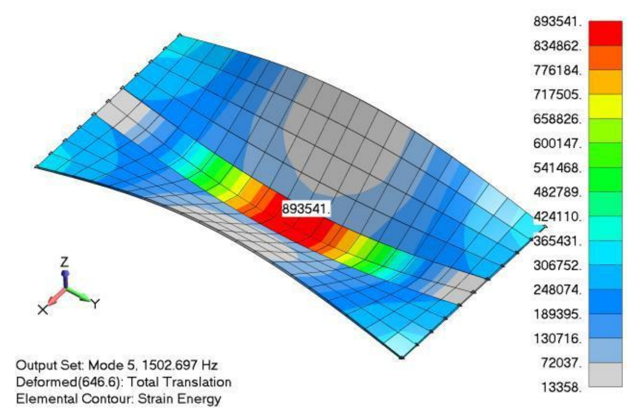
The default setup



Follow the Red - Increase the thickness of the outer strips



To Increase the Fifth Mode – Increase the thickness of the outer strips

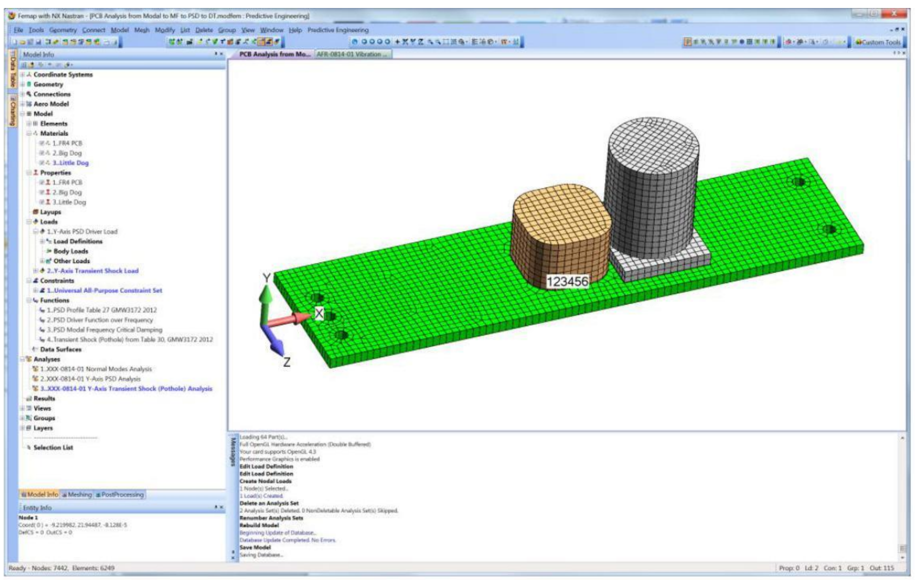


3. STANDARD NORMAL MODES ANALYSIS

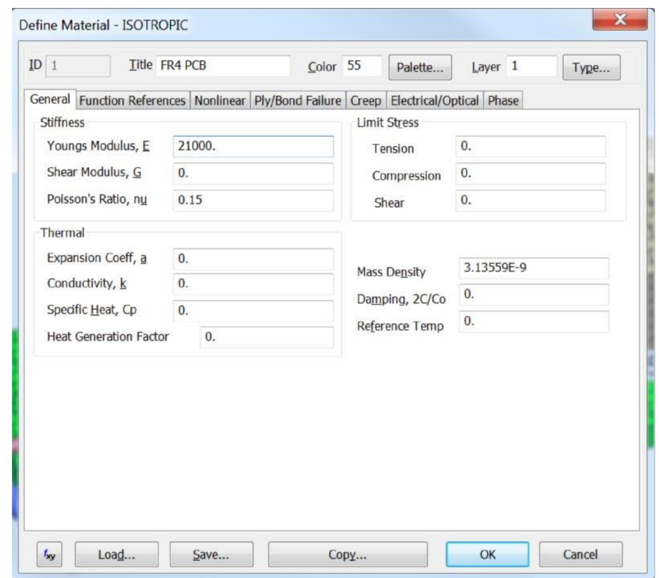
To see how this is applied in practice, we will run through an analysis project from start to finish (Normal Modes, Modal Frequency, PSD and Direct Transient). The model has been tweaked to protect the innocent.

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 Model



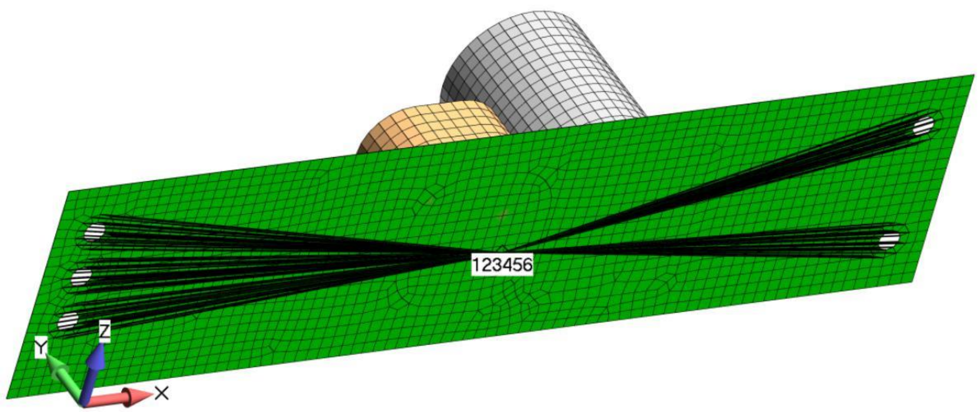
Units: N, mm, Tonne, s



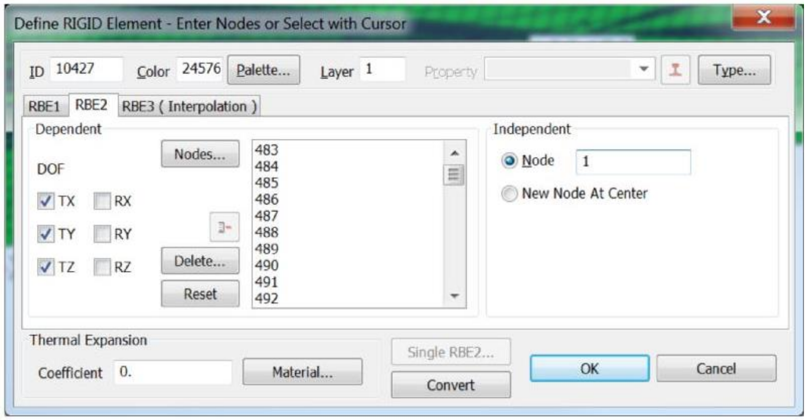
3.1 MODEL SETUP

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 Element to Common Node

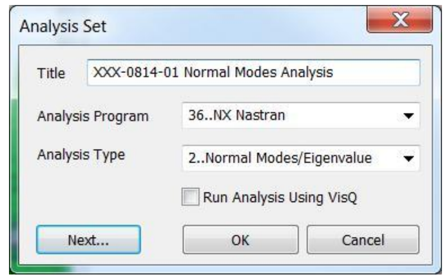


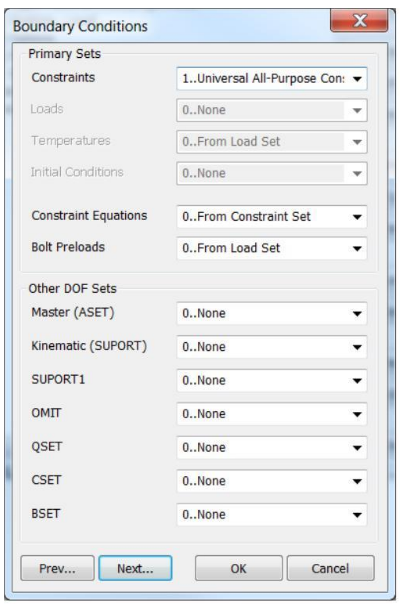
RBE2 with 3-DOF Dependent Nodes

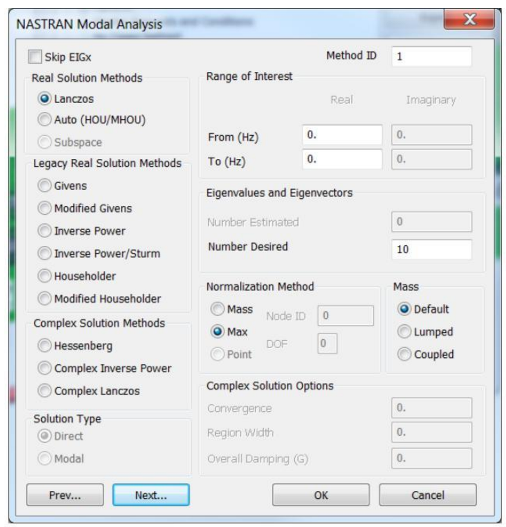


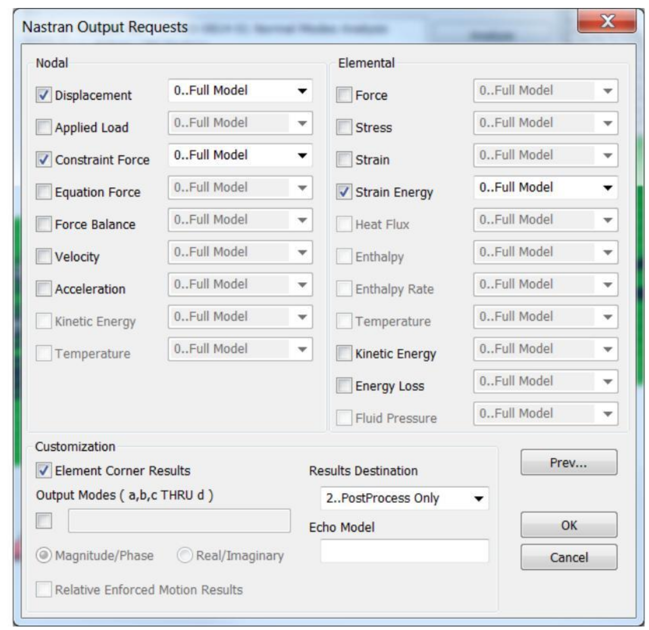
3.2 NATURAL FREQUENCY RESULTS AND INTERPRETATION

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.

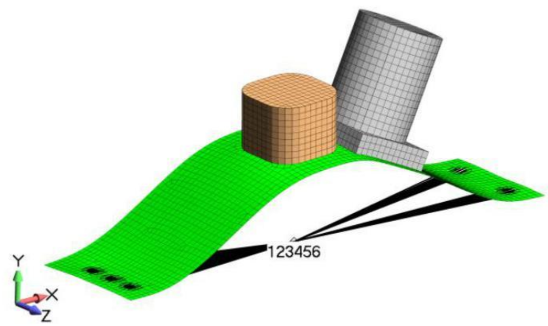




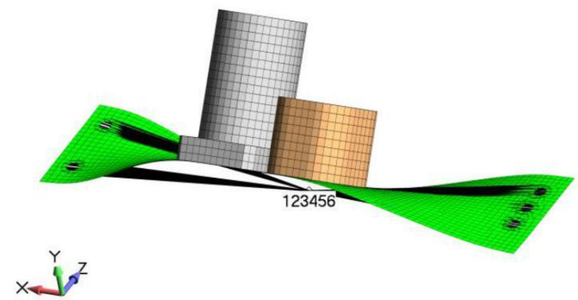




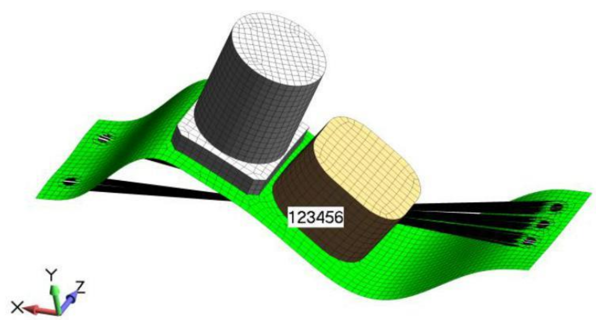
The mode shapes indicate the shape of that particular natural frequency. Since we are solving the EOM that has no {Force} or {Load}, the mode shapes have an arbitrary magnitude but they do tell us something very important. For example, the first mode flexes in the Y-direction and if excited in that direction, the structure would have a very strong response.



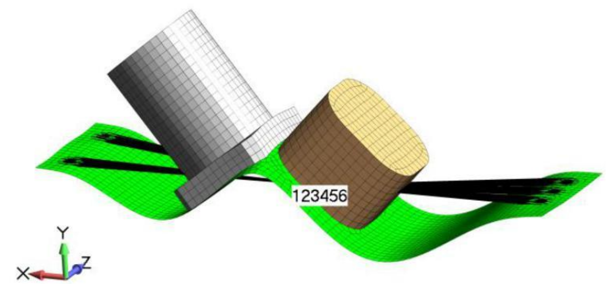
Output Set: Mode 1, 200.4665 Hz
Deformed(1.112): Total Translation



Output Set: Mode 2, 380.1996 Hz
Deformed(1.058): Total Translation



Output Set: Mode 3, 672.5712 Hz
Deformed(1.379): Total Translation

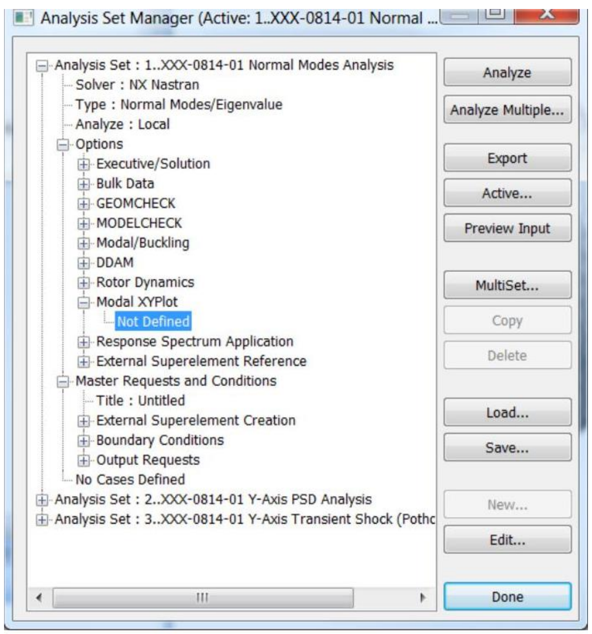


Output Set: Mode 4, 1015.848 Hz
Deformed(1.08): Total Translation

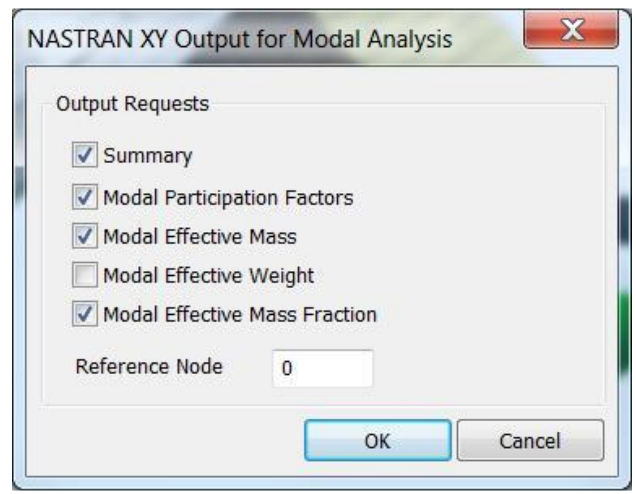
3.2.1 MASS PARTICIPATION

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).

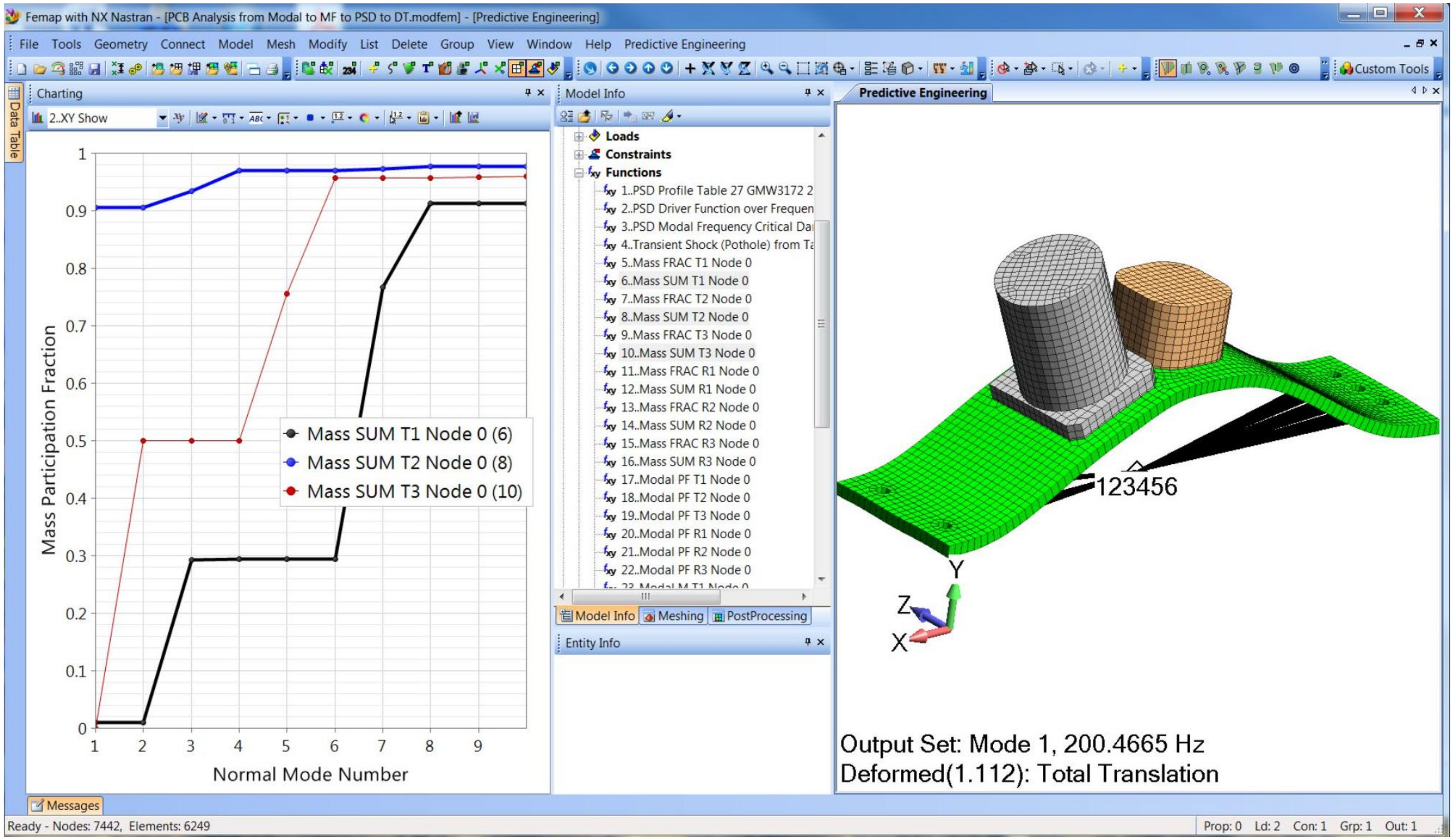
Analysis Set Manager/Normal Modes



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.



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:

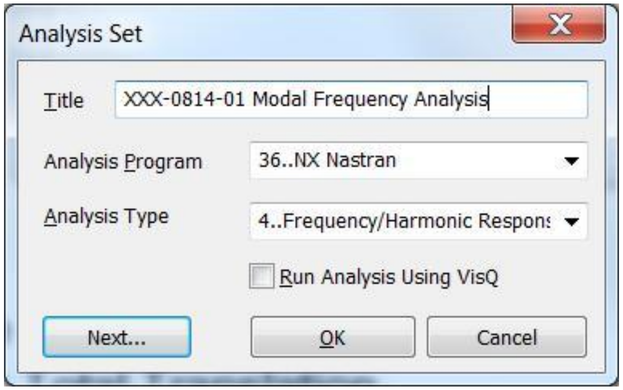
$$F_o \sin(\omega t - \theta) = m\frac{\partial^2 u}{\partial t^2} + c\frac{\partial u}{\partial t} + ku$$

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_o and Θ, and when requested, FEMAP can calculate the time varying response at any solved frequency (ω) as:

$$u = u_o \sin(\omega t - \theta)$$

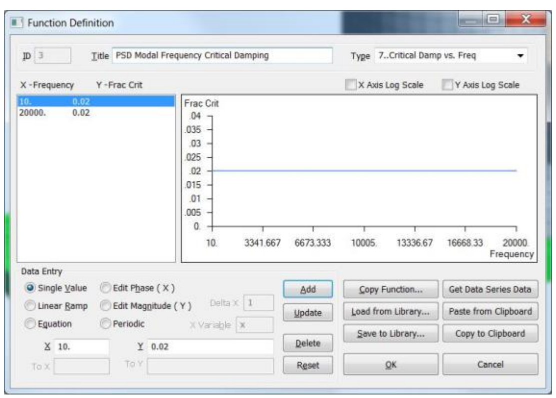
Thus, a modal frequency analysis assumes that the forcing function is sinusoidal and solves the EOM in the frequency domain with results kicked-out in the form of absolute magnitudes and phase analysis. This makes interpretation of the results somewhat challenging and requires a bit of understanding 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



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 Y-direction. 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.



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.

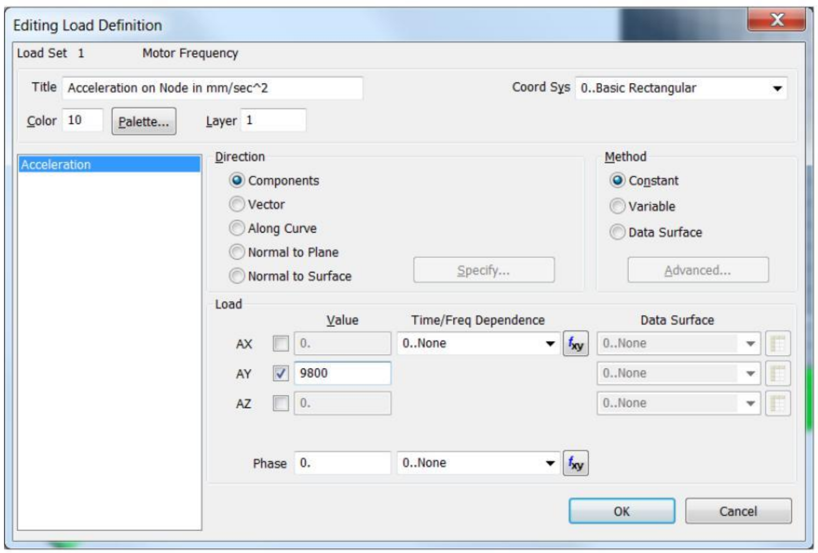
If one wants to know more, take a look at the documents:

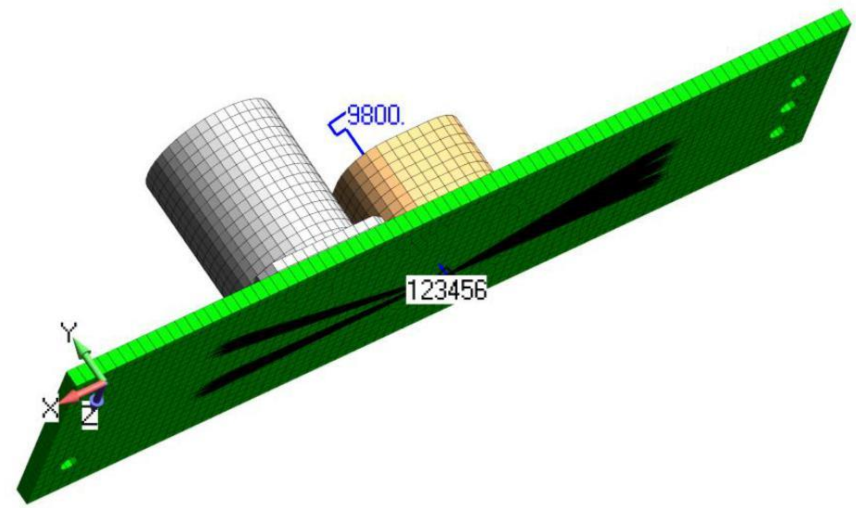
NX Nastran basic Dynamic Analysis

&

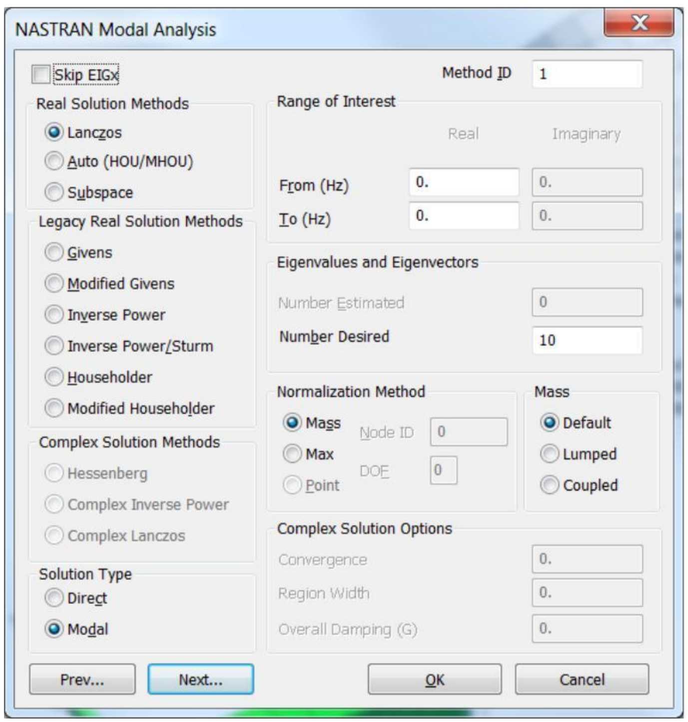
NX Nastran Advanced Dynamic Analysis User's Guide

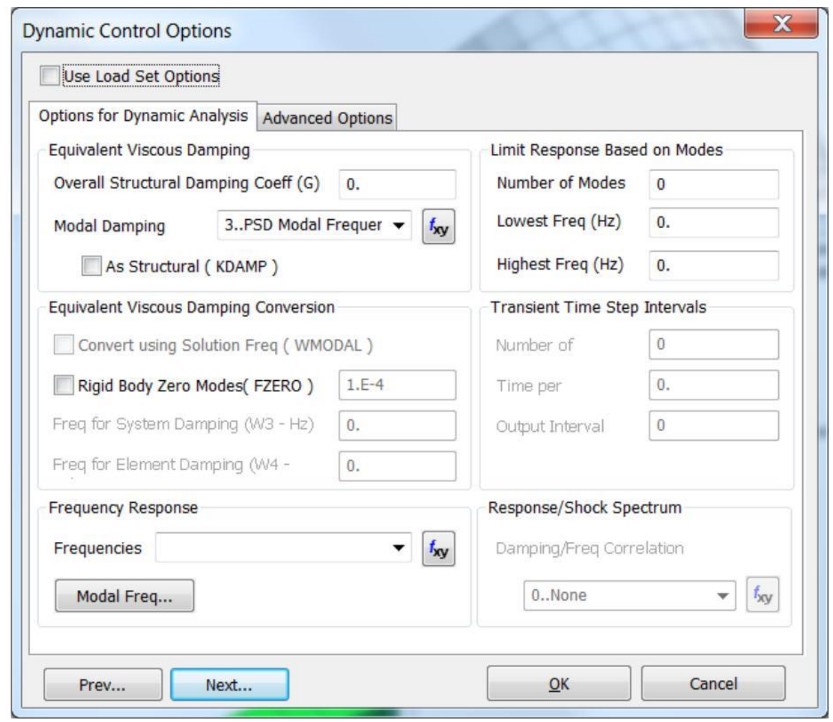
Units for dynamic analysis can be challenging. In this example model, the units are a modified SI system of N, tonne, mm and second. For the acceleration load of 1 g we have 9,800 mm/s2. The load is applied at the independent node of the RBE2 element. It may seem funny that one can apply an acceleration load to a node that has all six DOF fixed but the modal frequency analysis understands the request and ignores the T2 SPC tag.



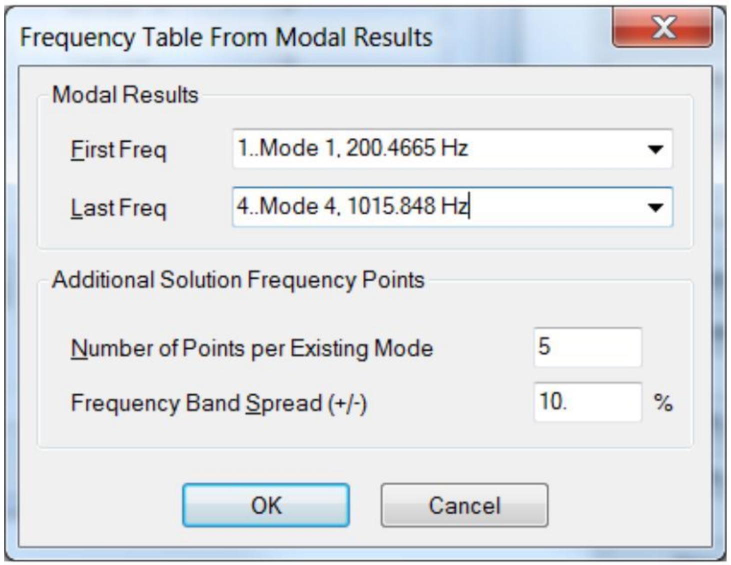


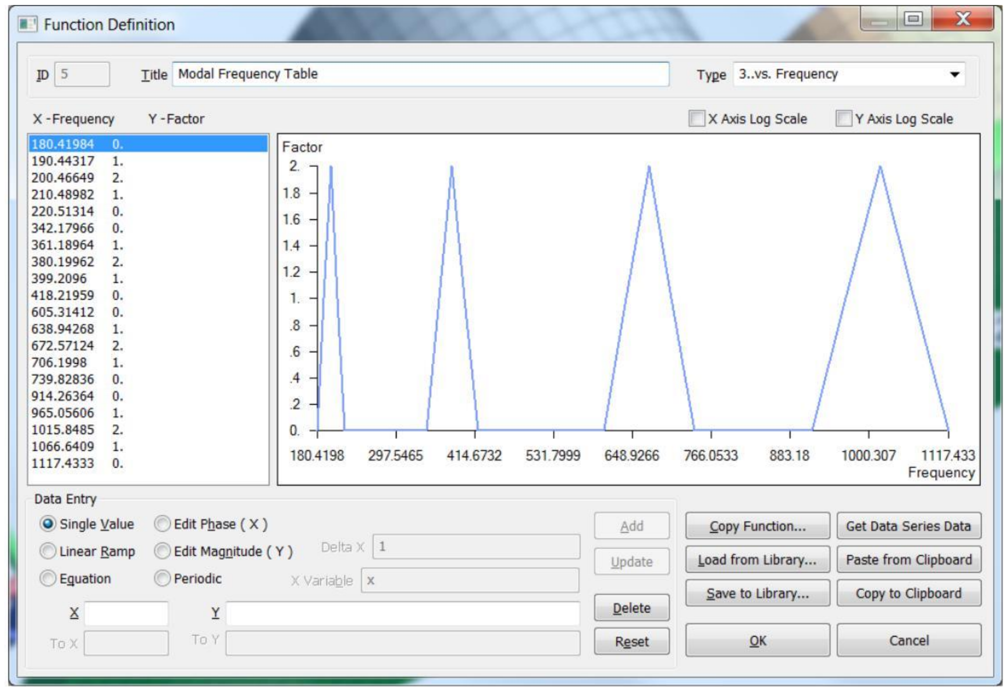
This is the heart and soul of the Modal Frequency Analysis setup. As one walks through the screens, we chose the Modal solution type, and request that 10 Eigenvalues and Eigenvectors be used to form the solution set. The next screen, we set damping to use our 0.02 critical damping curve and we request 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).



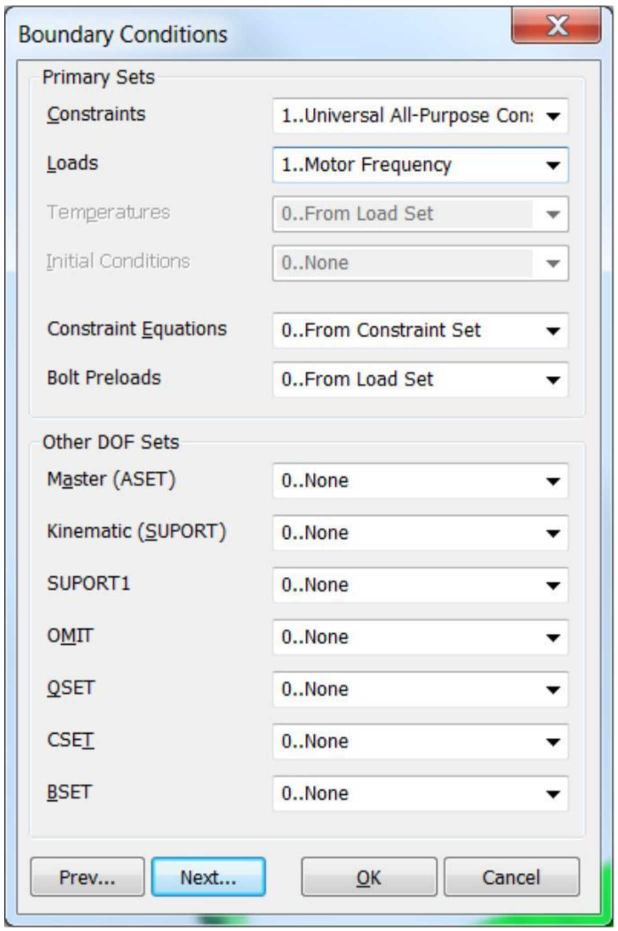


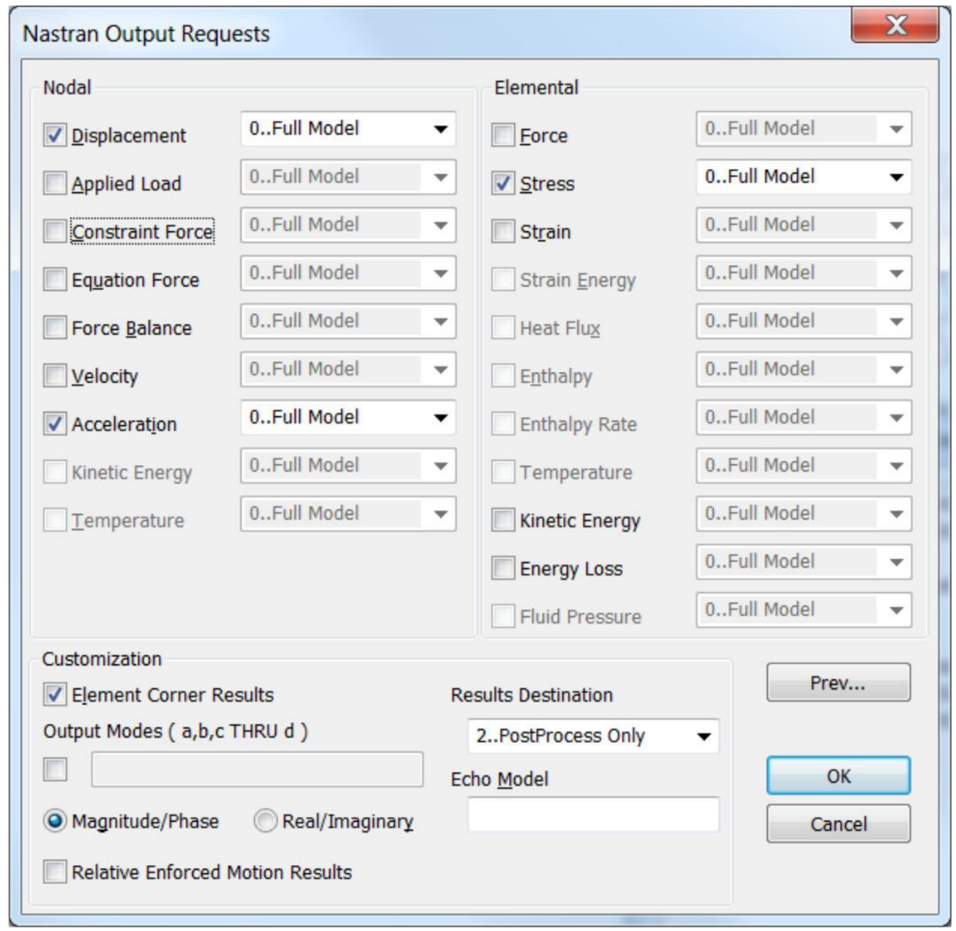
Since the linear dynamic response of a structure is determined or composed of its natural frequencies it often makes the most sense to request solutions at and around (Frequency Band Spread) these natural frequencies. Once this is done, the program creates a function showing how these solutions are spaced apart. The numerical value of the function is only for graphical utility since Nastran solves at each requested frequency.



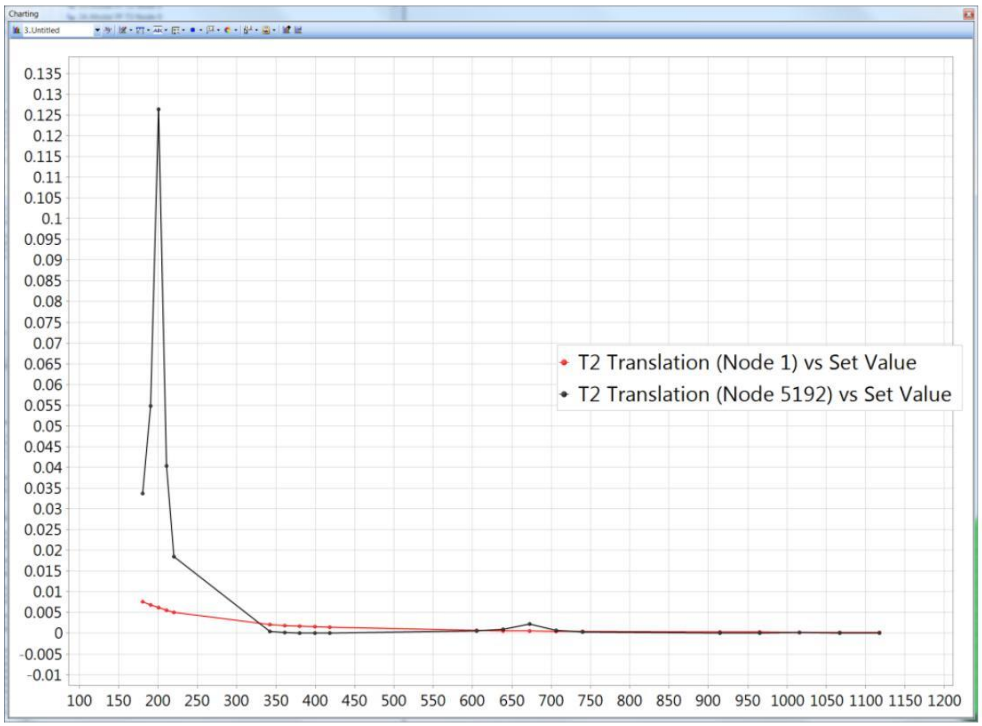


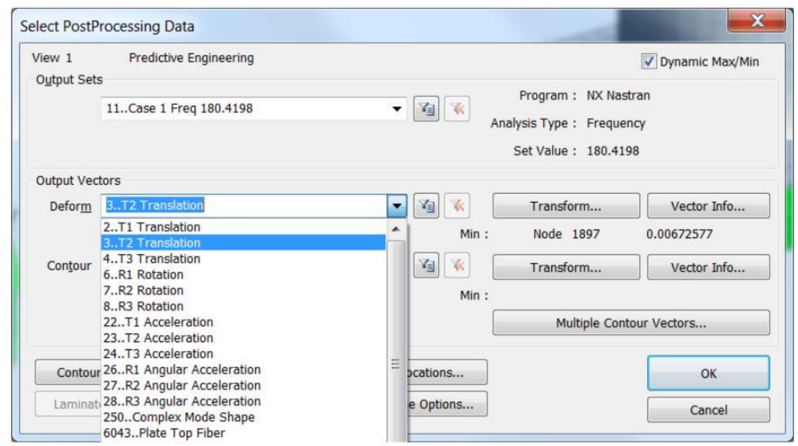
Then, one applies the boundary conditions and then lastly, one sets the output requests.



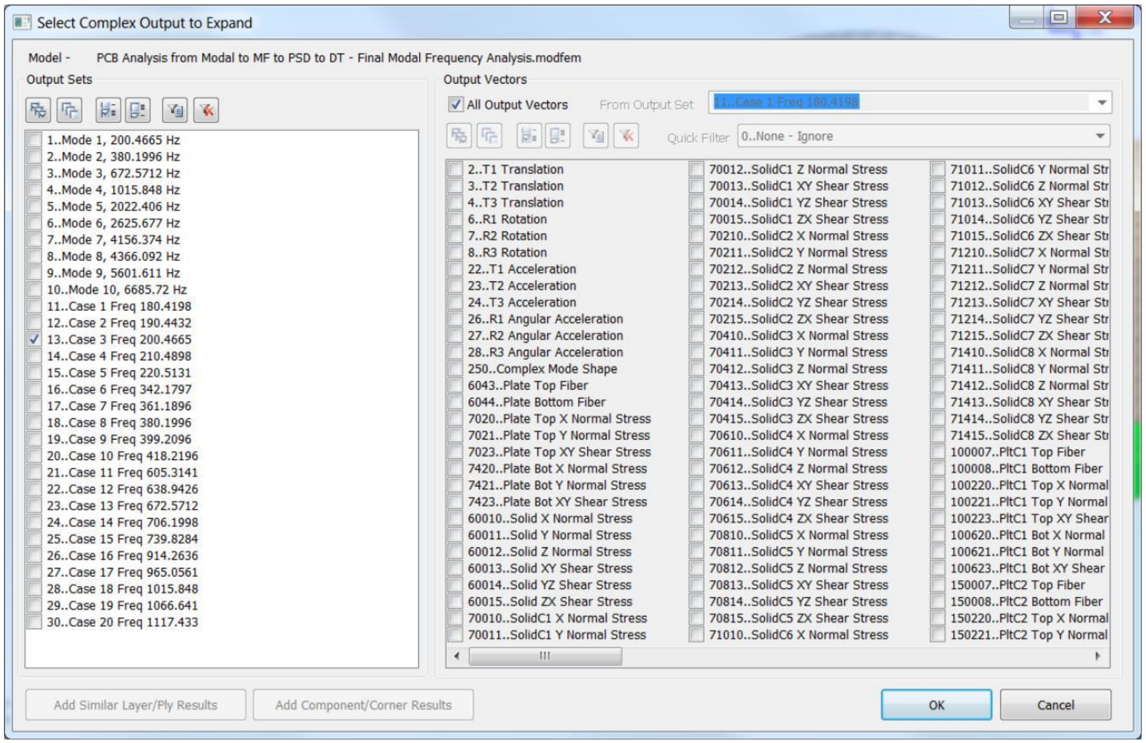


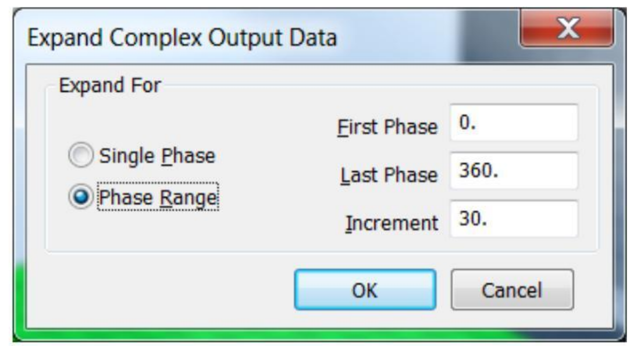
And the results show no surprises with the response peaking at the first normal mode at 200 Hz. If it is your first time with Modal Frequency, then the output results will seem a bit odd since you don't have Total Translation or a von Mises stress. All that you have are individual displacement and stress components. This goes back to the nature of the solution where the output is in magnitudes (uo) and phase angles (ϴ). Hence, to get the time varying nature, you need to expand the complex results.



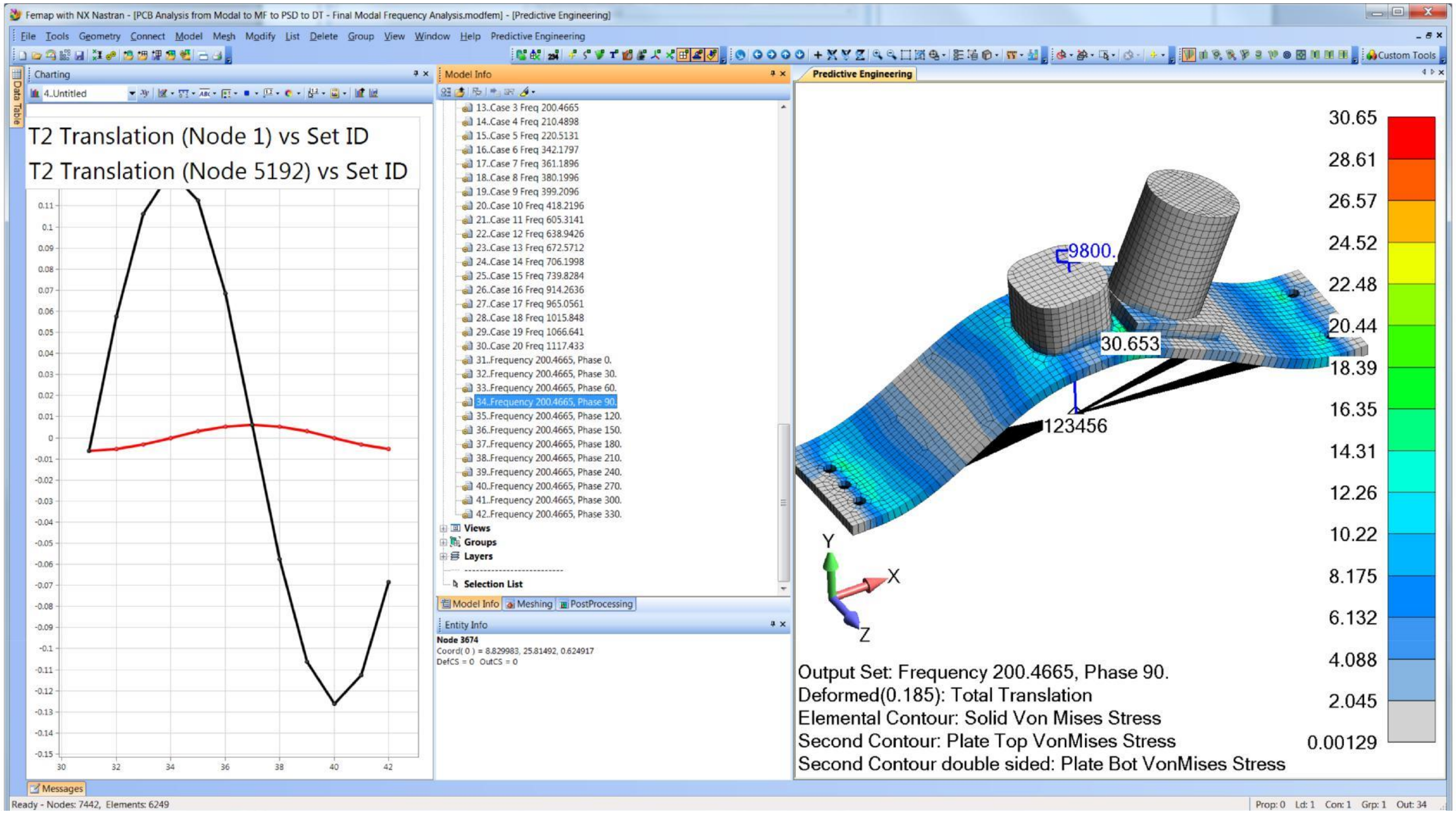


To obtain the time varying response from a Modal Frequency, one goes to Model / Output / Expand Complex and pick your solution of interest. For this structure it is the maximum response at 200 Hz and then we'll request that it is expanded into 12 solutions.





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. INTRODUCTION TO RANDOM VIBRATION

Random Vibration is vibration which can only be described in a statistical sense. The magnitude at any given moment is not know, but is instead described in a statistical sense via mean values and standard deviations

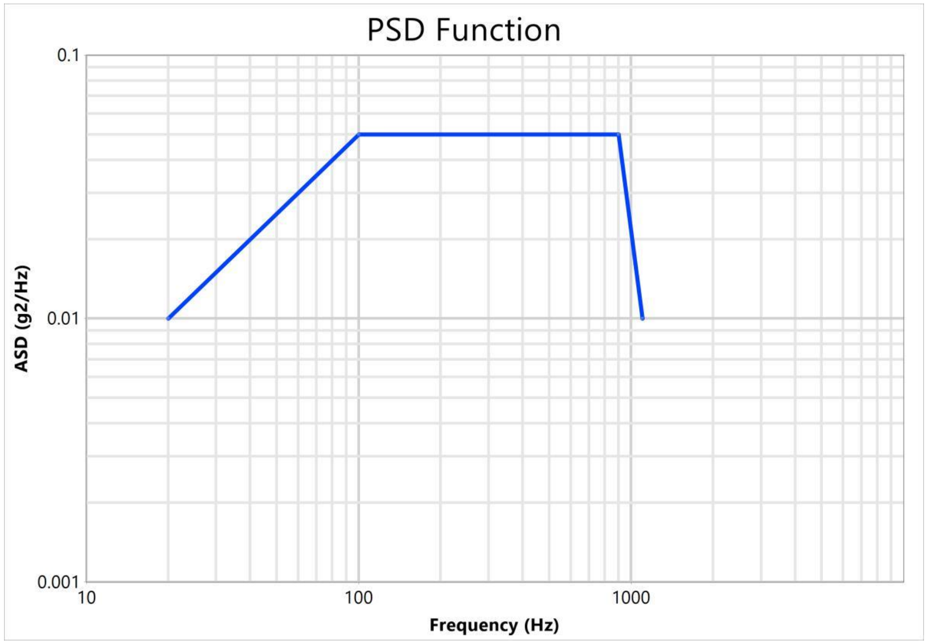
Random vibration problems arise due to earthquakes, tsunamis, acoustic excitation (e.g., rocket launches), wind fluctuations, or any loading which is inherently random. Often random noise due to operating or transporting conditions can also be considered. These vibrations are usually described in terms of a power spectral density (PSD) function.

5.1 THE PSD FUNCTION

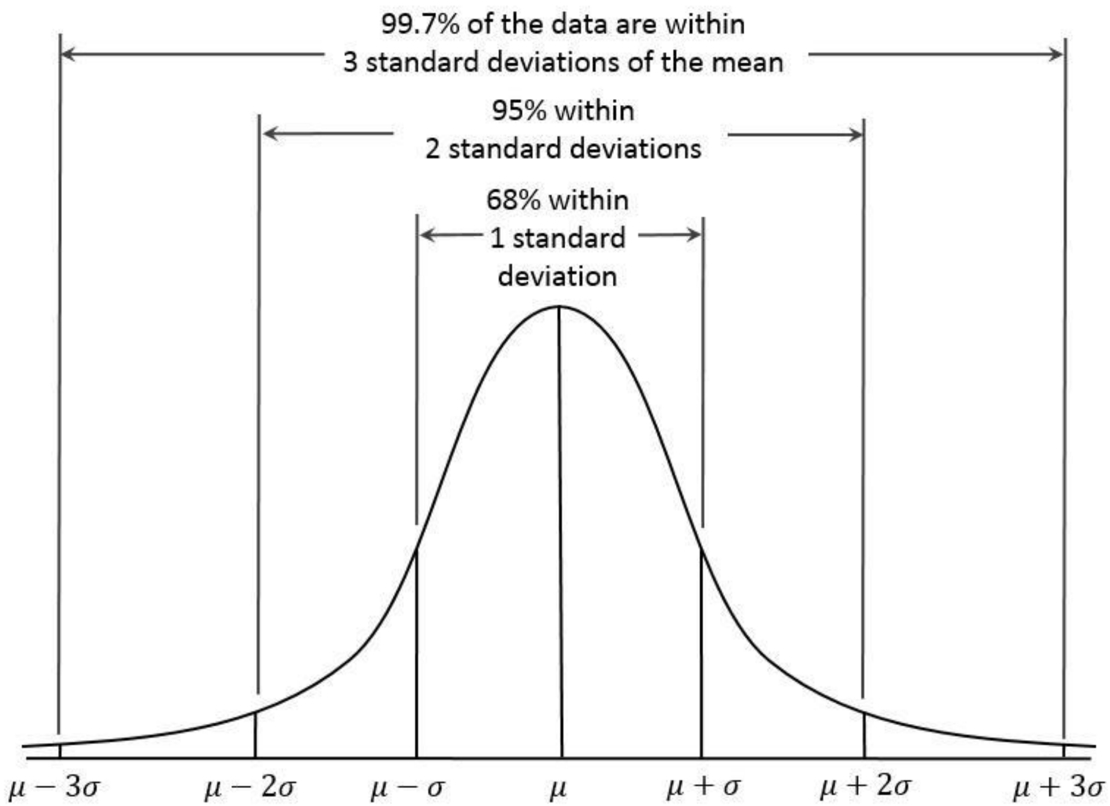
Random vibration is unique because it can excite all frequencies at once, whereas a sine sweep will excite one frequency at a time (think slamming all keys on a piano instead of sliding your hand across them). The PSD function is created by subjecting a structure to white noise vibration and measuring the RMS amplitude of the response of the structure across a range of frequencies, squaring the response, and dividing it by the frequency range which results in units of G2/Hz.

A typical power spectral density is shown below:

Frequency (Hz)	PSD (G^2/Hz)
20	0.01
100	0.05
900	0.05
1,100	0.01



A system subject to random vibration does not have a single resultant stress. Luckily for us, the stress results do typically follow a Gaussian distribution (think bell-curve):



The Gaussian distribution allows stress results to be reported statistically. FEMAP will generate 1-σ stresses, which represent the stress that the system will likely see 68% of the time. The 2-σ stress level covers 95% of cases, and 3-σ covers 99.7%. Most of the time a system is designed to the 3-σ stress level.

5.2 THE NX NASTRAN METHOD

Given an input PSD function, an output response can be calculated by using the systems transfer function.

$$PSD_{out} = |g(w)|^2 PSD_{in}$$

The g(w) represents the system transfer function. A system transfer function simply represents its output to input ratio. NX Nastran performs a frequency response analysis on the system to obtain the system transfer function, and then does the random vibration analysis as a post processing step based on this transfer function.

There are several steps to setting up the analysis in FEMAP:

1. Defining the system damping
2. Creating the PSD function
3. Creating a Modal Frequency Table (or Requested Solutions Function)
4. Creating the excitation node and tying it into the model
5. Loading the model
6. Constraining the model
7. Specifying output groups for nodal and elemental output
8. Setting up the analysis in the Analysis Manager

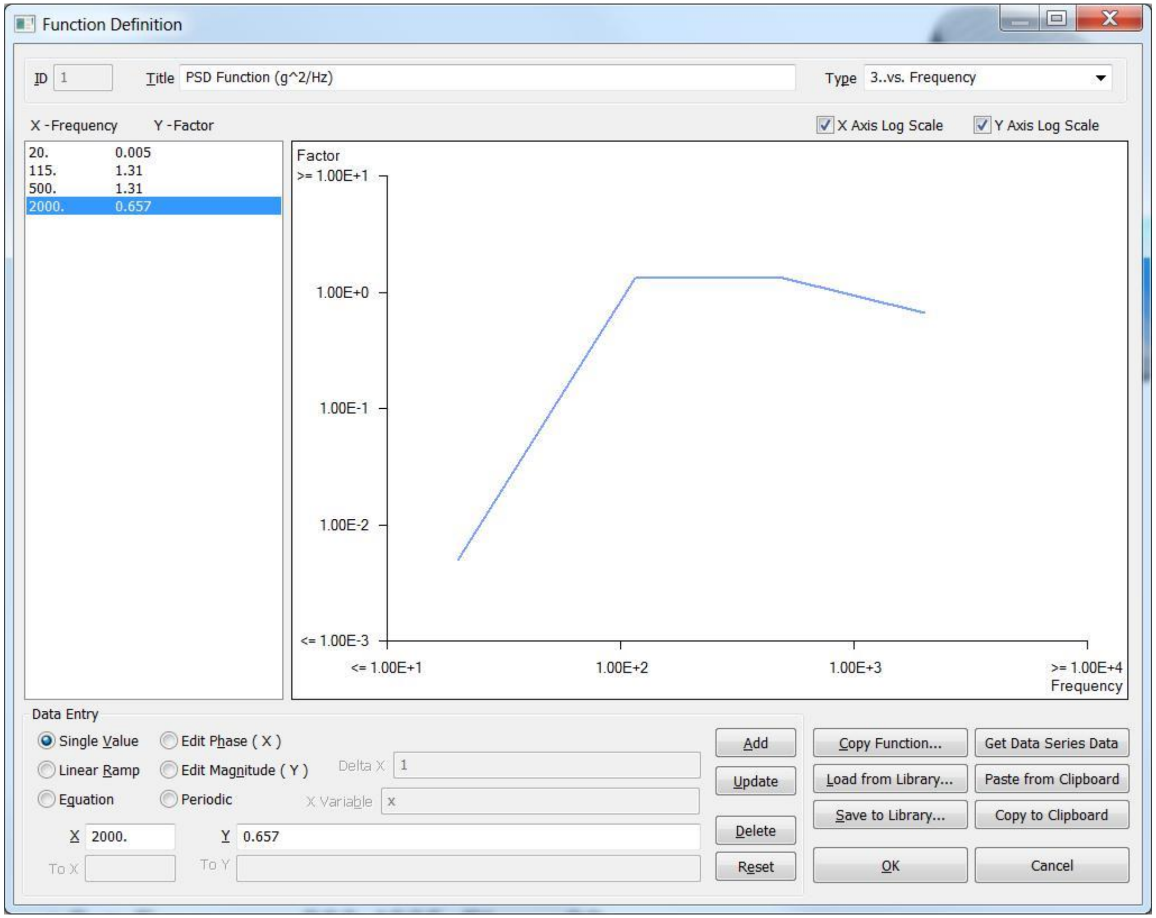
5.3 PSD UNITS

It can be tricky keeping track of the units in any analysis; this is especially true with PSD analysis. The table below shows the input and output units for a few of the most common unit systems. It doesn't matter which system you go with but be sure you are consistent throughout your analysis. The table below shows the properties for aluminum in each unit system.

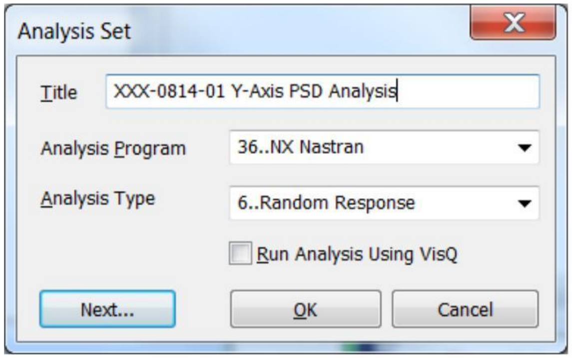
Units	Inputs				RMS Outputs	
	Young's Modulus	Mass Density	PSD Function	Acceleration Load	Deflection	Stress
SI (m,kg,sec)	6.89e10 Pa	2710 kg/m^3	1 g^2/Hz	9.807 m/s^2	1 m	1000 Pa
SI (mm,Mg,sec)	6.89e4 MPa	2.71e-9 Mg/mm^3	1 g^2/Hz	9807 mm/s^2	1000 mm	1.0e-3MPa
Imperial (in, snail, sec)	10.0 e6 psi	2.54e-4 snail/in^3	1 g^2/Hz	386.1 in/s^2	39.37 in	0.145 psi

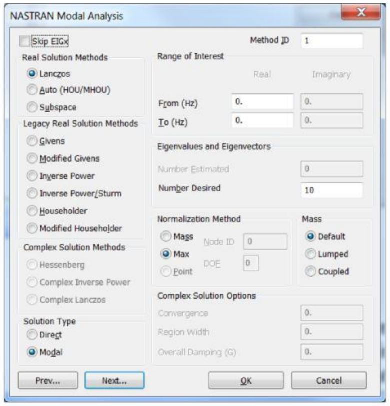
6. EXAMPLE 1: PSD ANALYSIS OF PCB WITH TWO HEAVY ELECTRICAL COMPONENTS

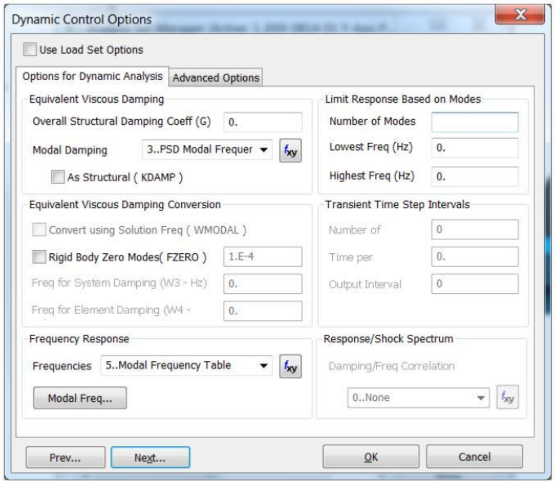
For completeness, let's do a simple PSD analysis on our circuit board. Again, units are very important. The PSD spectrum (load) is given as g2/Hz. In the center of the spectrum from 115 to 500 Hz, the PSD input is 1.31 g2/Hz and then tapers.

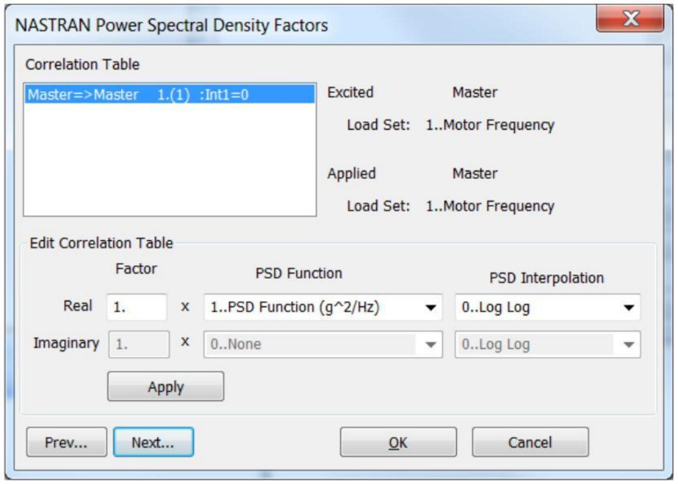


The PSD procedure is almost identical to the modal frequency analysis. There are some new screens but the only critical one is where you apply your PSD spectrum (lower-right-hand-corner). Otherwise, it is identical to that of the prior modal frequency analysis.

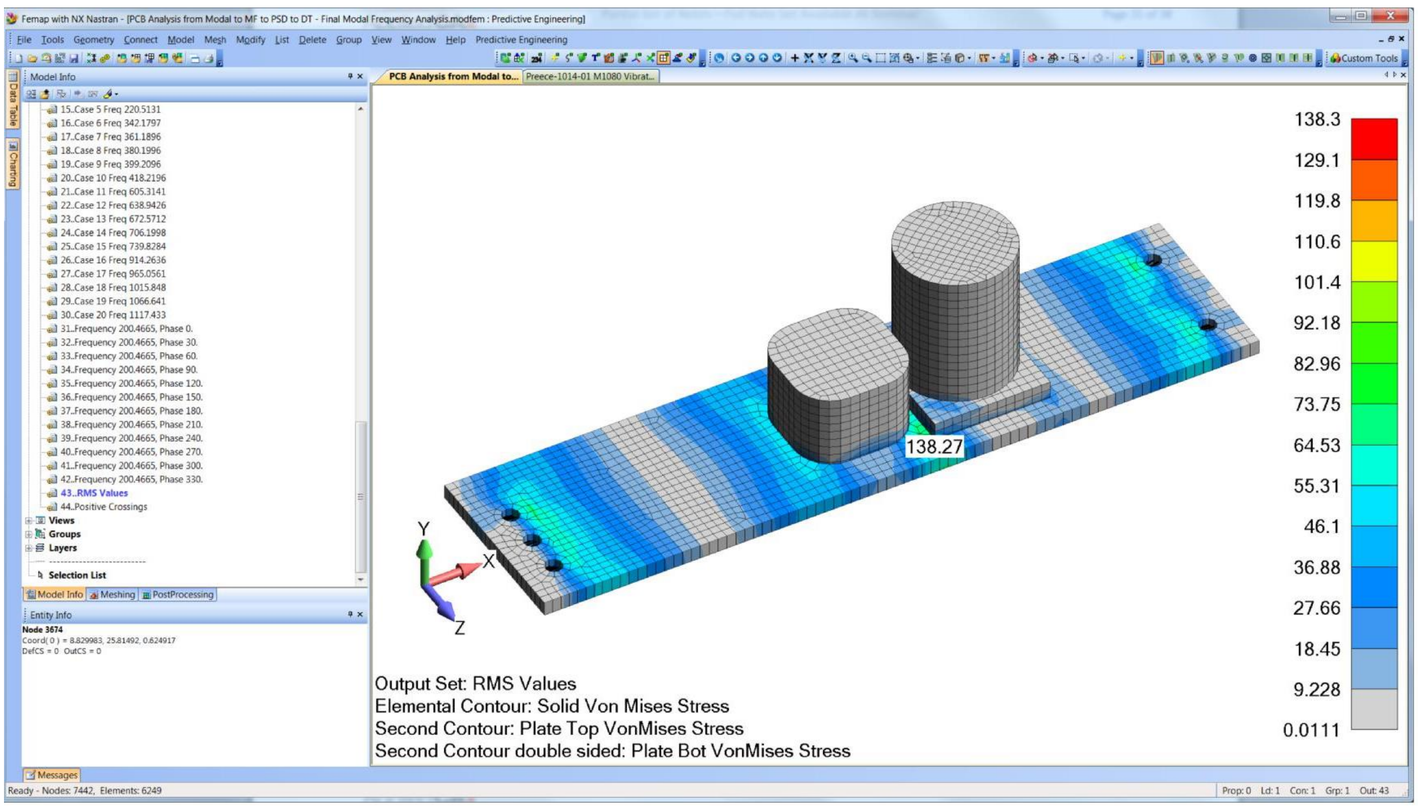








Given that a PSD analysis can be a numerically intensive calculation, FEMAP provides the ability to restrict your analysis output to just a few items or the complete model. For this analysis, all output requests are left blank except the very last screen where just displacements and stresses are requested. 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.



7. EXAMPLE 2: CANTILEVER BEAM

7.1 PROBLEM DEFINITION

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

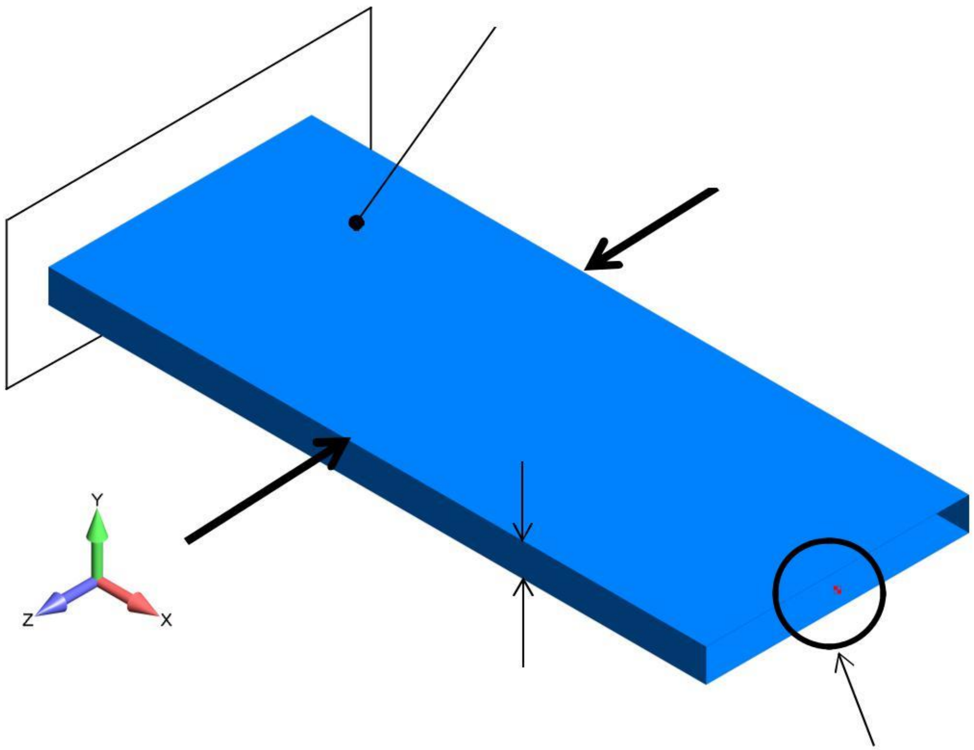
The FEA model is a single beam element. A picture of the beam element, with its cross section displayed is shown on the right.

We will compare the FEA results to an analytical solutionψ. The PSD input used by St

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

This Excitation was applied to the fixed end of the beam (where the rectangle is drawn)

Our unit system is lb/in/s and 1 g=386 in/s^2



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

7.2 ANALYTICAL SOLUTION

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

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

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

$$f_n = \frac{1}{2\pi}\sqrt{\frac{g}{Y_{St}}} = 110.5 \qquad Q = 2\sqrt{f_n} = 21$$

Miles' equation can be used to approximate the G_{out}(RMS) value:

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

This output is in G, if an equivalent value is desired in English units, simply multiply this by gravity.

$$27G = 27\,\frac{acceleration}{gravity} * gravity = 10{,}422\ {}^{in}/_{s^2}$$

The max output PSD can also be obtained using:

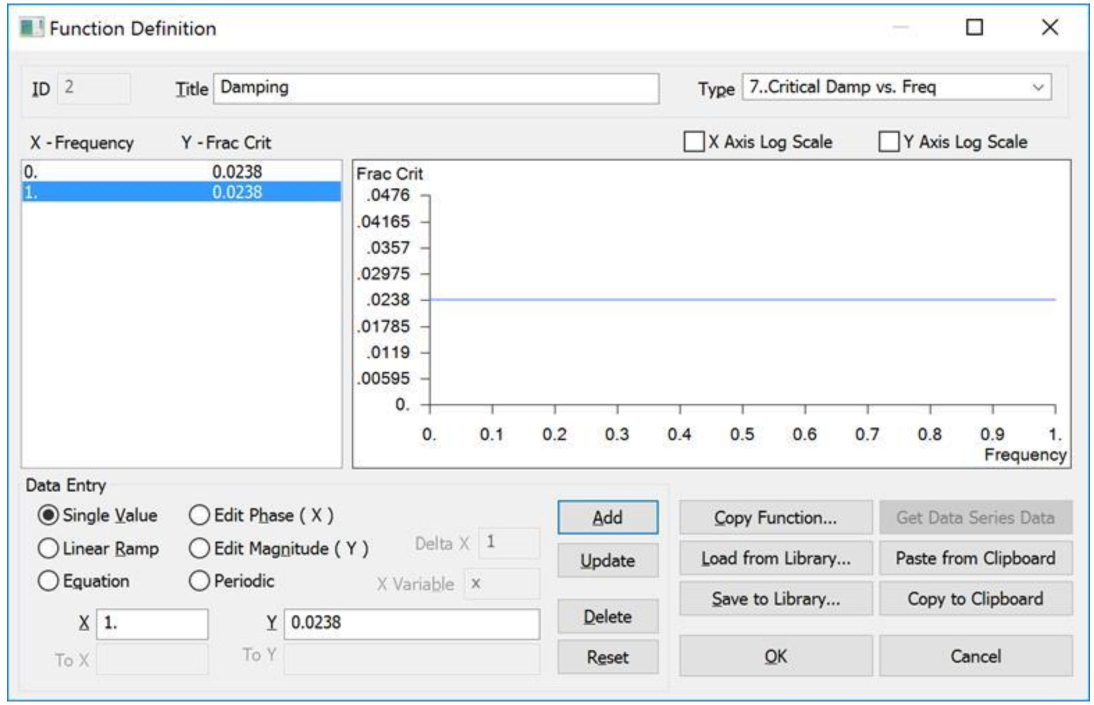
$$PSD_{out} = Q^2 * PSD_{in} = 21^2 * (0.2 * G^2)\ where\ G = 1g\ or\ 386\ in/s^2$$

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

7.3 DEFINING THE SYSTEM DAMPING

Determining how the system is damped can be complicated. In NX Nastran there are three ways to do this:

1. If the structural damping coefficient (G) is known then function type 6: Structural Damping vs. Frequency should be used.
2. If the critical damping ratio is known, then function type 7: "Critical Damping vs Frequency" should be used.
3. If the Quality/Magnification factor (Q) is known, then function type 8: "Q Damping vs. Frequency" should be used.

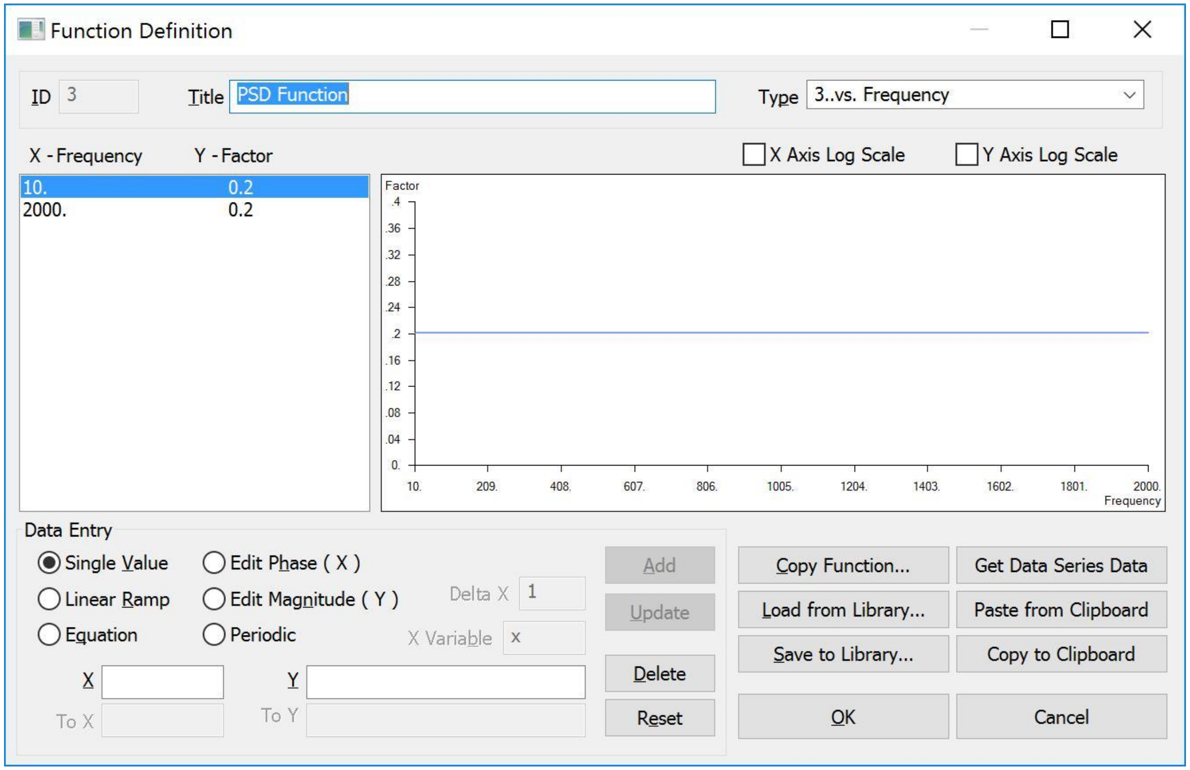


An approximation of the transmissibility of the beam is Q = 21. This value yields a critical damping ratio of 2.38%; this is what we will use.

7.4 CREATING THE PSD FUNCTION

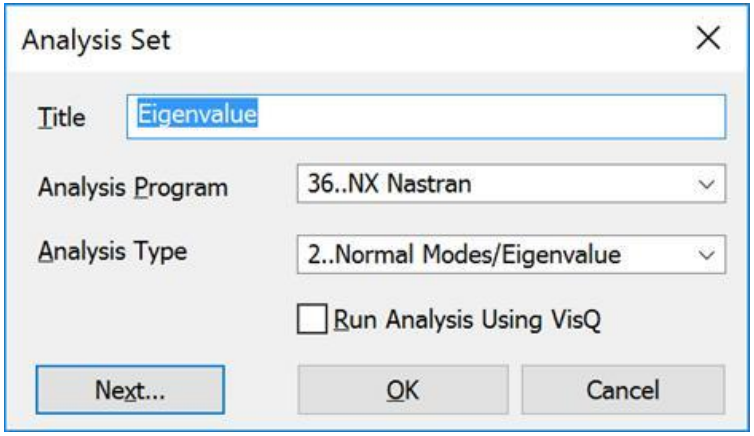
The input to the cantilever beam is a white-noise vibration with a PSD input of 0.20 G2/Hz from 20 to 2000 Hz.

This is entered directly with no scaling. It will be scaled for the desired unit system in the Load Definition dialog (Section 5.6).



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

The Modal Frequency Table is a function which defines which frequencies NX Nastran will obtain a solution for; that is, each frequency represents a separate solution that is written out to the results file. The function can either be created manually, or FEMAP can create one for you. If you do not know about which frequencies you'd like the analysis to focus, it is preferable to have FEMAP set it up, otherwise you will most likely end up with a large amount of extraneous output.

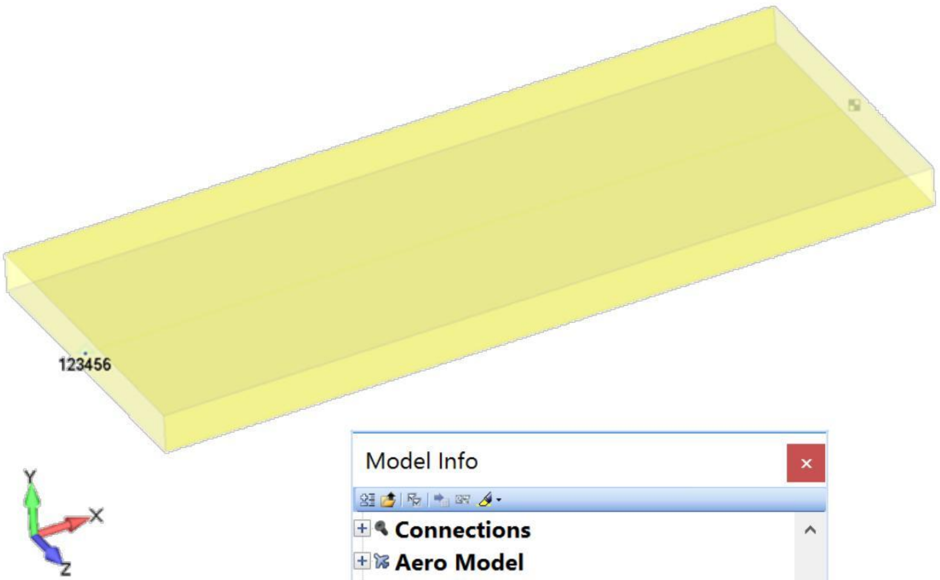


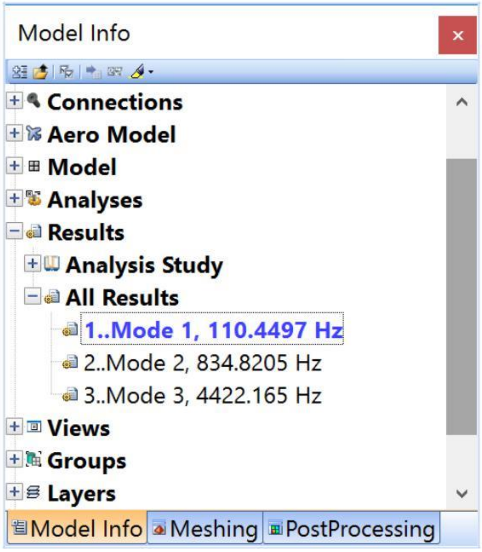
To have FEMAP set up the table for you, you must first run an eigenvalue analysis. Once the eigenvalue analysis has run, FEMAP will know about which frequencies to concentrate.

The normal modes will be used to define the solution frequencies of the Random Analysis. Think of it as guiding the Random Analysis such that only frequencies of interest (significant frequencies) are processed. This greatly limits the amount of post-processing that is required for the Random Analysis. More will be said on this later on….

It is good practice to run the normal modes analysis first to see how the structure will behave. In this simple beam model, we have fixed the end of the beam in all six DOF. The beam is also massless (material density of 0.0). This was done to allow us to exactly match the analytical solution.

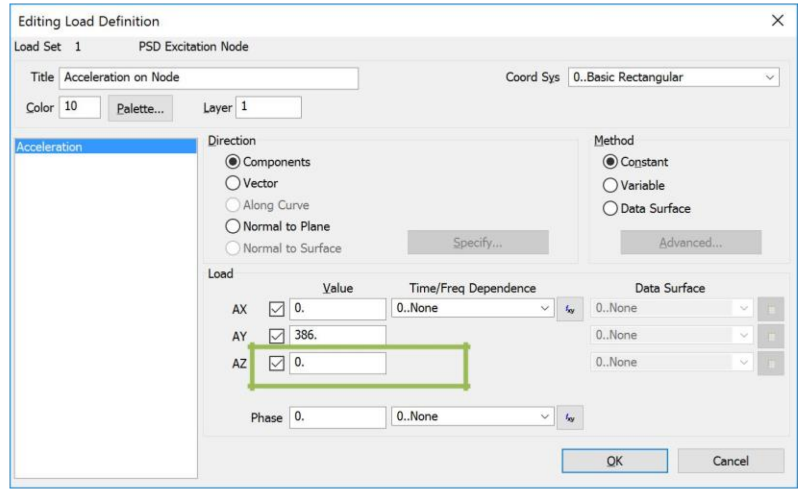
After the analysis has finished running, you should have three modes. In Section 5.8 we will show you how these Normal Modes are used to generate the Solution Frequencies for the Random Analysis.





7.6 LOADING & CONSTRAINING THE MODEL

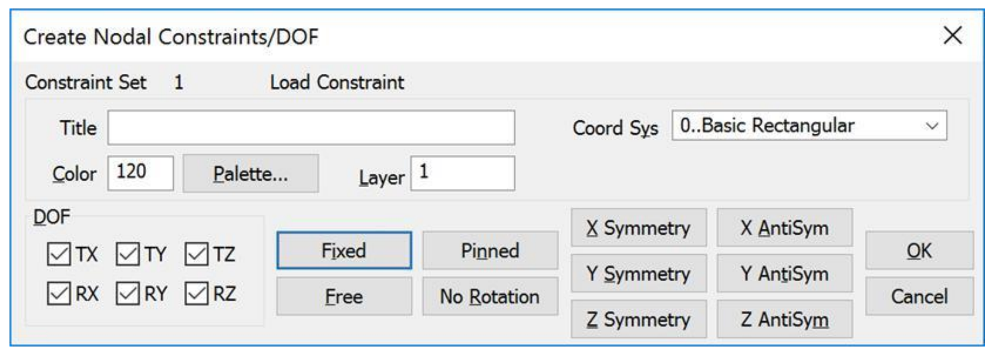
An acceleration load must be given to the base node in the direction of the excitation. Since the PSD is given in G^2/Hz, we must scale the load by a 1 g gravitational acceleration in our unit system of choice. We want our deflection results in inches so we will enter an acceleration of 386 in/s^2.



The **Load Constraint** constrains the base node in all sex degrees of freedom

This constraint set should identical to the constraint set used for the eigenvalue analysis. The node used to constrain the model is the same node to which the unit acceleration was applied.

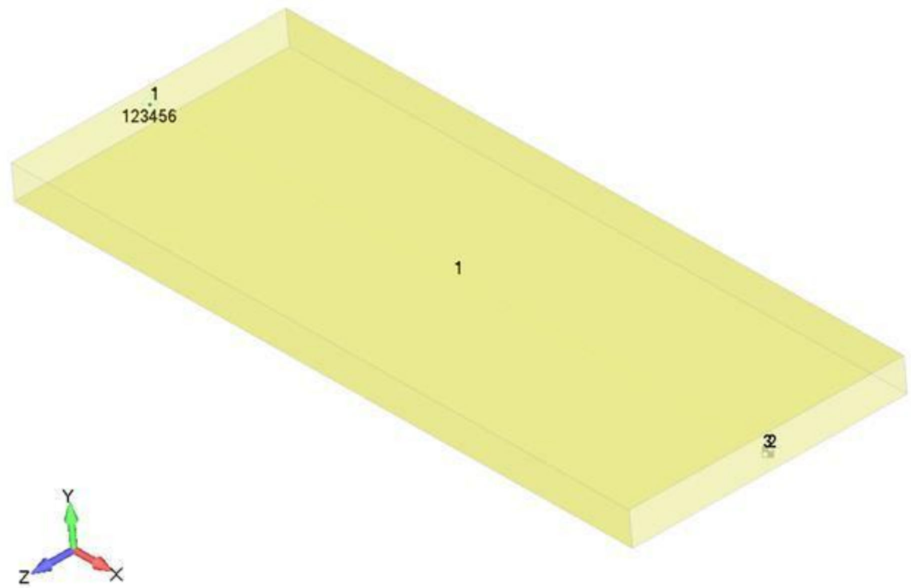
The idealization concept is that the base is fixed in the TX, TZ, RX, RY, RZ while the structure is excited in the Y-direction (i.e., there is displacement in the Y-direction).



7.7 SPECIFYING GROUPS FOR NODAL AND ELEMENTAL OUTPUT

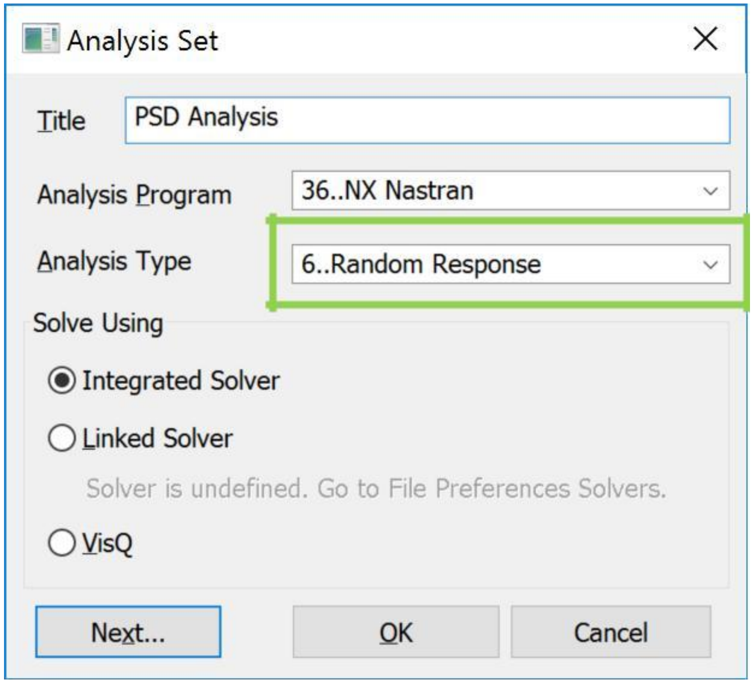
A group can be created to specify certain nodes and elements to recover data from. For this analysis we will skip creating a group to simplify the analysis.

If we wanted a group for the beam element we could create a single group with our single element and two nodes. We are not recovering any data from the Mass Element, so we can leave it out of the group.



7.8 CREATING AN ANALYSIS SET – SIMPLE PSD

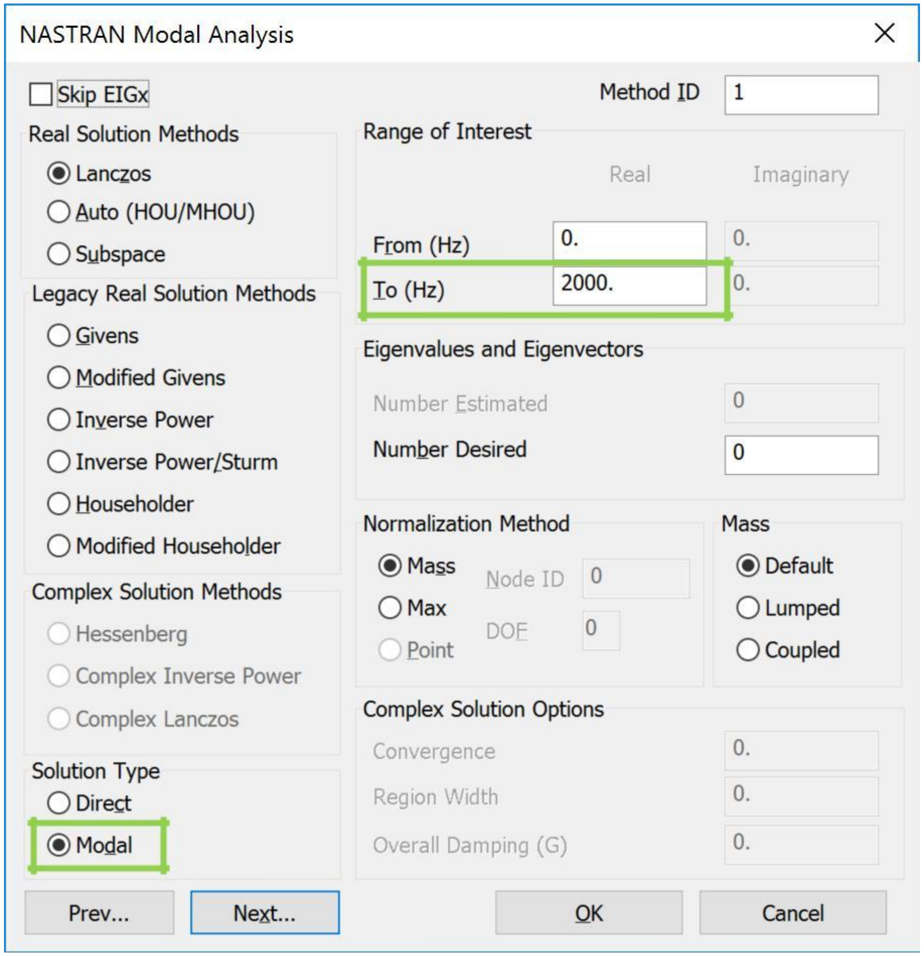
Next up is creating an analysis set. There are a lot of options to tailor the output to exactly what you need, but let's look at a straightforward analysis. This will allow you to see RMS Stress and positive crossings, which is enough information for a general PSD stress analysis and fatigue life estimate.



First, create a new Random Response Analysis Set.

Select **Next…**

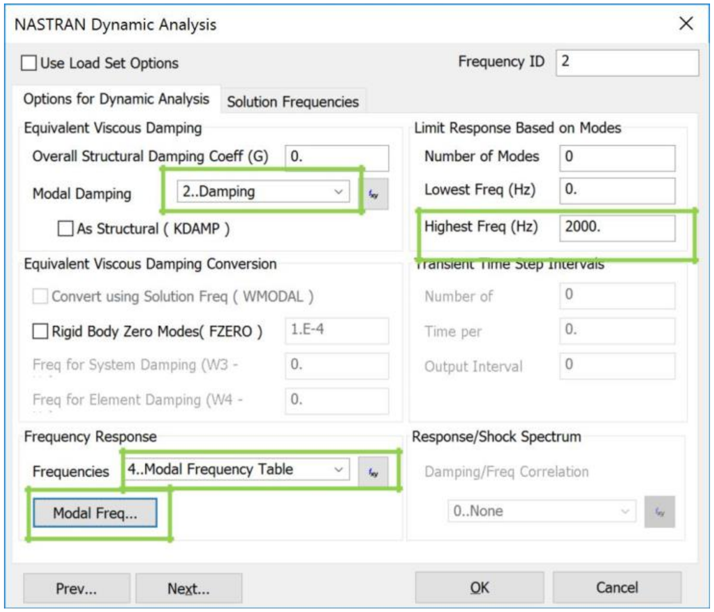
Keep pressing **Next…** until you arrive at the *NASTRAN Modal Analysis* window.



In the modal analysis tab you can decide between a Direct or Modal Solution Type. For this analysis, we will use a Modal solution. For more information about the difference in solution types take a look at the NX Nastran Basic Dynamic Analysis User Guide, Chapter 6.4 Modal Versus Direct Frequency Response.

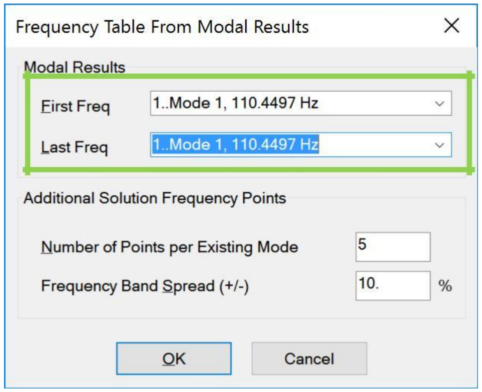
For Range of Interest you can set the maximum frequency at your upper limit of the PSD spectrum. This will guarantee your entire PSD spectrum is covered and not spend extra computing power (and time) processing frequencies above that.

In the Dynamic Analysis tab, one can specify the damping function and define the frequency range of the analysis (# of modes, or Lowest and Highest Frequency).

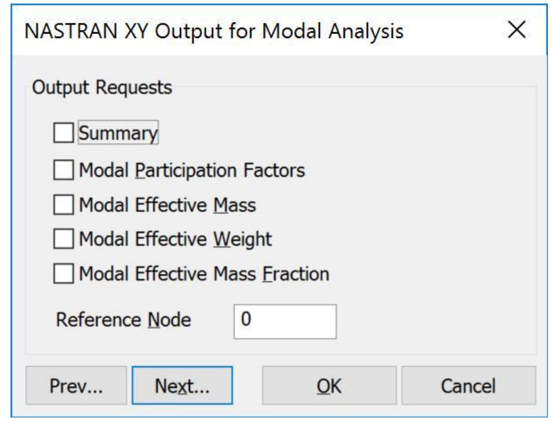


For Frequency Response, Select the "**Modal Freq…**" button, and then choose the modes you would like to create a Modal Frequency Table from. For this analysis only the first mode will be selected to match up with the analytical solution.

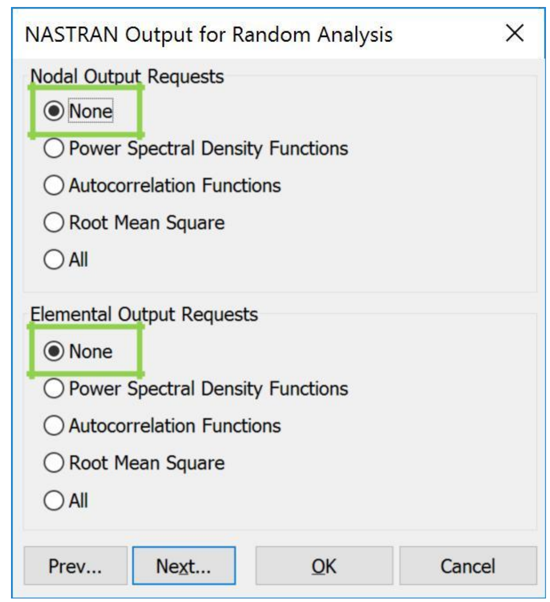
It is recommended to use the default values for the Points per Mode and Frequency Spread. See appendix for details.



In the *NASTRAN XY Output for Modal Analysis* window, you can leave all of the options un-checked. This information can be gathered when you run a standard modal analysis so there is no need to request it here.



In the *NASTRAN Output for Random Analysis* window, select none for nodal and elemental output.



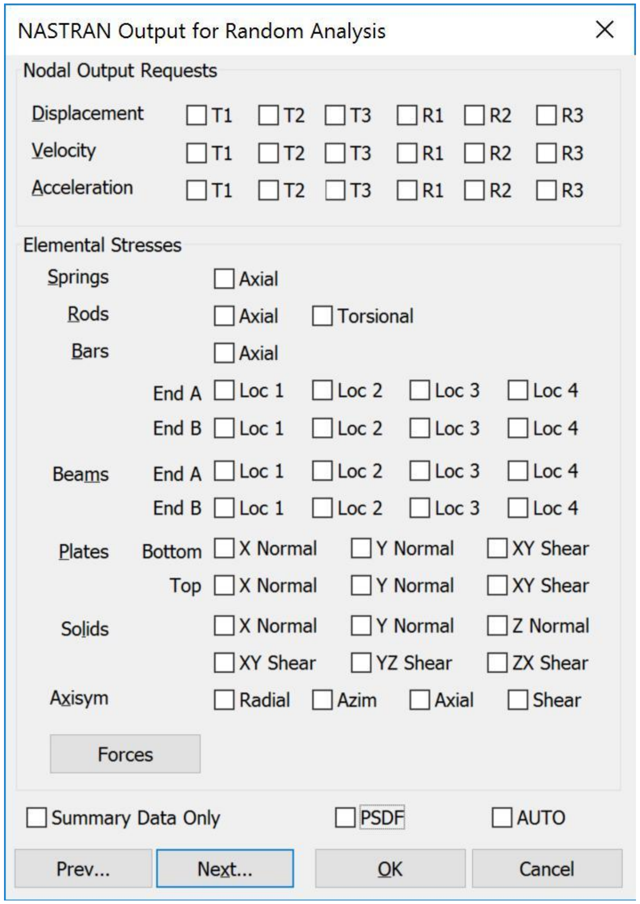
PSD Functions: Generates 'PSDF' output set for each frequency in the Modal Frequency Table

Autocorrelation Functions: Creates output for the autocorrelation functions if applicable

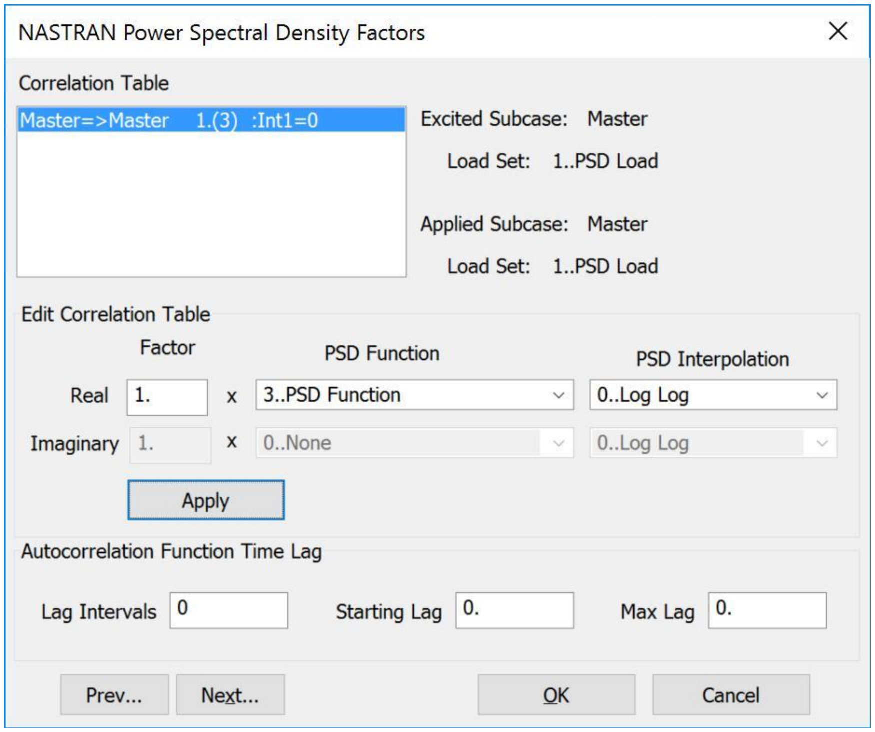
Root Mean Square: Generates 'CRMS' results for each frequency in the Modal Frequency Table

If you are interested in getting data for your entire structure, deselect everything in the *NASTRAN Output for Random Analysis* window. This will give you 1-σ stress results for your full model. For this example deselect all.

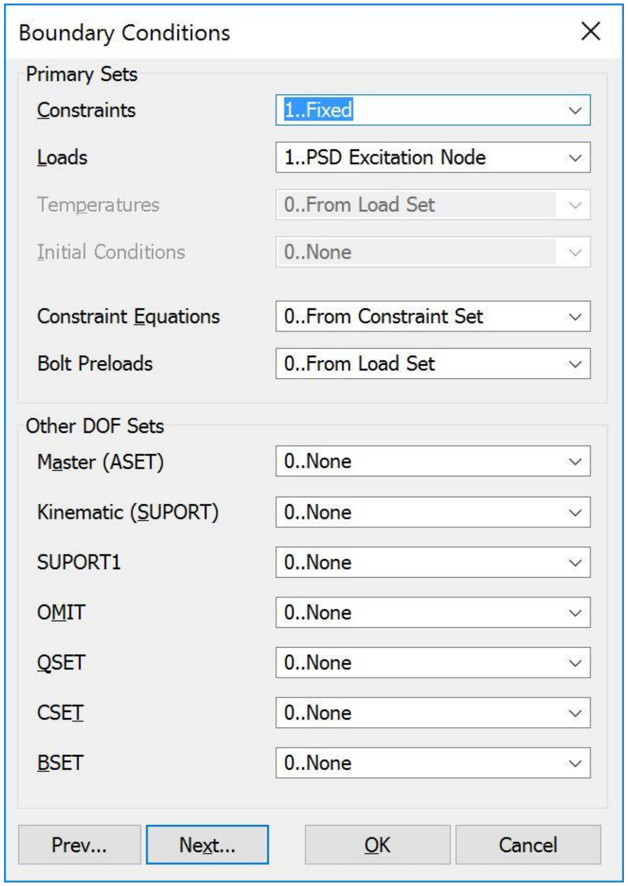
If you have an extremely large model and you only want specific nodal outputs, or results from certain elements, this is where you specify that. You can also use this window to request specific data such as T2 acceleration for a group of elements and nodes that you could have created in Section 5.7. If you select PSDF it will generate a function with the acceleration vs frequency for a group.



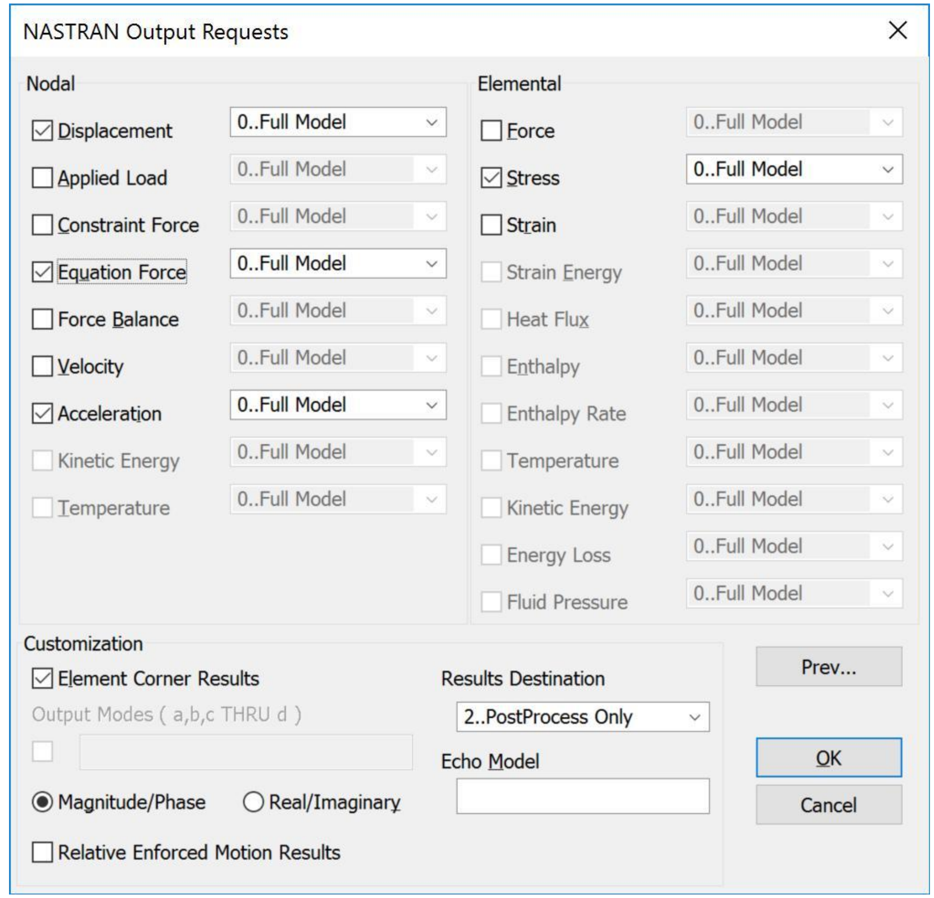
Select your PSD Function and be sure to select Apply. If desired you can scale the PSD function in the “Factor” input here.



Choose your constraint set and load created for the PSD analysis

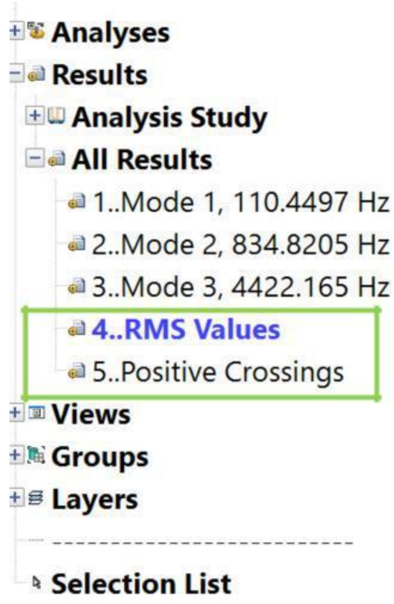


Choose the output requests desired. For this analysis we will request Displacements, Equation Force, Acceleration, and Stress.

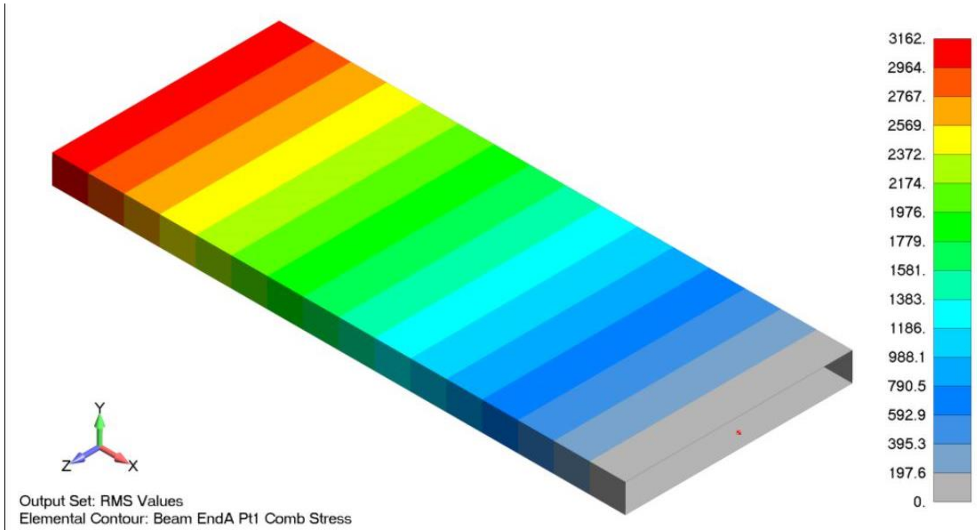


7.9 INTERPRETING THE OUTPUT

The PSD output sets are titled RMS Values and Positive Crossings. RMS Values will give all of the traditional stress, displacement, and acceleration data. Positive Crossings will output the frequency of positive crossings for each of the requested output vectors. This frequency is utilized to calculate fatigue damage based on the duration of excitation.

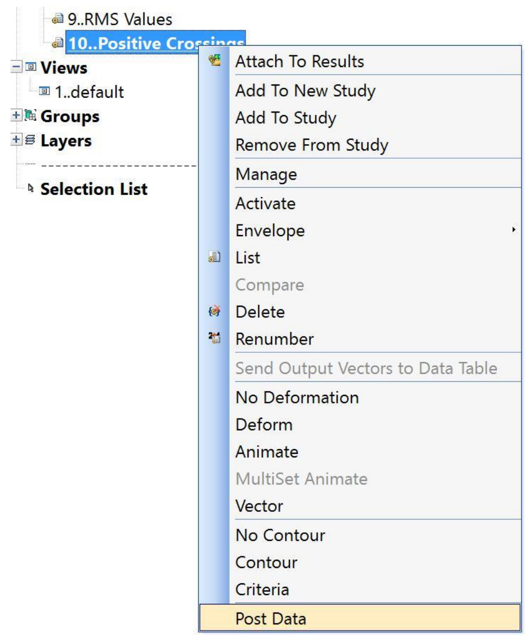


In the RMS Values output set you can contour all the usual output vectors. Beam EndA Pt1 Comb Stress is shown contoured over the beam. This output shown is the RMS Stress, and is also known as the 1-σ PSD stress value. This represents how much stress the beam will experience 68.3% of the time.



7.10 POSITIVE CROSSINGS

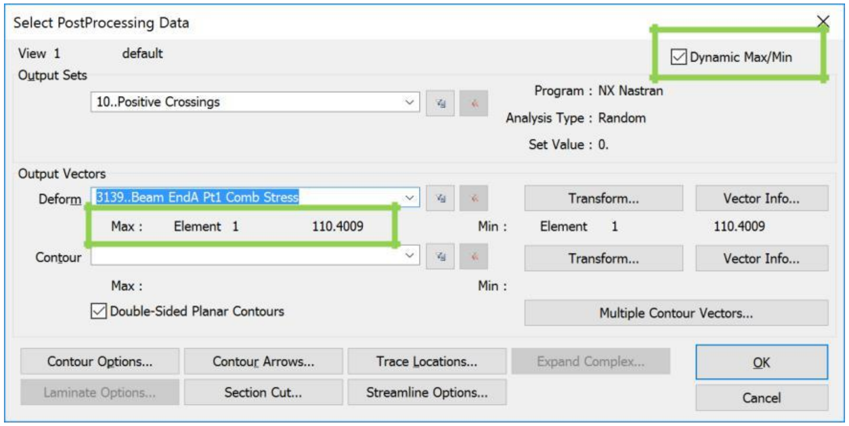
This is a vibration analysis, so of course we are also concerned about fatigue. We will use the output from positive crossings to calculate the fatigue life.



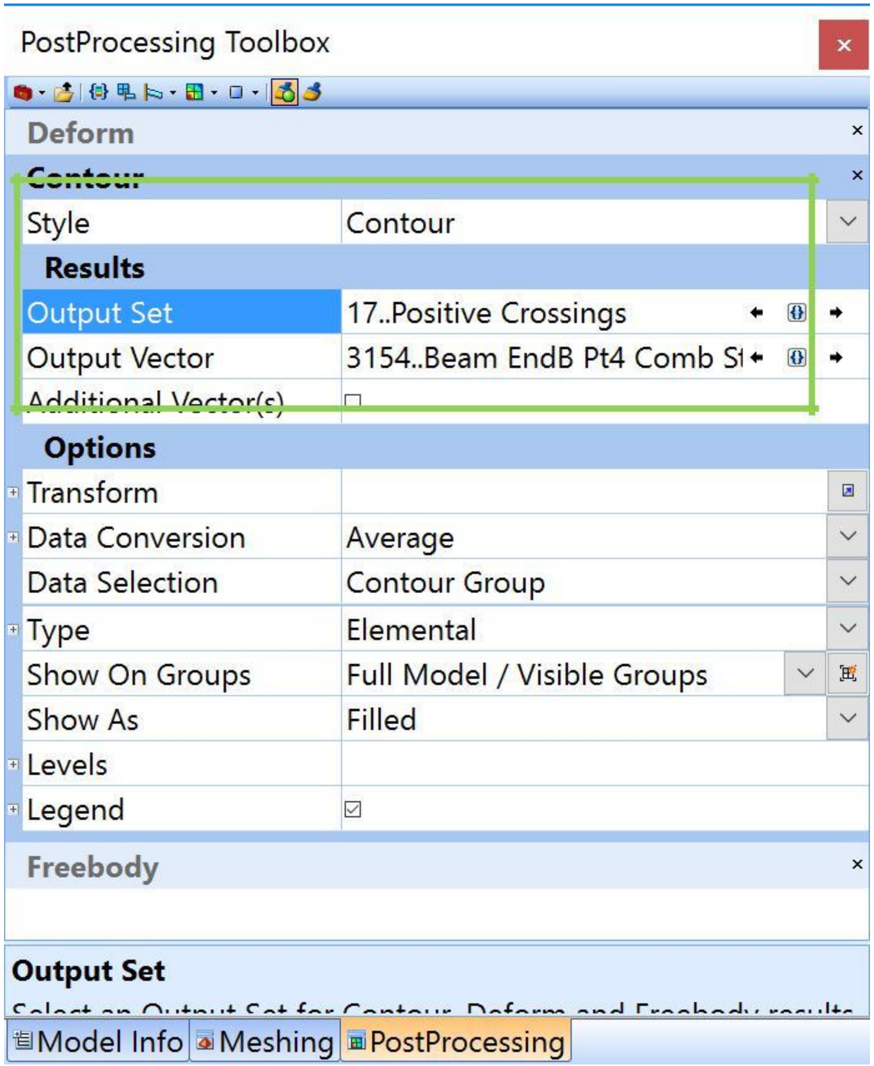
To access data for the positive crossings, Right click on the Positive Crossings result in the model info tree, and select "Post Data"

In the Post Data toolbar select the Dynamic Max/Min box in the upper right

Select the output vector for the positive crossing frequencies desired. In this model, all stress recovery points on Beam EndA show the same frequency.



Positive Crossings can also be contoured over the model. This can help the user understand how the positive crossing frequency changes throughout the model.



7.11 FATIGUE ANALYSIS USING RMS STRESS AND POSITIVE CROSSINGS

We can see that Beam **EndA Pt1 Comb Stress** vector gives a positive crossing frequency of 110.4 Hz. This means that given the white noise PSD input of 0.2 G2/Hz, the beam will experience a fully reversible stress of 3,162 psi at a frequency of 110.4 Hz.

Statistically speaking, this stress value represents the 1-σ value and will be experienced 68.3% of the time. A 2-σ stress of 2*3,162 or 6,324 psi will be experienced 27.1% of the time and a 3-σ value of 9,486 psi will be experienced 4.33% of the time. These values represent 99.73% of the stresses the beam will see at point A. It is probable that the beam will see stresses at and above the 4σ level, but this will only happen 0.27% of the time, so we will ignore them.

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

All three σ level stresses fall into the “run-out” range on a fatigue curve for aluminum. To demonstrate how to treat the problem if this is not the case, let us assume that there is a small hole in the beam which causes a stress concentration factor of 3. This would put the 1-σ stress level at 9,486 psi. We can use Miner’s cumulative damage index to get a sense of how long the beam will last under this condition. Miner’s cumulative damage is given by the equation on the right.

7.12 FATIGUE ANALYSIS – TIME TO FAILURE

On the right is a table containing values taken from a fatigue curve for aluminum. For a given stress, the amount of cycles needed to cause failure is given.

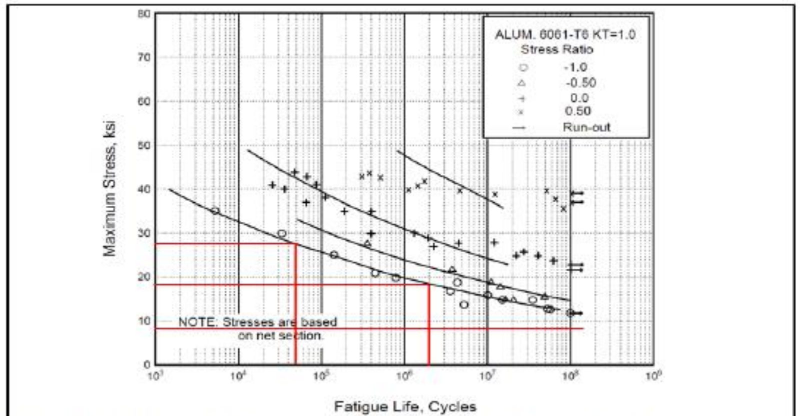


Figure 3.6.2.2.8. Best-fit S/N curves for unnotched 6061-T6 aluminum alloy, various wrought products, longitudinal direction.

Point A	1σ	2σ	3σ
Stress	9,486 psi	18,972 psi	28,458 psi
# of Cycles to Fail	infinite	11.0E6 cycles	14.0E4 cycles

These values can be substituted into Miner's equation to calculate how many cycles can occur until the beam fails. Substituting in the values and solving for n, yields a beam life of 1.80E6 cycles. If the beam is vibrating at a frequency (number of positive crossings) of 110.4 Hz, then it will take the beam approximately 16,300 seconds or about 4.5 hours to fail.

$$1 = \frac{0.6831 \cdot n}{\infty} + \frac{0.271 \cdot n}{11.0E5} + \frac{0.0433 \cdot n}{14.0E4}$$

As long as the beam is exposed to the while noise vibration for less than 4.5 hours, it should not fail.

8. EXAMPLE 3: SOLID MESHED BEAM

Let's take a look at the same beam geometry modeled with solid elements. The beam is massless, with a point mass of 0.5lbf (1.30e-3 snails) attached via RBE2 on the end.

The beam properties are shown below:

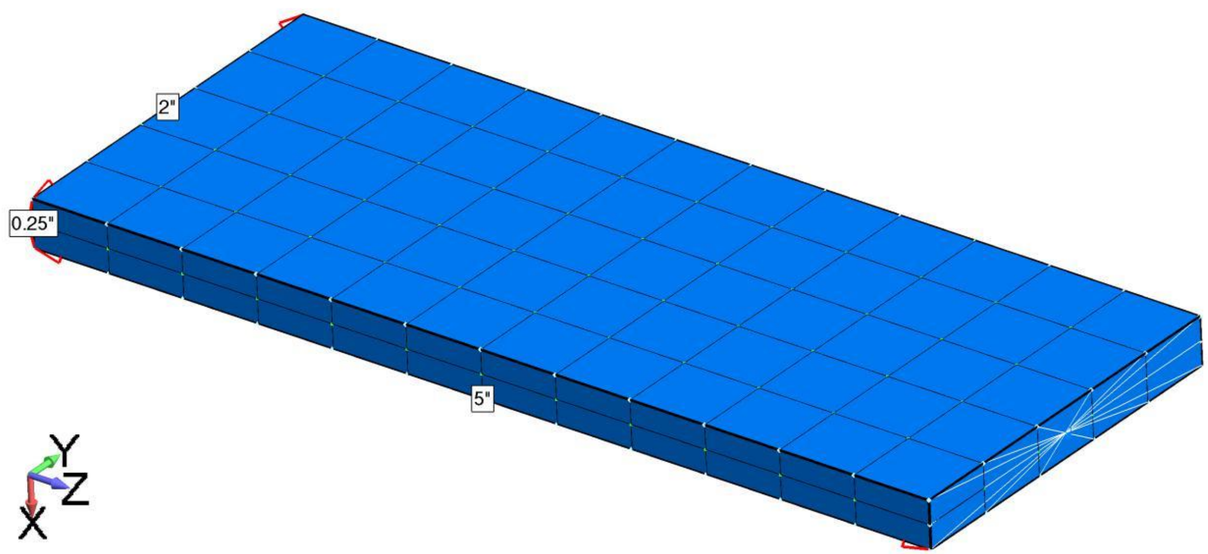
w = 2 in

T = 0.25 in

Lbeam = 5 in

W = 0.5 lbf

E = 10e6 psi



8.1 ANALYTICAL SOLUTION

Let's first take a look at the hand calculations to show how the beam is expected to behave.

First up is maximum deflection Y_{max}

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

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

$$f_n = \sqrt{\frac{1}{2\pi}\left(\frac{g}{Y_{max}}\right)} = 110.6\ Hz \qquad\qquad Q = 2\sqrt{f_n} \approx 21$$

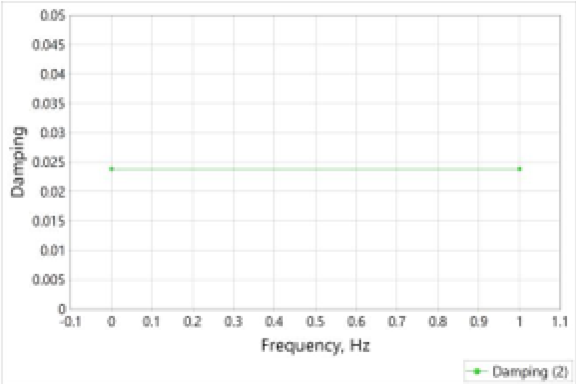
Utilizing Miles' Equation to estimate Grms we see that Grms is approximately 27 Gs:

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

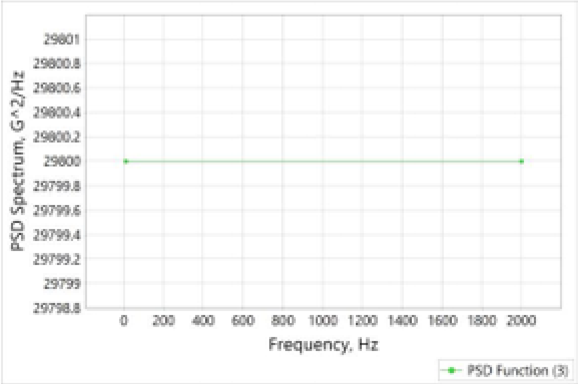
8.2 PSD FUNCTION INPUT

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

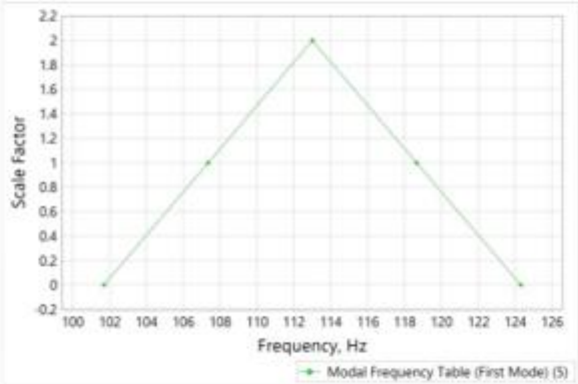
Damping Function



PSD Spectrum

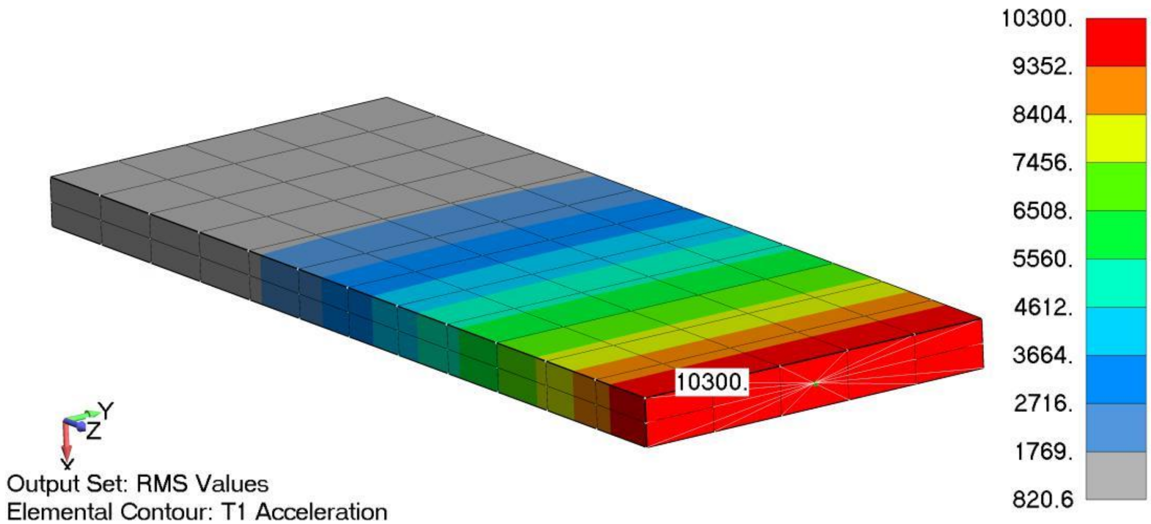


Modal Frequency Table



8.3 PSD STRESS RESULTS

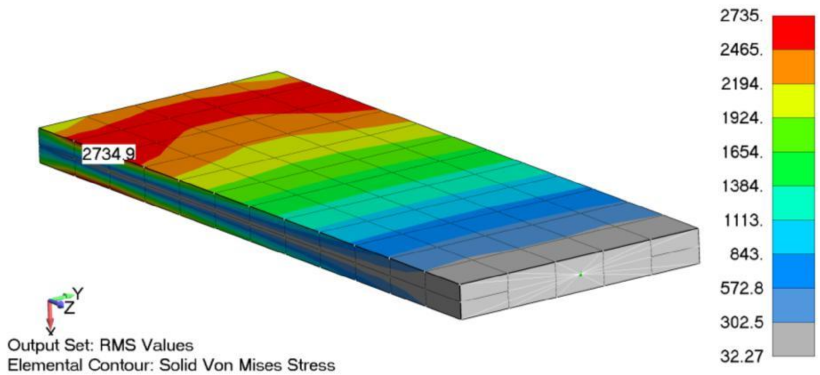
After running the analysis, let’s take a look at the results. The PSD results can be validated by checking the resultant acceleration against the Miles’ equation prediction. Miles’ equation predicted 27 G’s for the maximum acceleration. The results show an acceleration of 10,300 in/s2 which matches up with the Miles’ equation prediction.

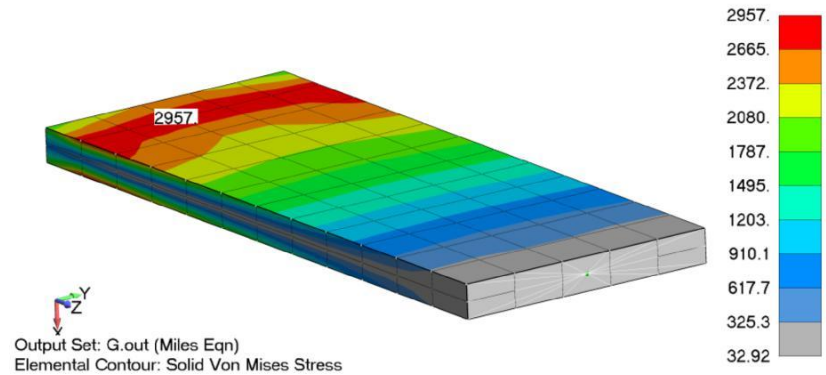


$$\frac{10{,}300\ \frac{in}{s^2}}{386.09\ \frac{in}{s^2}} = 27\ Gs$$

8.4 COMPARING MILES' APPROXIMATION AND PSD RESULTS

An additional verification is done by comparing the PSD stress results to the static analysis with the acceleration given by Miles' equation. The images below show an 8% difference between the two results, with similar stress patterns. In addition, the hand calculations show ~10% higher stresses than the static analysis.





Hand Calculations:

$$F_d = 27 * W * S_a = 13.5\ lbf$$

$$Stress = \frac{Mc}{I} = \frac{(F_d L)(\frac{T}{2})}{I} = 3{,}240\ psi$$

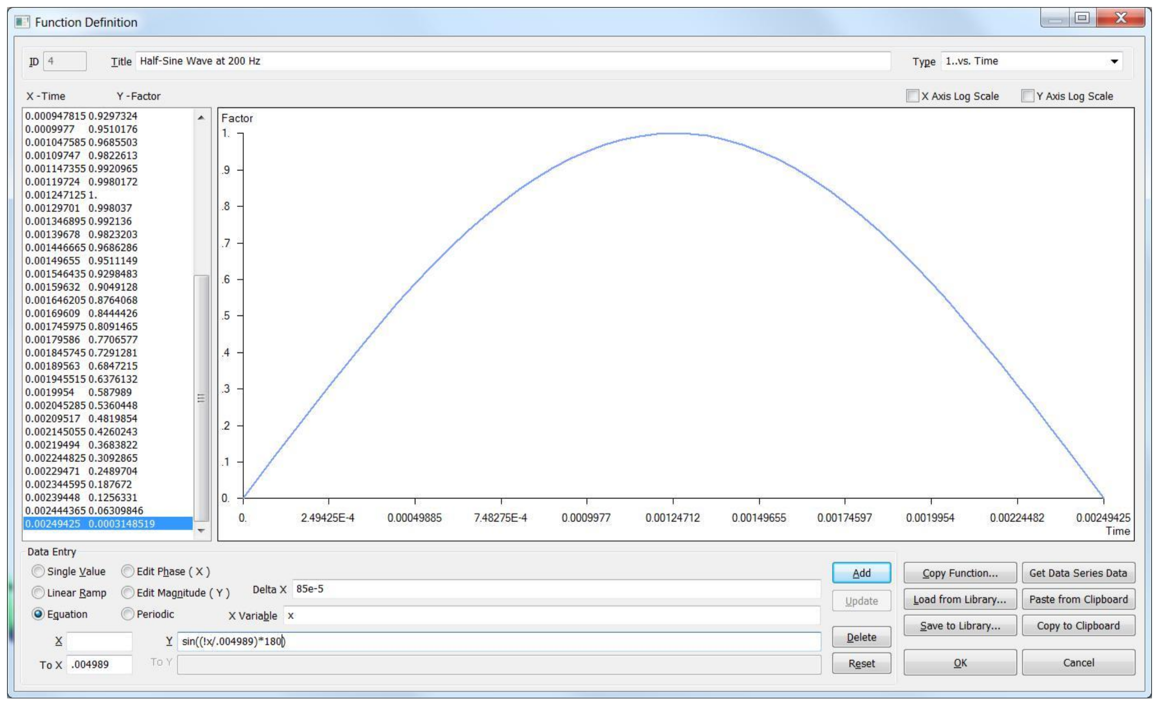
This comparison between the PSD results, Miles' equation, and hand calculations offer some insight into the relative accuracy of the analysis.

9. DIRECT TRANSIENT ANALYSIS

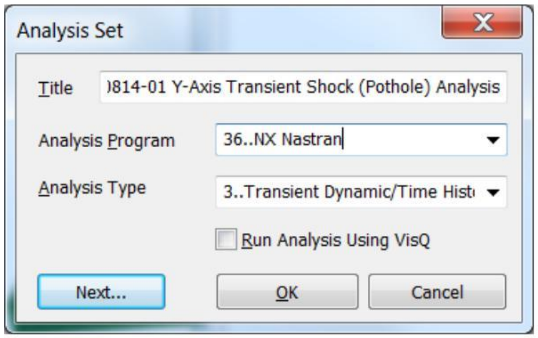
Sometimes you just want to whack the structure and not mess around. In this scenario, we are going to hit the circuit board with a 100 g pulse at a frequency of 200 Hz in the Y-direction (one can detect a theme to this seminar?). The procedure just requires a function for the hit and then a few setup screens. The equation of motion is even simpler:

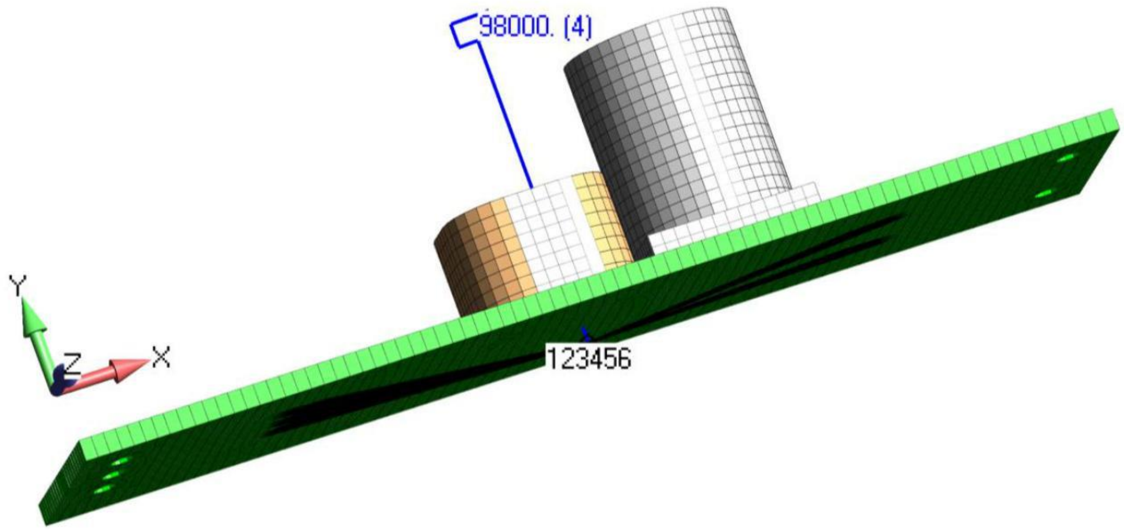
$$F_o(t) = m\frac{\partial^2 u}{\partial t^2} + c\frac{\partial u}{\partial t} + ku$$

Our equation is developed in FEMAP using a sin((!x/0.004988)*180) to create a 200.4 Hz half-sine wave:

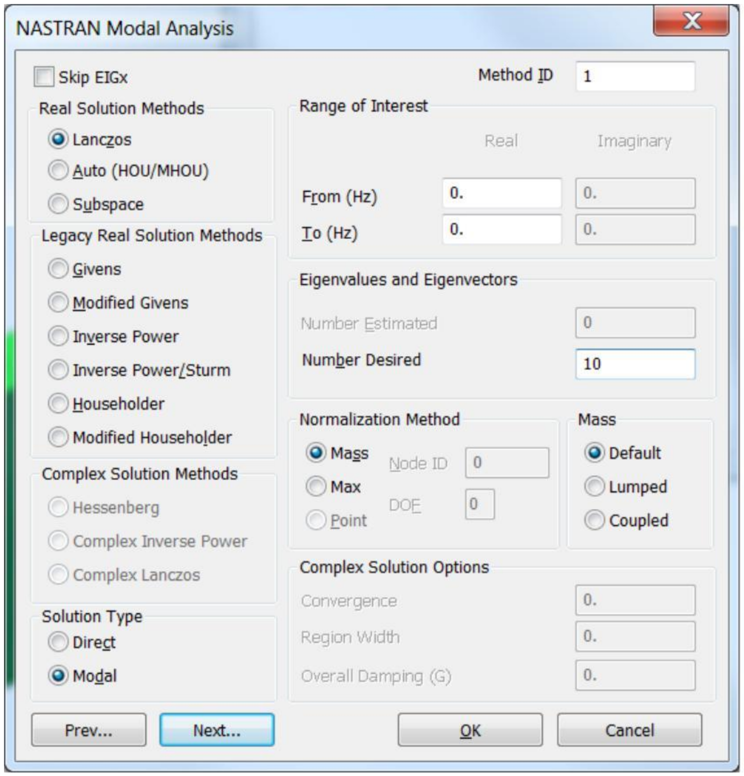


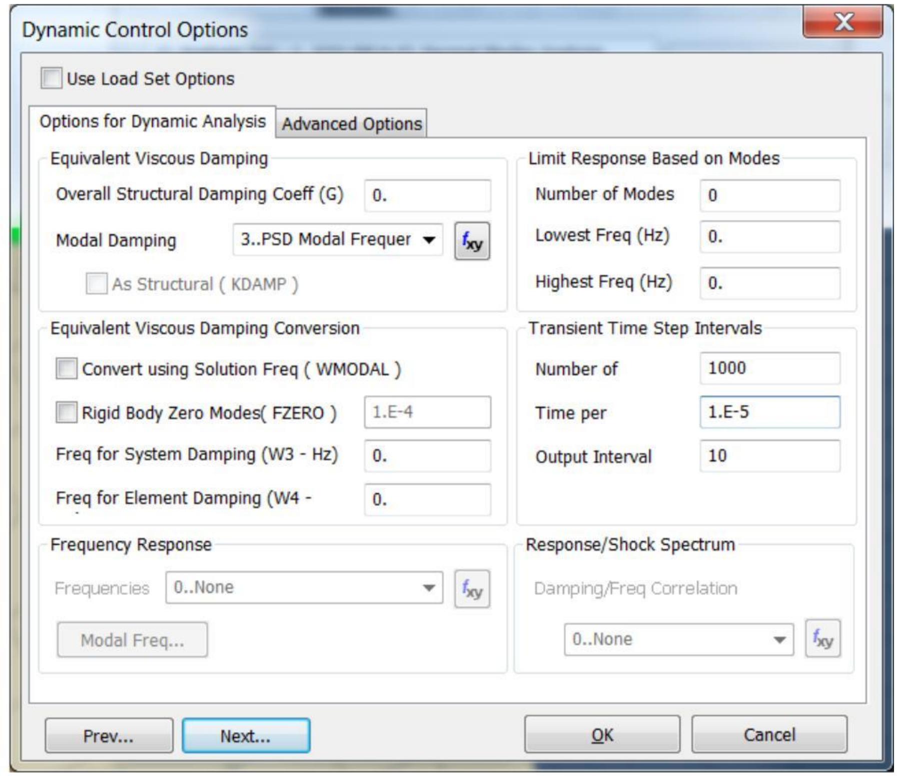
For our work, we are just going to use half the sin wave to give the system a shock pulse. The load for this analysis is 100 g (98,000 mm/s2) with our half-sine function at 200.4 Hz.



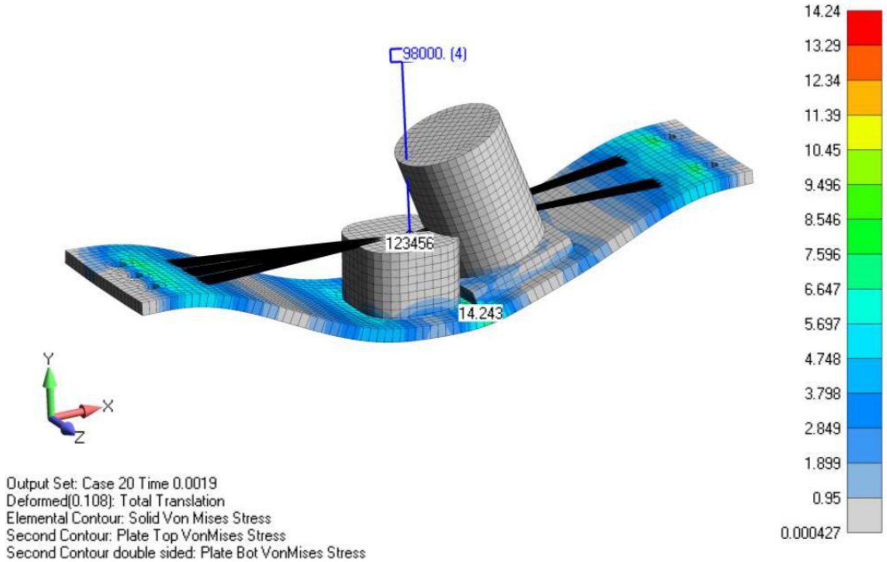


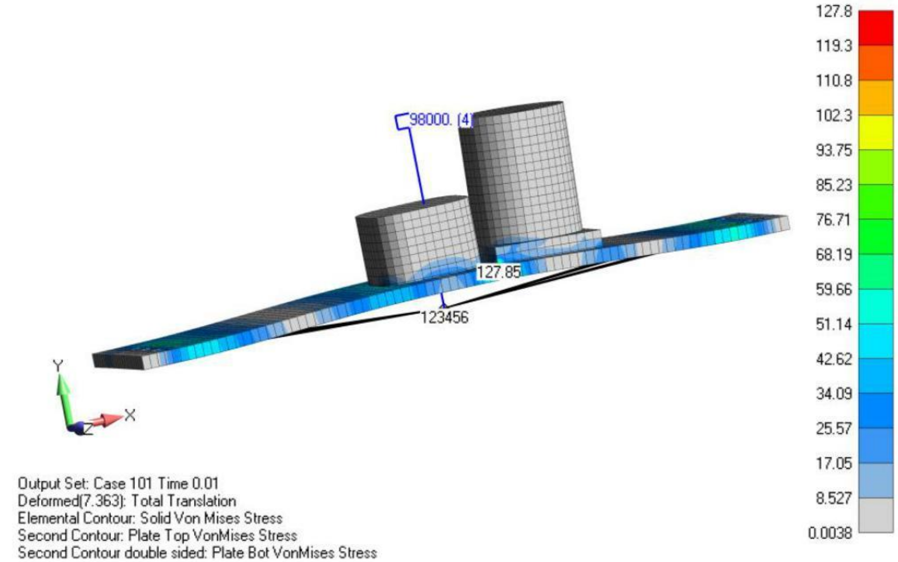
Our transient analysis is based on the first ten Eigenvalues and Eigenvectors.



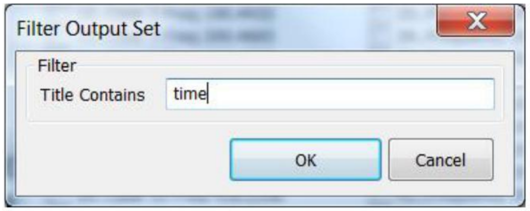


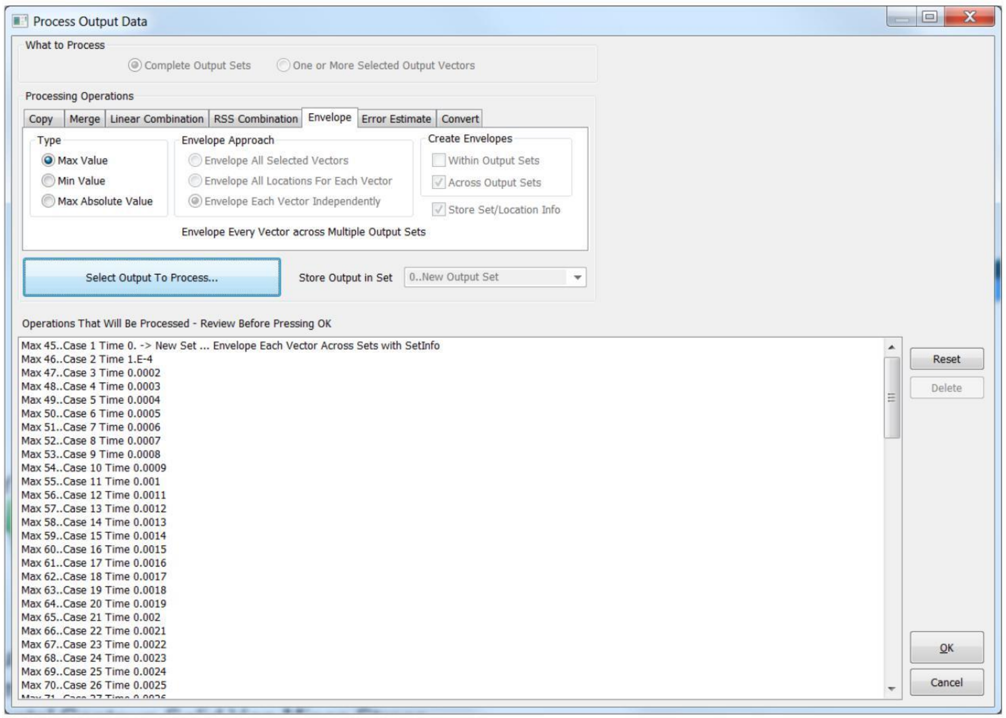
At the end of the simulation, one has a hundred result sets to claw through.



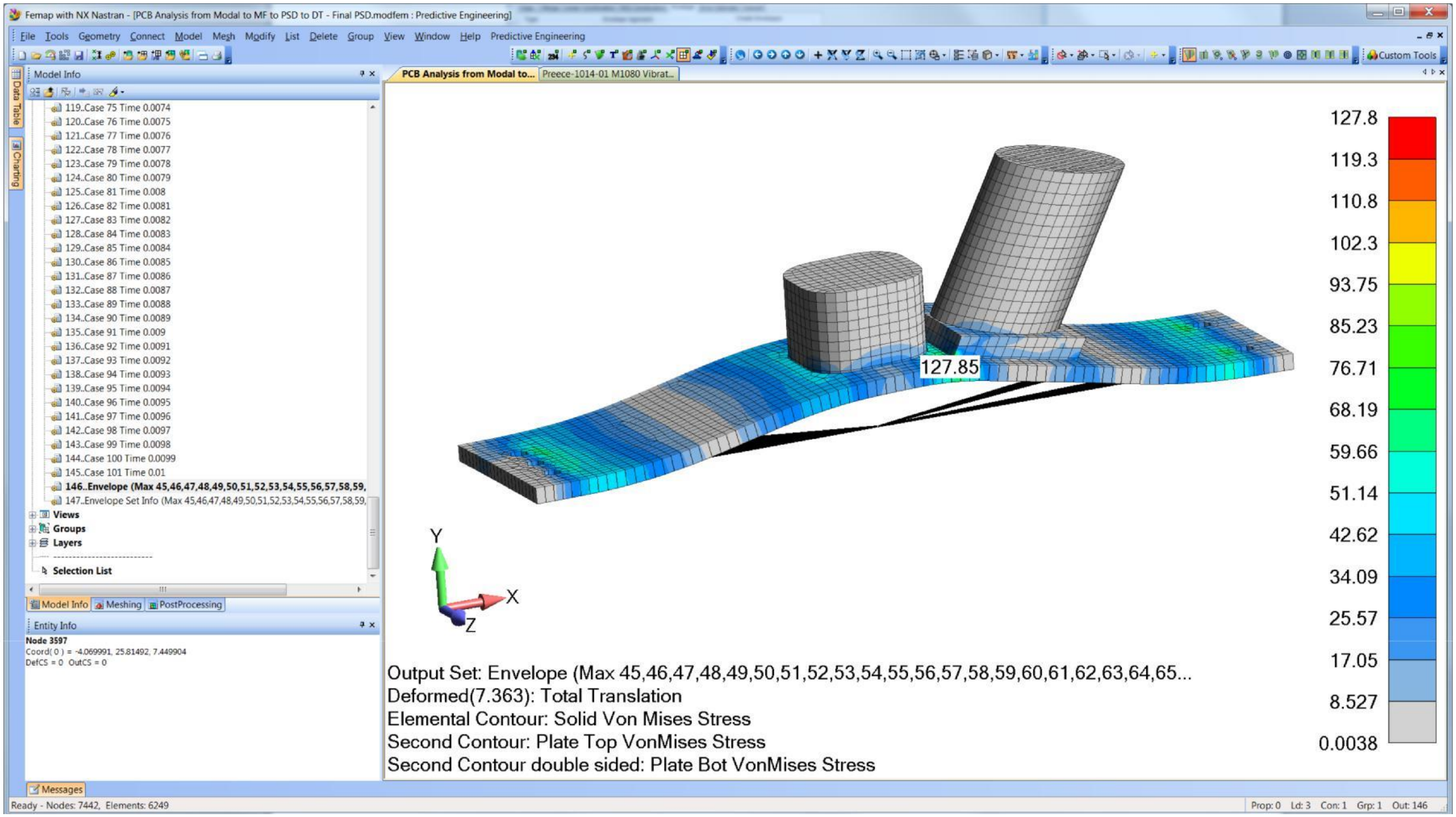


A much simpler way to process transient results is to use the FEMAP envelope function and then select all the output sets with "time" in the title:





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



10. 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.

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

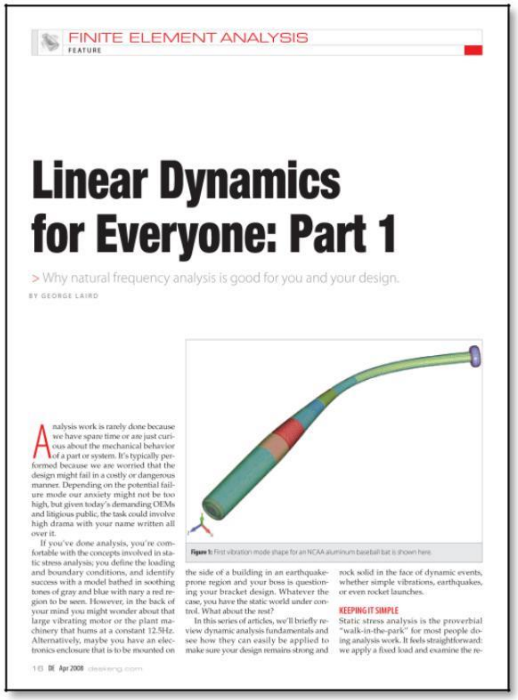
11.1 CHECK FO6 FOR MASS SUMMATION AND KNOW WHAT YOU KNOW

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 “OUTPUT FROM GRID POINT WEIGHT GENERATOR”
- Error and Warning Messages?

Vibration Analysis White Paper



FINITE ELEMENT ANALYSIS
FEATURE

Linear Dynamics for Everyone: Part 1

> Why natural frequency analysis is good for you and your design.

BY GEORGE LAIRD

Analysis work is rarely done because we have spare time or are just curious about the mechanical behavior of a part or system. It's typically performed because we are worried that the design might fail in a costly or dangerous manner. Depending on the potential failure mode our anxiety might not be too high, but given today's demanding OEMs and litigious public, the task could involve high drama with your name written all over it.

If you've done analysis, you're comfortable with the concepts involved in static stress analysis; you define the loading and boundary conditions, and identify success with a model bathed in soothing tones of gray and blue with nary a red region to be seen. However, in the back of your mind you might wonder about that large vibrating motor or the plant machinery that hums at a constant 12.5Hz. Alternatively, maybe you have an electronics enclosure that is to be mounted on the side of a building in an earthquake-prone region and your boss is questioning your bracket design. Whatever the case, you have the static world under control. What about the rest?

In this series of articles, we'll briefly review dynamic analysis fundamentals and see how they can easily be applied to make sure your design remains strong and rock solid in the face of dynamic events, whether simple vibrations, earthquakes, or even rocket launches.

Figure 1: First vibration mode shape for an NCAA aluminum baseball bat is shown here.

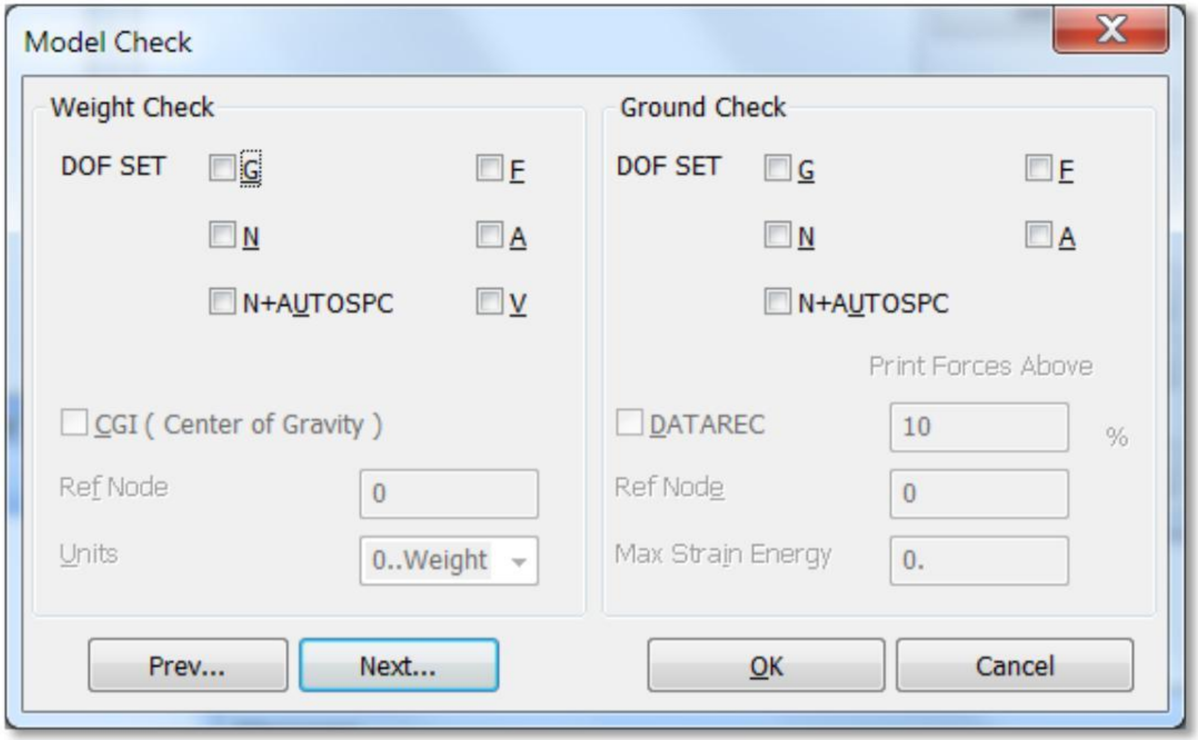
KEEPING IT SIMPLE

Static stress analysis is the proverbial “walk-in-the-park” for most people doing analysis work. It feels straightforward: we apply a fixed load and examine the re-

16 DE Apr 2008 deskeng.com

11.2 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.



For more information see our User Guide: What is Groundcheck?

12. RANDOM VIBRATION CONCLUSION

The topic of Random Vibration is complex. What is presented here is a brief introduction to the theory and implementation of the subject. It is suggested that the user read a bit of the documentation provided on this subject within the NX Nastran library that is installed with every license of FEMAP & NX Nastran.

For a lot of FEA work, a straightforward recipe to accomplish your analysis task is seldom available and if it does, could easily lead you down the wrong path. Thus, I'm fond of saying that nothing beats having a good theoretical understanding of what you are doing and being highly suspicious of any result generated in "color". Or as I have read "Computer models are to be used but not necessarily believed."

13. INTRODUCTION TO RESPONSE SPECTRUM ANALYSIS

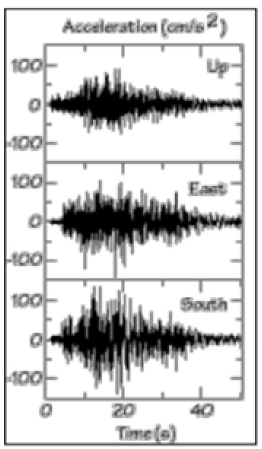
Response spectrum analysis is widely used for the design and assessment of structures that are subject to earthquakes or shock events. The reason we want to use a response spectrum is that it allows us to analyze transient events (time based events) without having to review hundreds or thousands of results sets. In essence, it allows us to assess the maximum dynamic response (stress, acceleration, velocity or displacement) of a structure using a very simple analysis technique (normal modes). Moreover all of this goodness can be had by only having to interrogate one (1) output set.

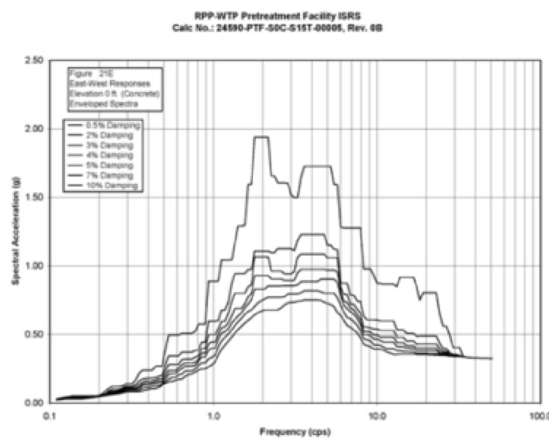
This tutorial will walk you through the theoretical background of response spectrum analysis and how to actually implement it within FEMAP & NX Nastran.

13.1 THE ACCELEROGRAM

The whole key to a response spectrum analysis is generating the response spectrum that is used to drive the FEA model. A response spectrum is the “load” to the FEA model that allows us to reduce a complicated transient analysis (time based) to simple normal modes analysis (frequency based). In general terms, response spectrums are generated from acceleration versus time measurements or accelerograms. An example of an accelerogram is shown below for an earthquake event. The three traces (Up, East and South) were generated by a tri-axial accelerometer mounted onto a concrete foundation. Similar traces can be obtained during shock events (e.g., rocket launches).

A response spectrum (an example is shown on the right) is created in a somewhat non-intuitive manner from an accelerogram. The conversion from time based data into a response spectrum is mathematically complex (Note: it is not a Fast Fourier Transform (FFT)). In the next section we will give an equation-less explanation of this mathematical procedure.



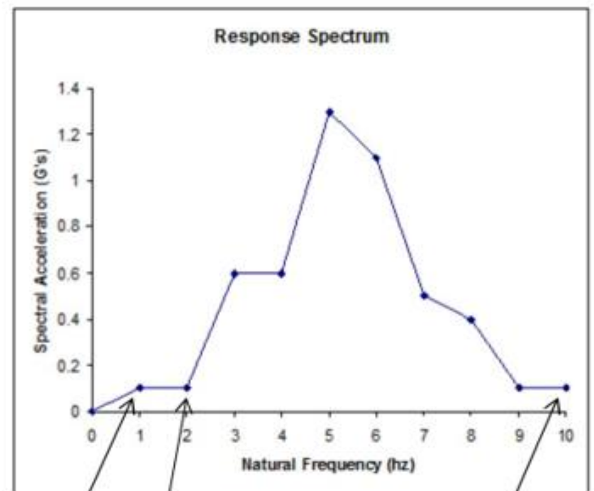


Accelerogram image courtesy of Charles Ammon, Penn State), http://earthquake.usgs.gov/learning/glossary.php?term=accelerogram

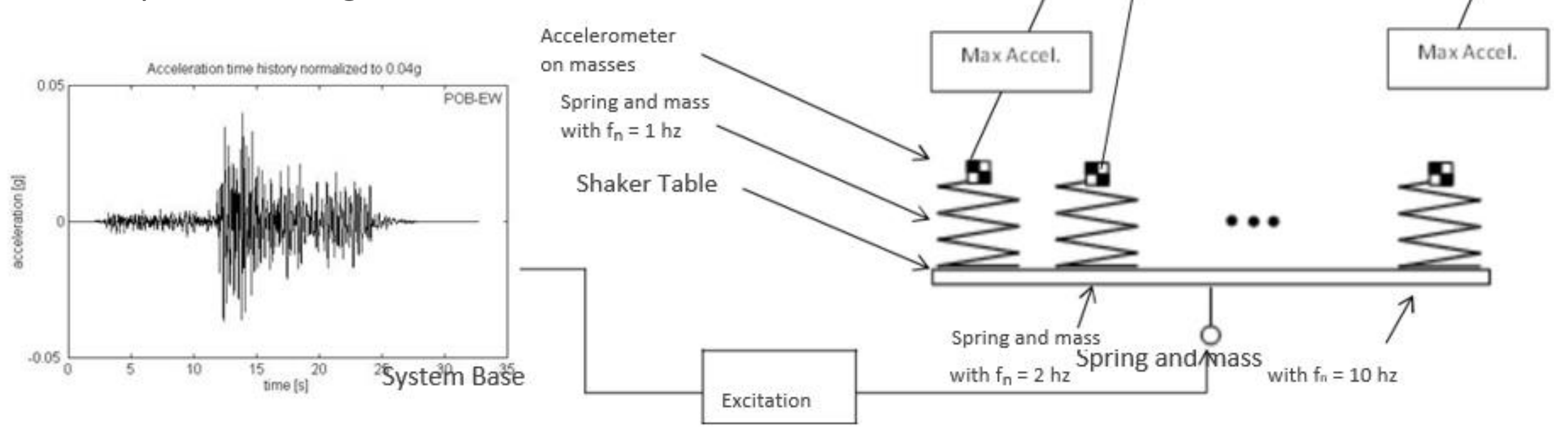
The response spectrum image is courtesy of the U.S. Taxpayers.

13.2 CREATING A RESPONSE SPECTRUM

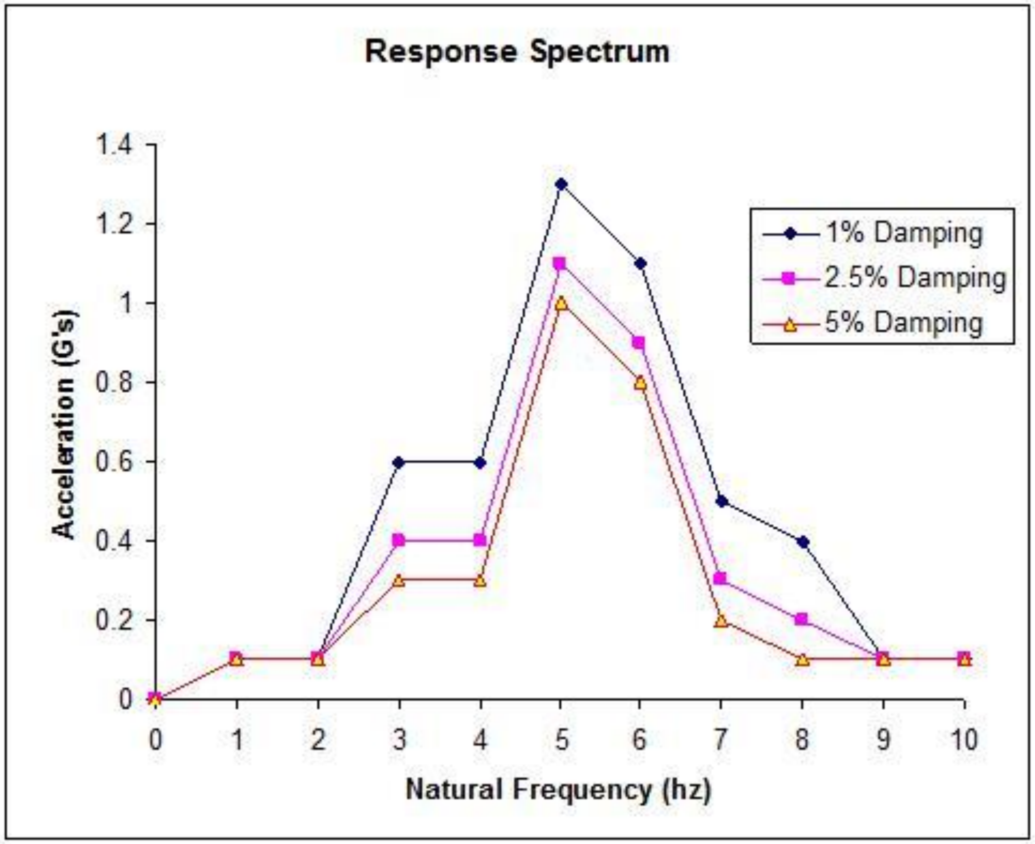
As we mentioned before, converting an accelerogram (Time domain) to a response spectrum (frequency domain) is not done with a FFT. Let's explore this procedure using a simple example. We want to create a response spectrum that goes from 0 to 10 Hz using our acceleration versus time history data (i.e., accelerogram).

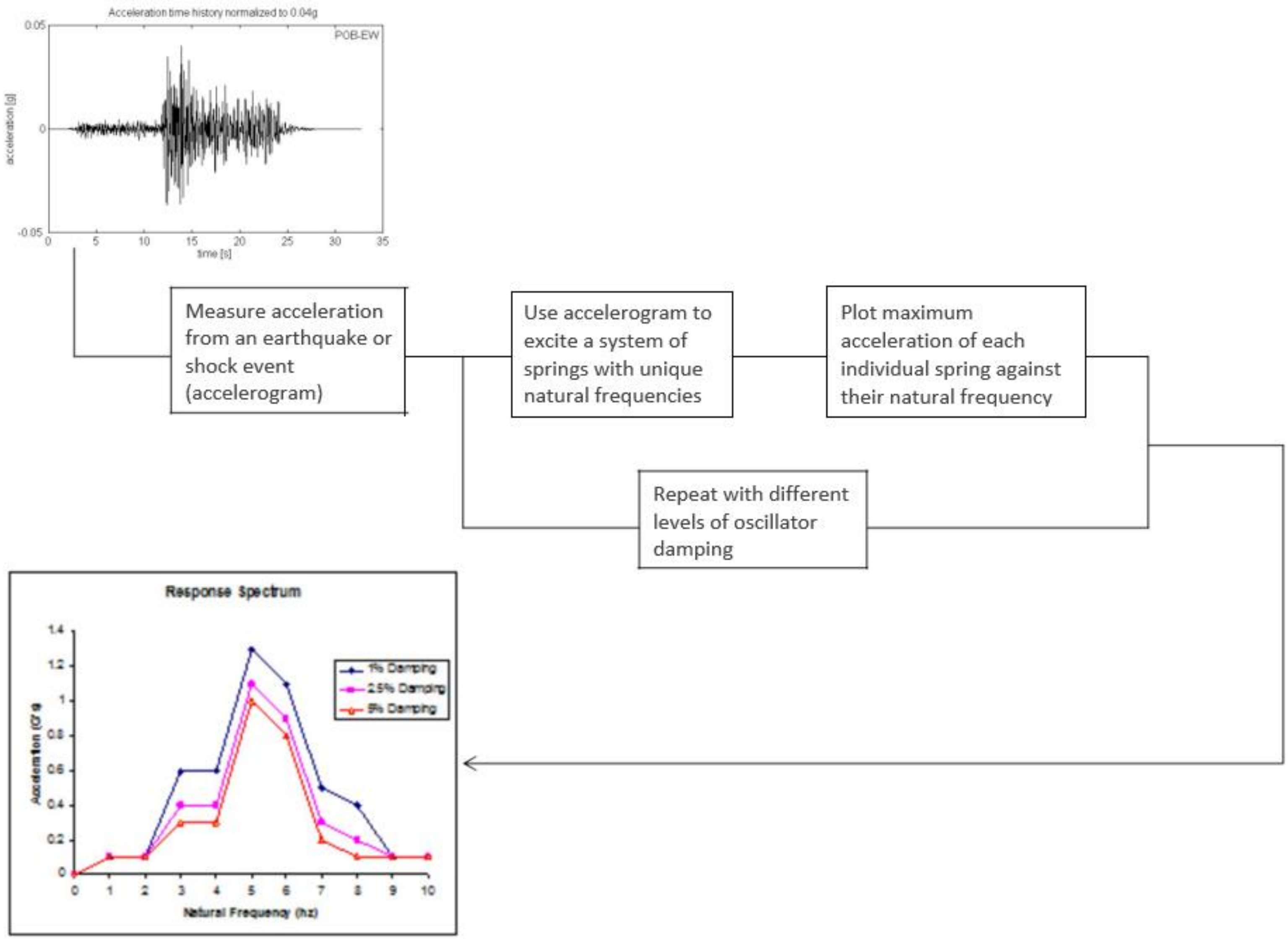


Think of a shaker table having ten little spring/mass units attached onto its surface. At the top of each spring is a mass block. To measure the acceleration of the spring/mass system we have fixed an accelerometer onto the block. Our spring/mass systems are then tuned to range in eigenvalues or natural modes from 1 to 10 Hz in 1 Hz increments. The shaker table is then driven with our accelerogram. The output from the ten accelerometers are normalized with respect to gravity giving us g's (dimensionless). Our final step is to plot these normalized values against their respective eigenvalues. The resulting graph is then called a "Response Spectrum". Of course, there exists a purely mathematical approach for turning accelerograms into response spectrums but the theory is a bit dry and left for graduate students in mechanics.



To account for system damping (which exists to some degree in all structures), we can attach dampers to our springs and rerun the shaker table experiment. As you might expect, another curve is generated that follows the exact same profile but with lower acceleration values. Below is a classic example of a family of response spectrum curves for 1%, 2.5% and 5% of critical damping.




Acceleration time history normalized to 0.04g
POB-EW
acceleration [g]
time [s]
Measure acceleration from an earthquake or shock event (accelerogram)
Use accelerogram to excite a system of springs with unique natural frequencies
Plot maximum acceleration of each individual spring against their natural frequency
Repeat with different levels of oscillator damping
Response Spectrum
1% Damping
2.5% Damping
5% Damping
Natural Frequency (hz)

13.3 NX NASTRAN METHOD

Nastran starts off the response spectrum analysis by calculating the normal modes of the system and then the modal participation factors (PF). We seek the solution to;

$$([K] - \lambda[\mathrm{M}])[\emptyset] = 0, \lambda = \omega^2$$

Which is an eigenvalue problem, the non-trivial solution to the problem is found from;

$$\det([K] - \lambda[\mathrm{M}]) = 0$$

This gives non-zero eigenvectors [Ø]≠0. The response spectrum is used in conjunction with the PF's and a modal combination method to determine the system response to the spectrum. The PF's are calculated by;

$$PF = [\emptyset]^T[M][I]$$

Where [Ø] is the mode shape vector, M is the mass matrix and I is a unit vector of the same dimensions as M.

What we are saying is that each normal mode shape has a bit of mass associated with it. To generate displacements and stresses within the structure, each mode shape (and its PF) is factored against the response spectrum at that particular frequency. That is to say, a standard linear stress analysis is performed at each normal mode submitted in the response spectrum analysis. After all of the requested mode shapes have been processed, the resulting displacements and stresses are summed up using some sort of combination method (see next slide) and then written out into one final result set for subsequent post-processing.

As we had mentioned, at the end of the analysis, the individual results set generated at each mode are summed up. An idea of some of the methods that are available:

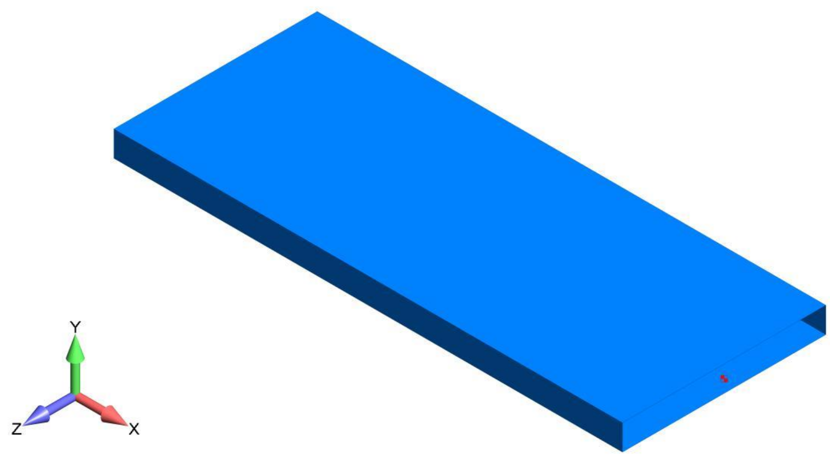
1. ABS – This combination method is the most conservative, the absolute value resulting from each mode are summed.
2. SRSS – Square root of the sum of the squares, also contains a provision to use the ABS method for mode combination that are within a user specified closeness (default is 1.0).
3. NRL – U.S. Navy Shock Design Modal Summation Convention, this method combines the SRSS and the ABS methods
4. NRLO – Updated NRL method to comply with NAVSEA-0908-LP-000-3010 specification.

The choice of the combination method is dependent on the analysis, the SRSS method is generally the method of choice since it includes the provision for using the ABS method for close modes.

14. RESPONSE SPECTRUM ANALYSIS EXAMPLE: CANTILEVER BEAM

Doing a Response Spectrum Analysis in FEMAP:

1. Creating the FE model
2. Defining the response spectrum or spectrums
3. Creating the interpolation table
4. Creating a modal damping function
5. Creating the excitation node and tying it into the model
6. Constraining the model
7. Setting up the analysis in the analysis manager
8. Post-Processing the results



14.1 PROBLEM DEFINITION

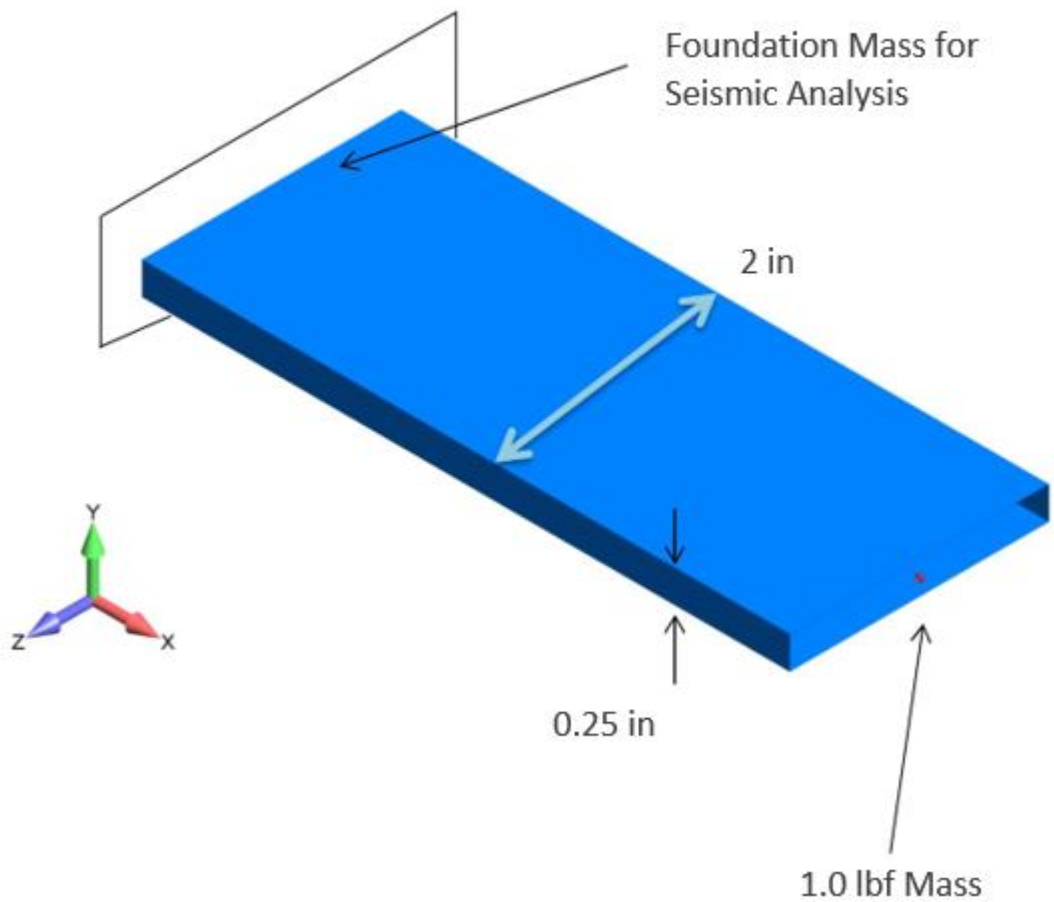
A cantilevered beam six inches in length is used to support a mass of 1.0 [lbs·s2/in]. Our objective is to determine the dynamic stresses of the beam for vibration along the vertical axis.

The FEA model is a single beam element. A picture of the beam element, with its cross section displayed is shown on the right.

We will compare the FEA results to an analytical solution.

Our unit system is lb/in/s and

1 g = 386 in/s2. The elastic modulus of steel is taken as 2.9E+07 psi.



14.2 ANALYTICAL SOLUTION

The stiffness of the cantilever beam can be determined from the Young's modulus for steel, the moment of inertia and the length of the beam:

$$k = \frac{3EI}{L^3} = \frac{3 * 2.9E7 * (\frac{2 * 0.25^3}{12})}{6^3} = 1048.9 \frac{lbs}{in}$$

With the stiffness and the weight, the circular natural frequency can be determined:

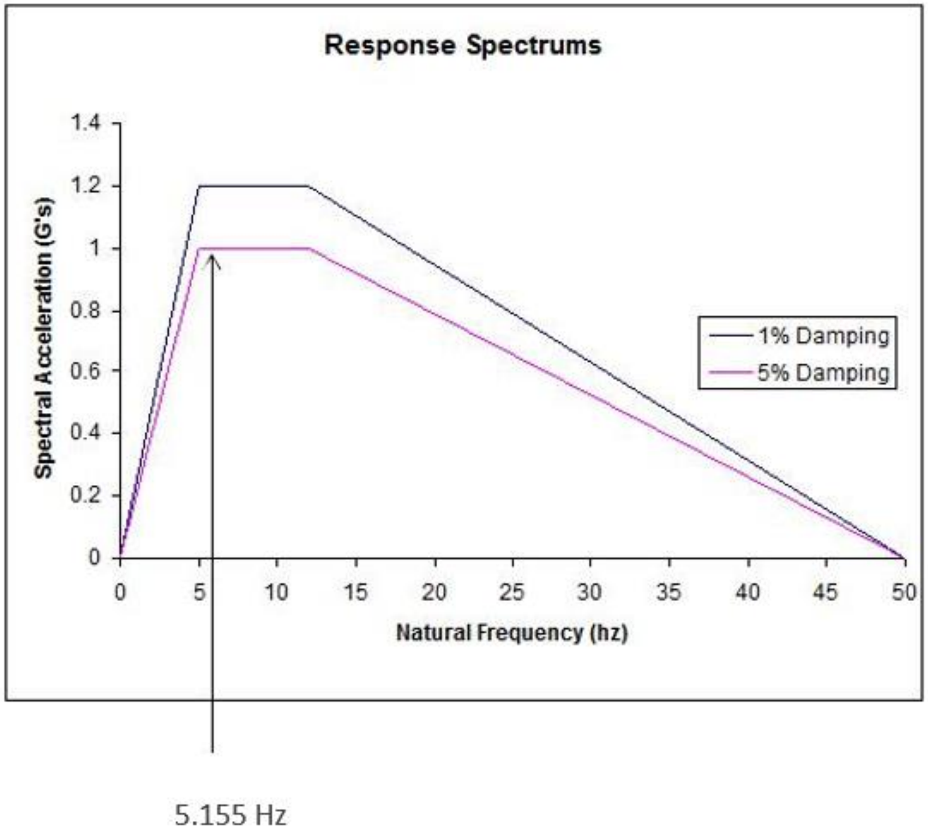
$$\omega_n = \sqrt{\frac{k}{m}} = \sqrt{\frac{1048.9}{1.0}} = 32.387 \ \frac{rad}{s}$$

And the natural frequency can be found:

$$f_n = \frac{\omega_n}{2\pi} = \frac{32.387}{2\pi} = 5.1545 \ Hz$$

We will determine the systems response to a simple response spectrum, to highlight the abilities of Nastran, we will use a damping value of 3%, this value is intermediary between the given response spectrum curves (just like a standard engineering problem, i.e., we never get exactly what we want for an analysis). This is no big deal since NX Nastran can interpolate between the given damping values.

Values for the problem have been chosen in order to provide a clean and straight forward solution. Since the beam model is a 1 degree of freedom system, we know that the modal participation factor (PF) for the above calculated natural frequency will be 1.0 (since all the mass of the system is at the only mode that the system has – never in real life but great for this example).



From the value that we previously determined from the natural frequency (5.155 Hz), using the response spectrum shown below, we can see that the response will be 1.2 and 1.0 G's. The system is linear, and as such linear interpolation can be used to determine the system response with 3% damping.

- Response with 1% damping (Sa) – 1.2 G's
- Response with 3% damping (Sa) – 1.1 G's (using linear interpolation) = 1.1*386 in/s2 = 424.6 in/s2
- Response with 5% damping (Sa) – 1 G

Knowing that the spectral acceleration (Sa) is 1.1 G's or 424.6 in/s (the response spectrum acceleration at the normal mode frequency), we can find the spectral velocity (maximum velocity at end of the beam) and the spectral displacement (the maximum displacement at the end of the beam).

$$S_v = \frac{S_a}{\omega_n} = \frac{424.6\frac{in}{s^2}}{32.38\frac{1}{s}} = 13.11\frac{in}{s}$$

$$S_d = \frac{S_a}{{\omega_n}^2} = \frac{424.6\frac{in}{s^2}}{32.38^2\frac{1}{s}} = 0.405\ in$$

The dynamic force can be determined from the weight at the end of the beam and the maximum acceleration (Samax). Then the dynamic stresses can be found;

$$F = ma$$

$$F_{dyn_{max}} = m * S_a = 1.0\frac{lbs * s^2}{in} * 424.6\frac{in}{s^2} = 424.6\ lbs$$

Finally the dynamic stress is given by;

$$\sigma = \frac{Mc}{I} = \frac{(424.6 * 6) * (\frac{0.25}{2})}{(\frac{2 * 0.25^3}{12})} = 122{,}285\ psi$$

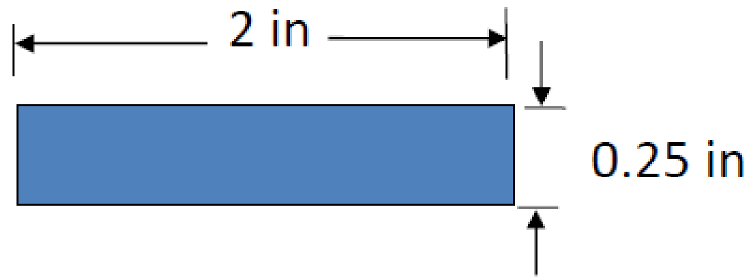
14.3 STEP 1: CREATING THE FE MODEL

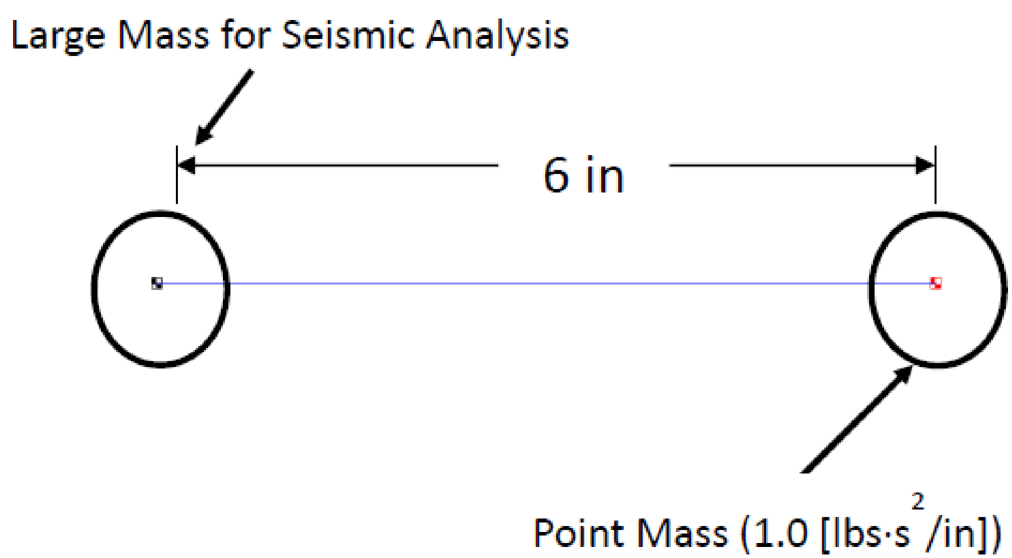
Create a beam element with the cross section as shown. The length of the beam is 6 in.

Add a point mass of 1.0 at one end of the beam. Add a large point mass of 10e6 to the end of the beam where the excitation will later be applied.

The cross section of the beam is chosen to be thin along the bending direction in order to minimize shear effects, this provides very close agreement between the FE results and the analytical results.

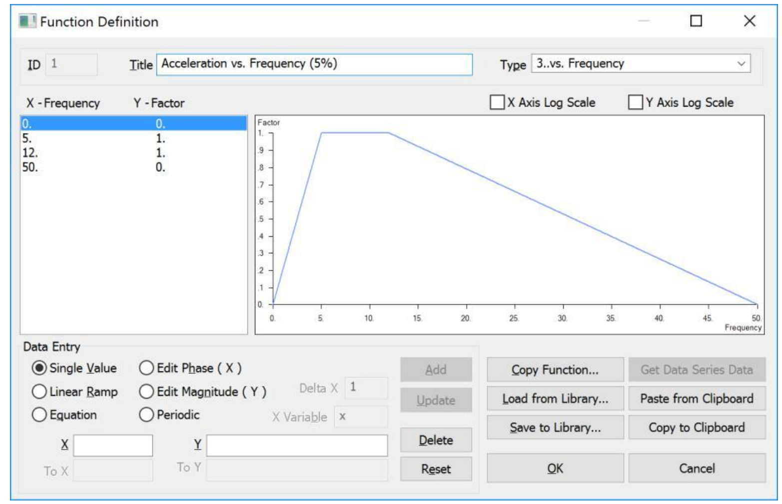
For this type of analysis, no load set is required, the analysis is performed by creating response spectrum, interpolation and damping functions. Then everything is tied together in the Analysis Manager as a Normal Modes/Eigenvalue analysis



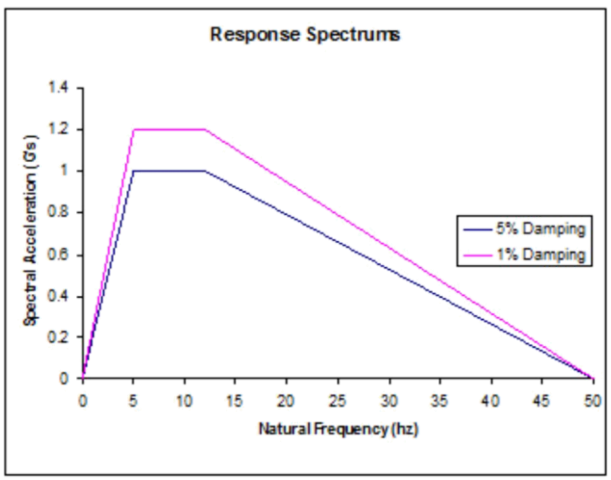


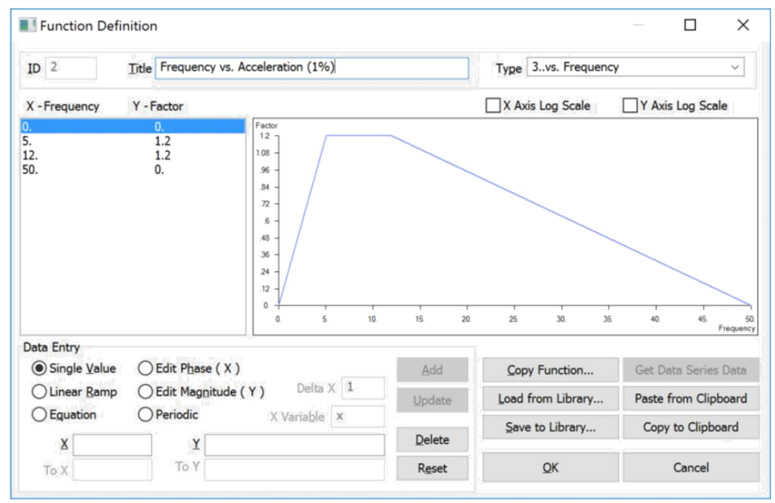
14.4 STEP 2: DEFINING THE RESPONSE SPECTRUM

A total of 2 functions will be created to define the response spectrum for the analysis, they are Type 3: vs. Frequency. One will be created for the 5% damping curve (Function ID 1)



and another for the 1% damping curve (Function ID 34). The curves are plotted below. Any number of response spectrums can be defined, depending on the requirements of the design. For this example, we will be using 3% of critical damping. The user can choose from Q damping, structural damping or critical damping





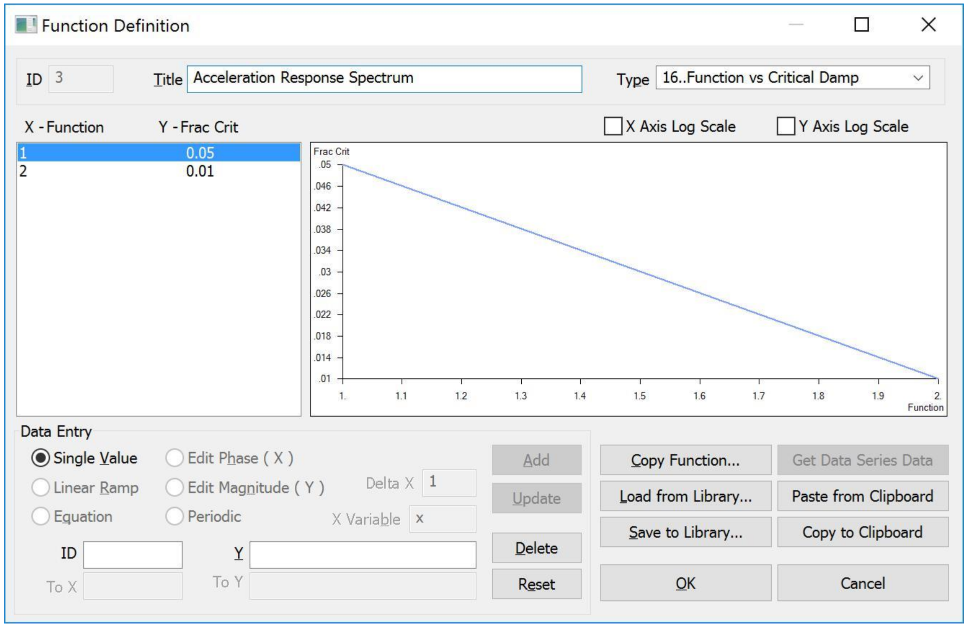
14.5 STEP 3: CREATING INTERPOLATION TABLE

Nastran requires an interpolation table to determine the system response, if the system damping is not given directly from a response spectrum curve. IMPORTANT, the interpolation table is mandatory, even if no interpolation is required between curves.

A function is created that assigns damping values to the previously created functions.

The function is Type 16..Function vs. Critical Damp. We will give the function a title of "Acceleration Response Spectrum"

A value of 0.05 (5% of critical damping) is assigned to function ID 1 and a value of 0.01 (1% or critical damping) is assigned to function 2. In this manner, the system response to a damping value other than the damping associated to the defined response spectrum curves can be calculated via interpolation between the defined curves.

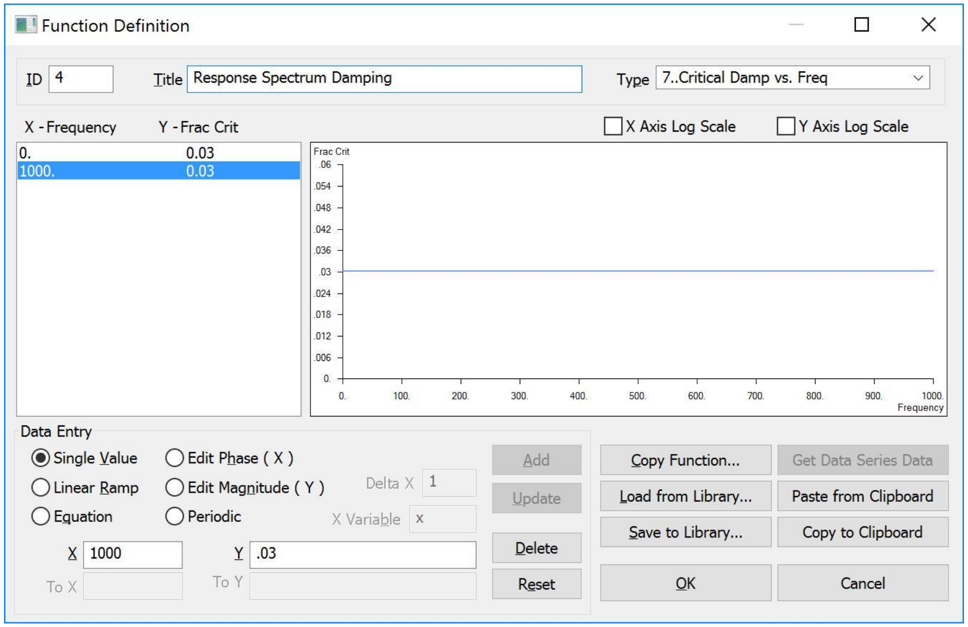


14.6 STEP 4: CREATING A MODAL DAMPING FUNCTION

The system damping can be defined as a function of frequency; we will use a 3% of critical damping across the frequency range.

The damping function is defined as Type 7..Critical Damp vs. Freq. We will give the function a title of "Damping"

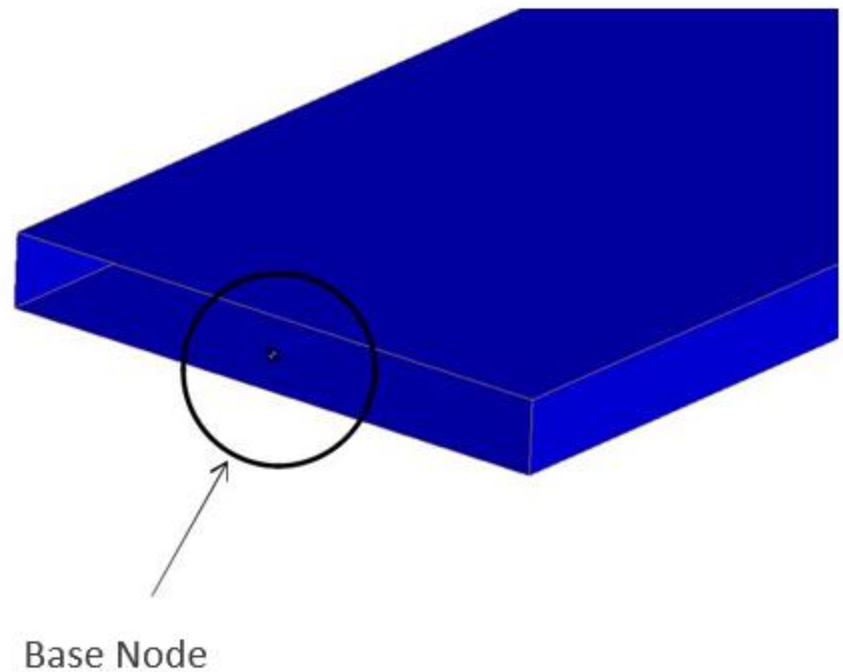
The function is used in association with the interpolation table. Since we have chosen a modal damping value of 3%, the system response will be found in the response spectrum between the 5% damping and the 1% damping curves. Note that if the system response at 5% damping was desired, the interpolation function previously created would only need to be defined for the function ID 1, and the damping function would then be defined as 5% over the frequency range.



14.7 STEP 5 CREATING THE EXCITATION NODE

For the response spectrum analysis, a foundation mass must be added to the excitation node. The mass should be 103 to 106 times the mass of the system.

Since this is a base excitation problem, and the base of the structure consists of one node, it is that node to which we will apply our acceleration response spectrum. In the case where the base of the structure is not one node, a rigid link approach is used to tie the multiple nodes of the base to a single node.

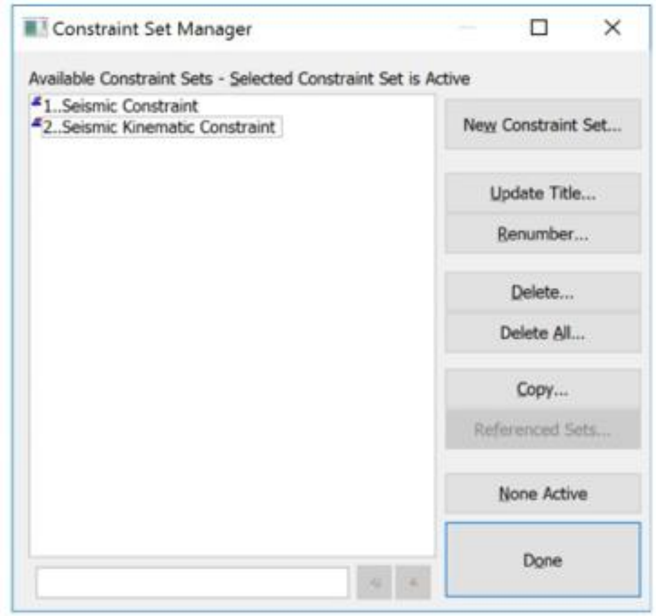


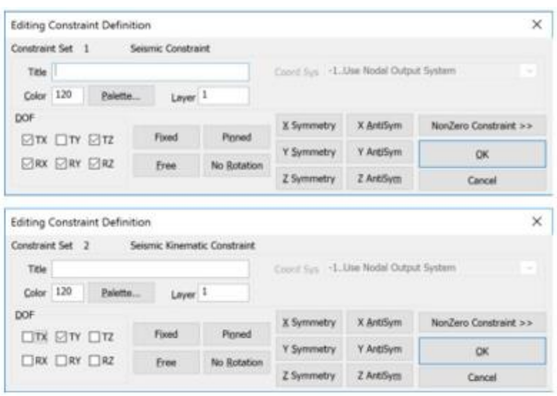
14.8 STEP 6: CONSTRAINING THE MODEL

In order to perform the response spectrum analysis, 2 constraint sets must be created.

The first set will constrain the model in all degrees of freedom except the excitation direction (in this example, y-direction).

The second constraint set will be used as a kinematics DOF. This is known as a SUPPORT set in Nastran. The DOF that the excitation is in is constrained.

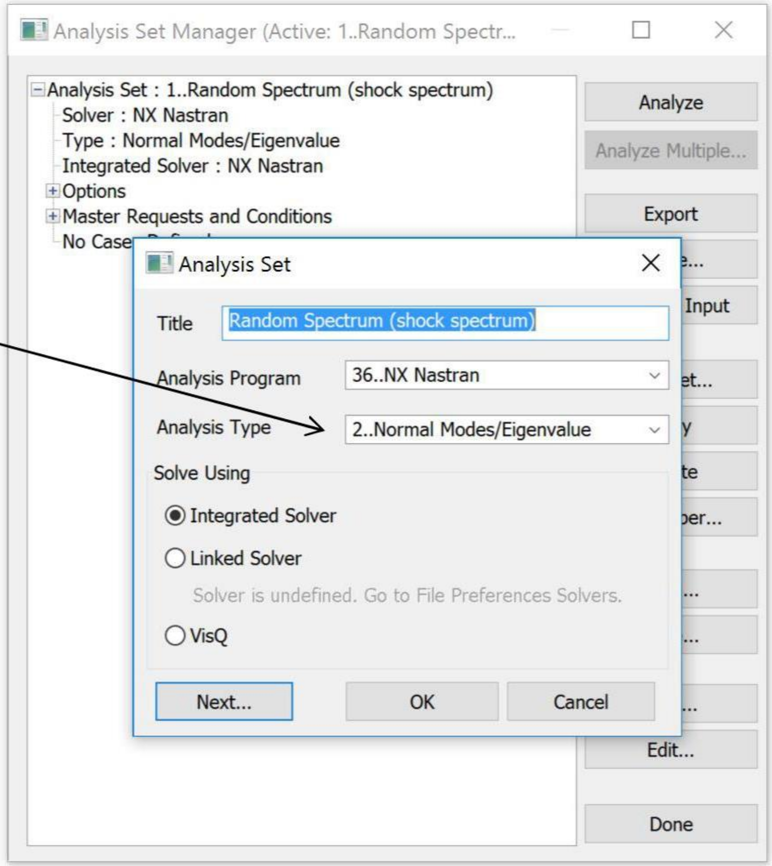




Note: Foundation mass not shown

14.9 STEP 7: SETTING UP THE ANALYSIS

Set up a new analysis set as type 2..Normal Modes/Eigenvalue.



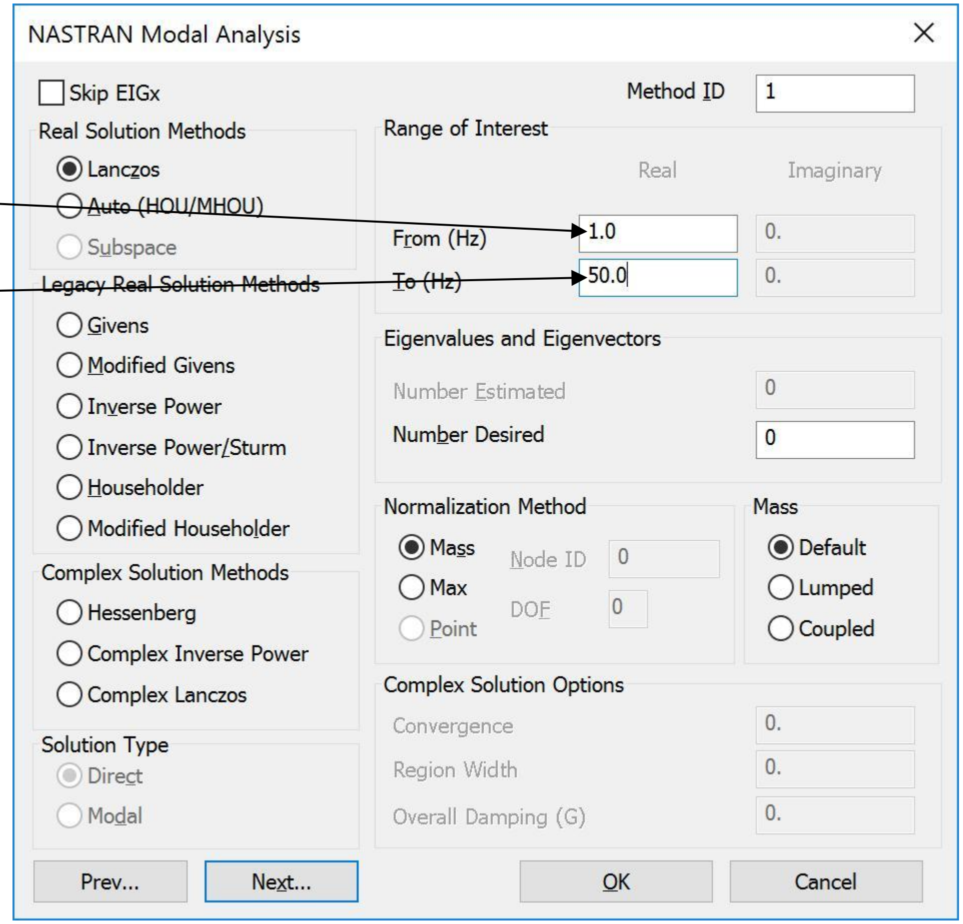
Click Next button 5 times, in the Range of Interest, type in;

1.0 in the From (Hz) field

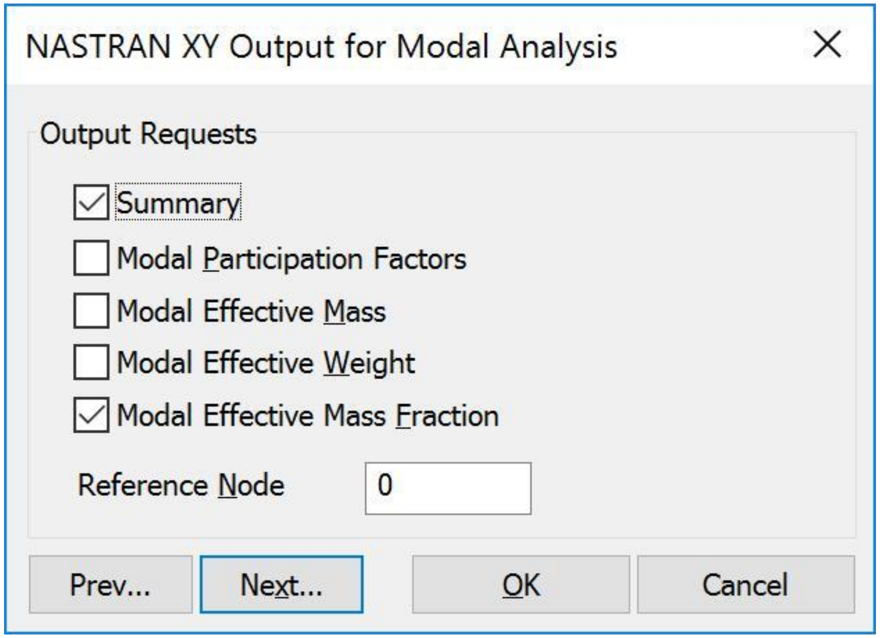
50.0 in the To (Hz) field.

This is very important since the first mode that will be calculated will be the rigid body motion (natural frequency of zero). If this field is left blank, the analysis will be performed from 0 Hz, this will give the incorrect value for the nodal velocity and displacement, and as such the stresses will be incorrect.

Note: The minimum frequency does not have to be 1.0, it can be any small number such that mechanisms are excluded.



Press Next button once. For the modal analysis, it is very useful to ask Nastran to output the Modal Participation Factors (PF). Select Summary and Modal Effective Mass Fraction. Nastran will output the PF's to the .f06 file. This is a very important tool when analyzing larger models that have multiple DOF and many eigenvalues. The PF's are used in combining the response from the individual modes based on the combination method selected.



However, you don't need to request'em since they are automatically calculated. This toggle just dumps them out to the f06 file and creates functions in FEMAP.

Press Next button once. This is the step that ties all the functions together.

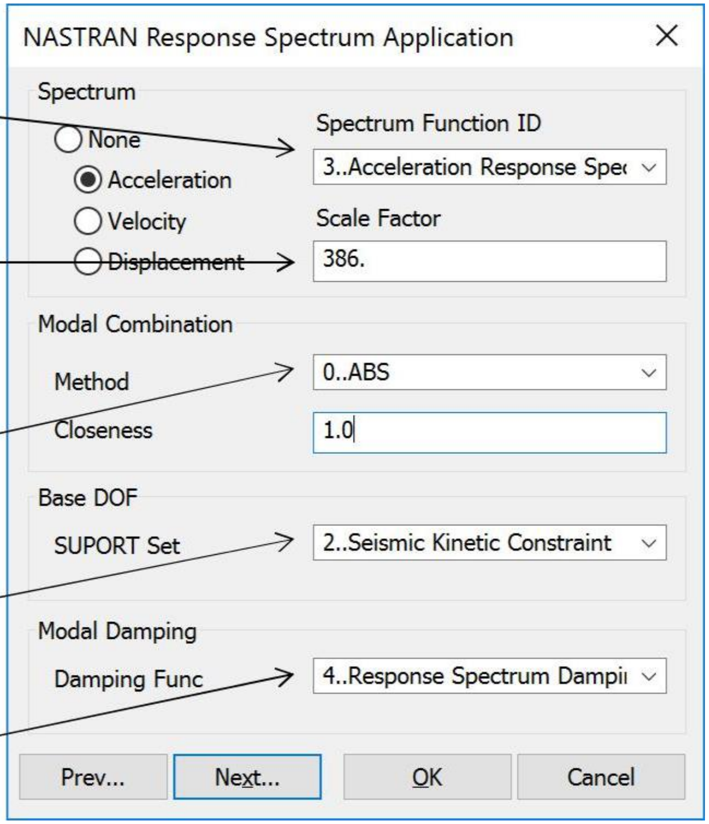
In the Spectrum – Spectrum Function ID drop down box, select “Acceleration Response Spectrum”.

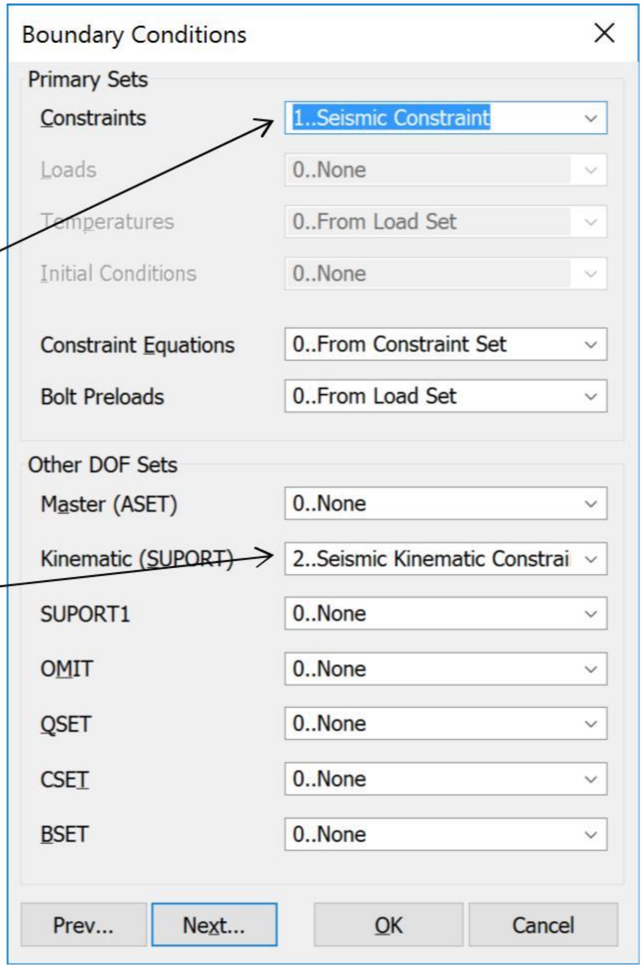
In the Scale Factor box, type in 386, this will scale the response spectrum curve from G’s to actual acceleration values in in/s2.

In the Modal Combination - Method box, select any type since only one mode is calculated in the 1.0 Hz to 50 Hz range. Here we are using ABS method, but for this example, the same results are obtained with all methods.

In the Base DOF - SUPORT Set drop down box, select “Seismic Kinematic Constraint”.

In the Modal Damping – Damping Func drop down box select “Damping”





Press Next button twice.

In the Boundary Conditions – Primary Sets – Constraints drop down box select "Seismic Constraint".

In the Other DOF Sets – Kinematic (SUPPORT) drop down box select "Seismic Kinematic Constraint".

Press Next button once.

In the Nastran Output Request – Nodal select,

Displacement

Constraint Force

Velocity

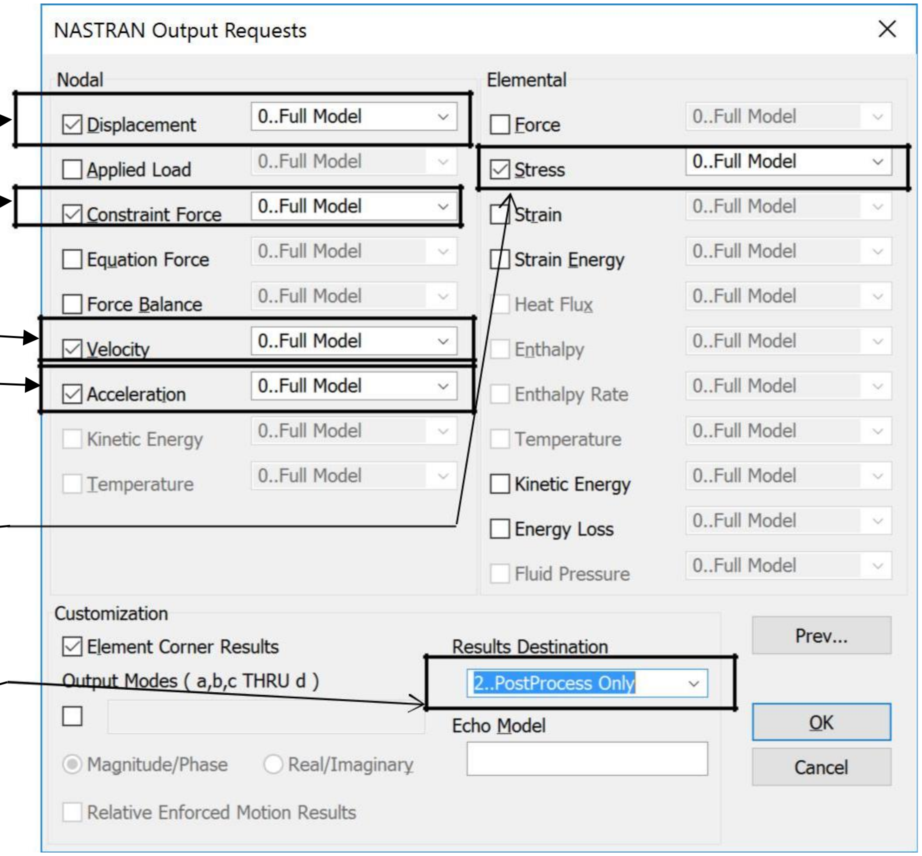
Acceleration

In the Nastran Output Request – Elemental select;

Stress

In the Customization area;

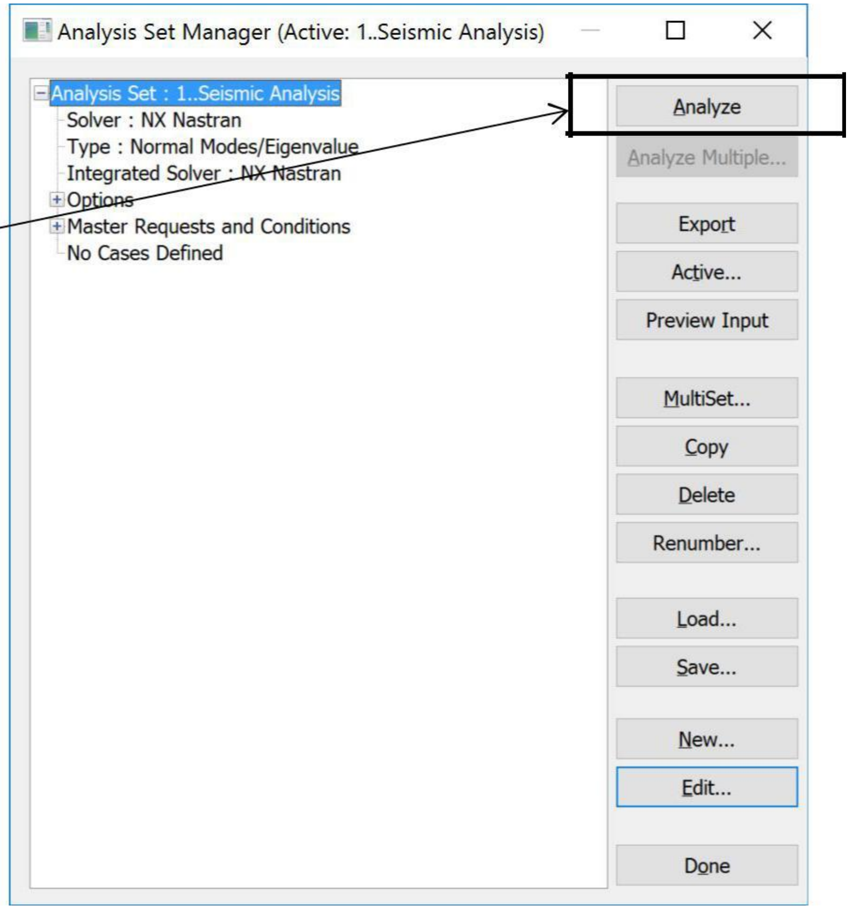
Set Results Destination to "2..PostProcess Only"



Click Ok

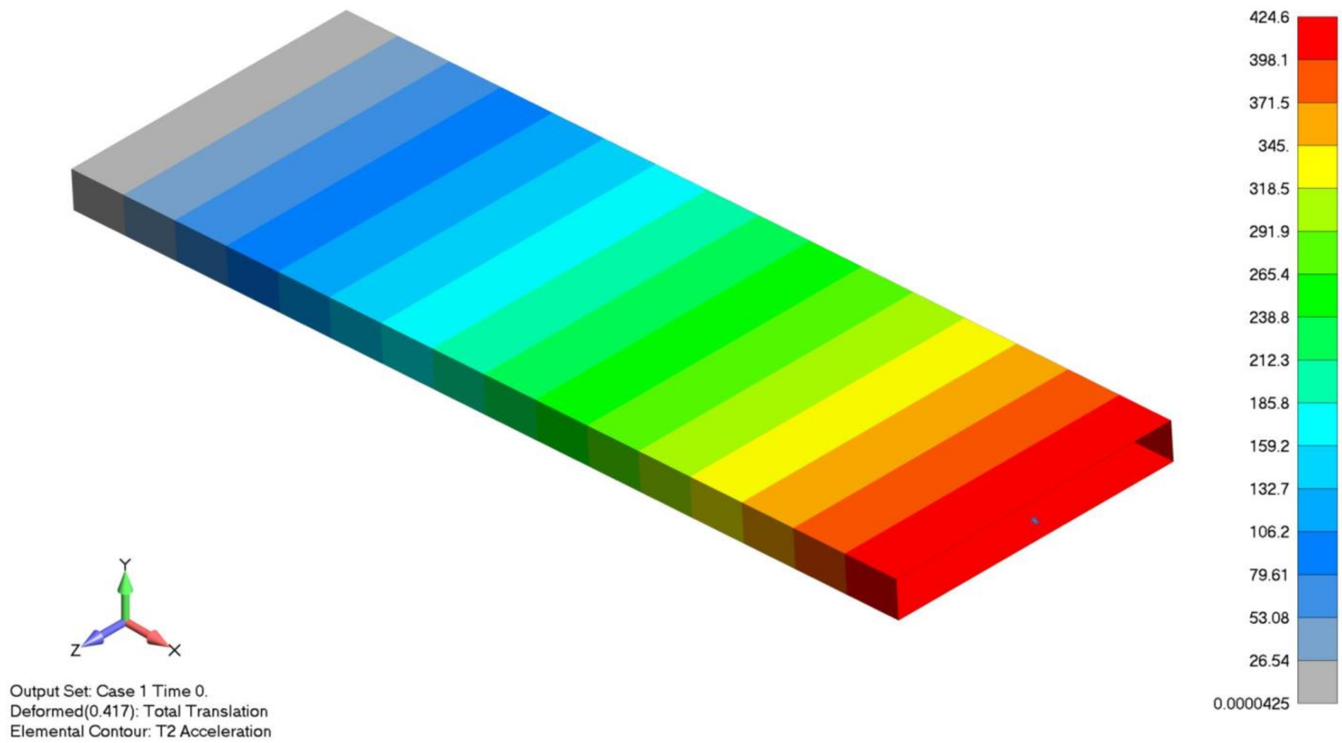
In the Analysis Set Manager, make sure that the Seismic analysis set is active, then click Analyze

We will also set up a Normal Modes analysis to determine the natural frequencies of the model (should get two eigenvalues, one being zero and the other being a bending mode).



14.10 POST PROCESSING THE RESULTS

Press F5 and select Beam Diagram from the Contour Style. Use the Post Data tool to look at the beam diagrams for the acceleration, velocity, stress etc. Below is the acceleration contour plot, note that the max acceleration is at the free end and has a value of 424.6 in/s2.



14.11 RESULTS COMPARISON

The results from the FE analysis follow very closely with the analytical results. The acceleration, velocity and displacement are exact (slight difference between analytical and FE velocity). Note that the natural frequency found in Nastran is slightly different than the analytical calculated value, this is because Nastran calculates the shear area of the beam. One can obtain the exact same results for the natural frequency in the FE model by excluding the shear area from the calculations, when this is done, the FE results match with the analytical results to 3 decimal places.

Results Comparison		
	Analytical	Nastran
Natural Frequency	5.1545	5.1509
Acceleration (in/s^2)	424.6	424.6
Velocity (in/s)	13.11	13.12
Displacement (in)	0.405	0.405
Stress (psi)	122,285	122,293

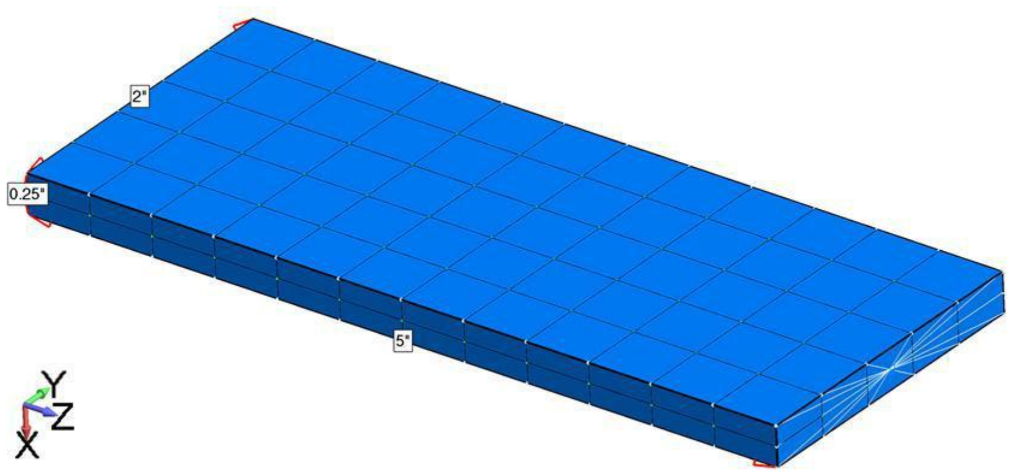
15. EXTRA CREDIT: SOLID MESHED BEAM

We can verify this method using a hex meshed solid beam. The beam is massless, with a point mass of 0.5lbf (1.30e-3 slugs) on the end. The beam properties are shown below:

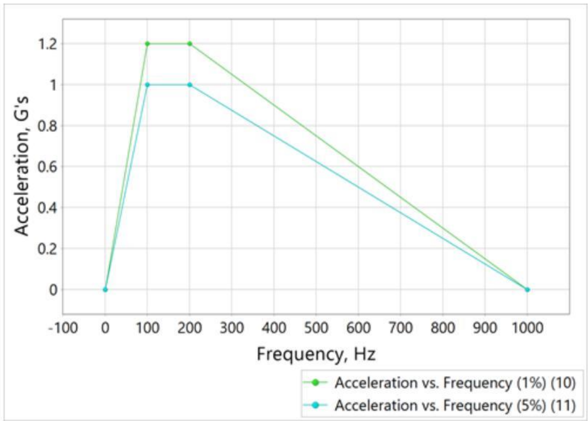
W = 2 in   W = 0.5 lbf

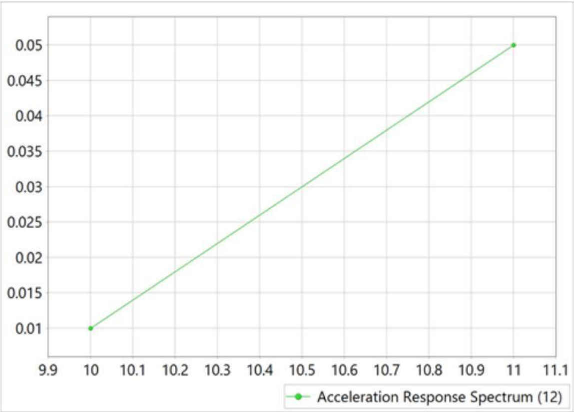
T = 0.25 in   E = 10e6 psi

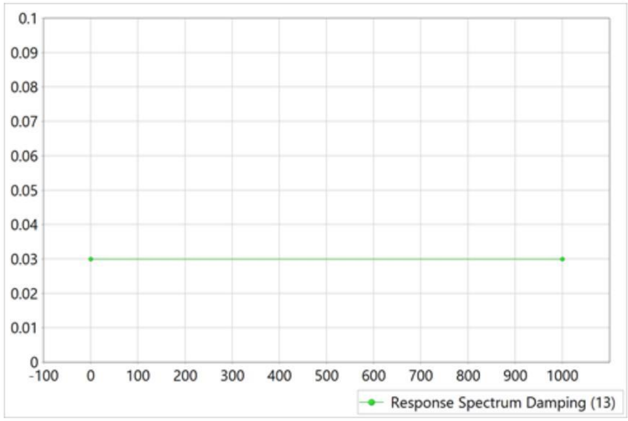
L_{beam} = 5 in I_{XX} = 2.6e-3 in^4



An Acceleration vs. Frequency spectral plot was created for 1% and 5% damping, an interpolation function was created to find intermediate damping values, and a damping function was generated.







Let's first take a look at the hand calc's to show how the beam is expected to behave.

Deflection is calculated first:

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

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

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

Spectral acceleration for this condition is

$$S_a = 1.1 * 386.09 = 424.6\ \frac{in}{s^2}$$

Continuing, we can approximate the spectral velocity, displacement, and stress:

Velocity:

$$S_v = \frac{S_a}{w_n} = 0.611\,\frac{in}{s}$$

Displacement:

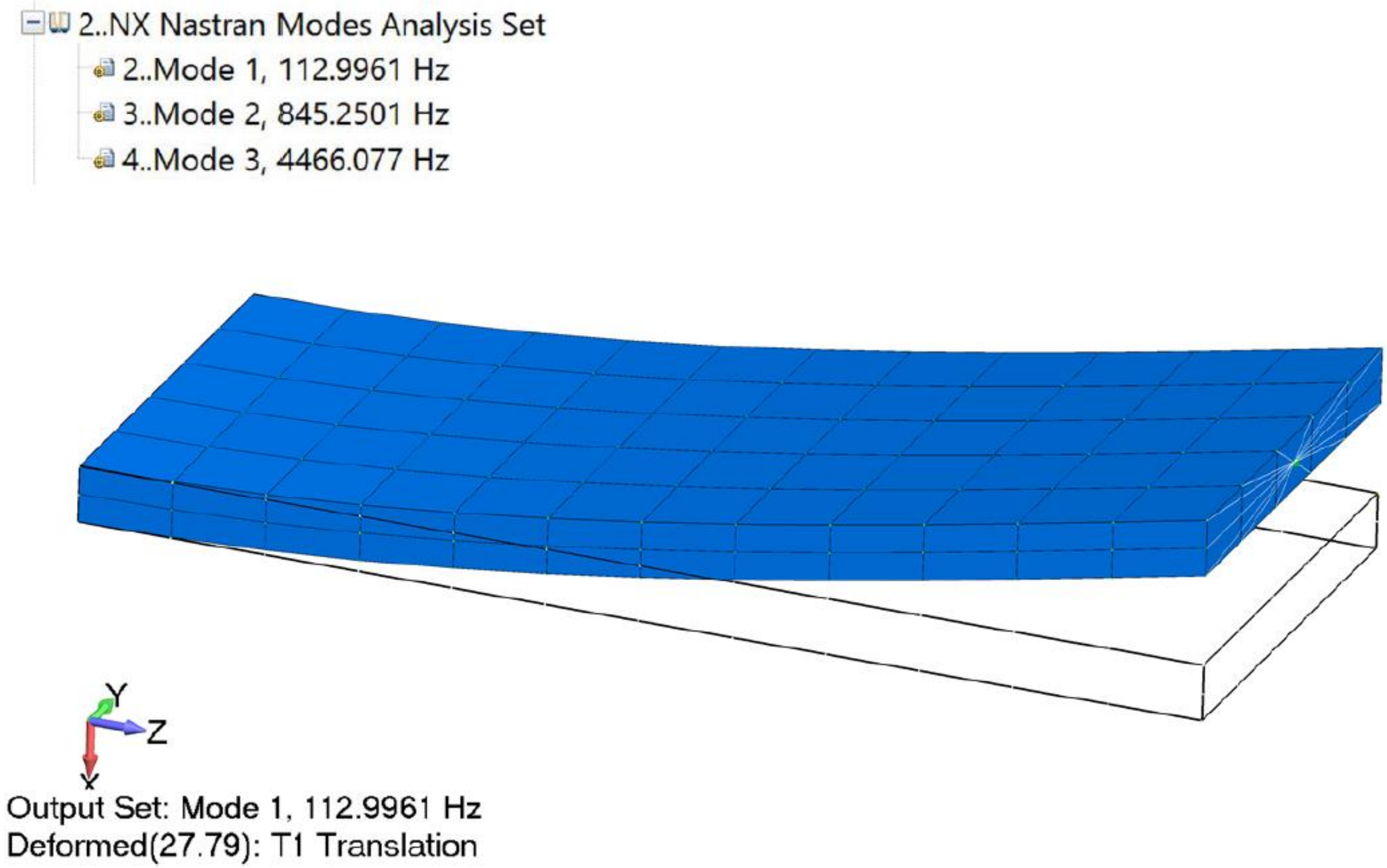
$$S_d = \frac{S_a}{{w_n}^2} = 8.80 \, * 10^{-4} in$$

Stress:

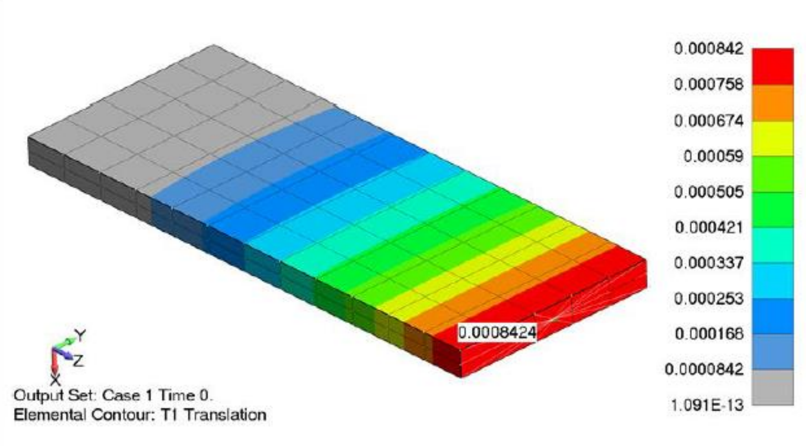
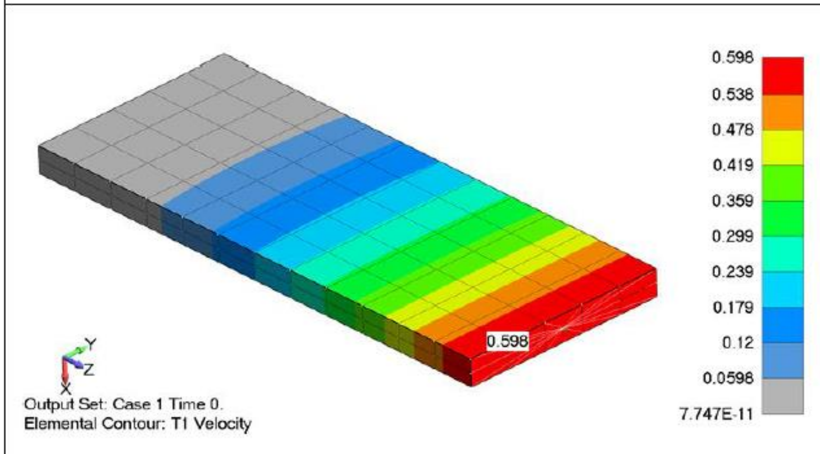
$$F_d = W * S_a = 0.55\ lbf$$

$$Stress = \frac{Mc}{I} = \frac{(F_d L)(\frac{T}{2})}{I_{xx}} = 132\ psi$$

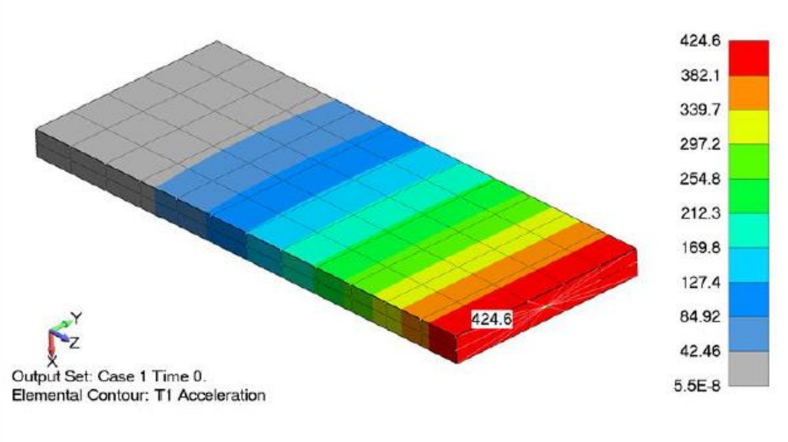
First up is a normal analysis for the solid model, the first mode is at 113 Hz:



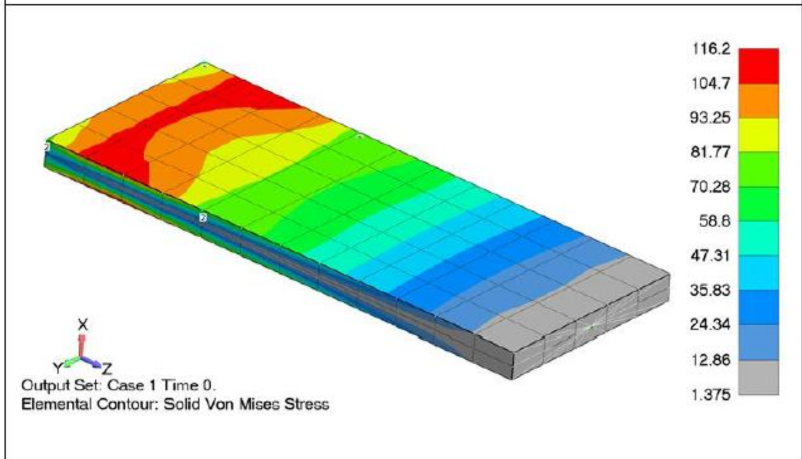
With the normal modes calculated, next is a shock response analysis:

	T1 Translation shows 8.42e-4 inches, hand calcs predicted 8.80e-4 inches.
	T1 Velocity shows 0.598 in/s, hand calcs predicted 0.611 in/s.

With the normal modes calculated, next is a random response analysis:



Max T1 Acceleration result is 424 in/s^2, the hand calcs showed 424 in/s^2



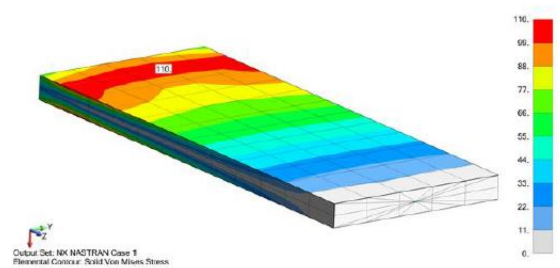
Max stress in the Z direction is 116 psi, the hand calcs predicted 132 psi

The FEA results for the solid model are not as close to the hand calculations as the beam element, but you can see the relative accuracy below:

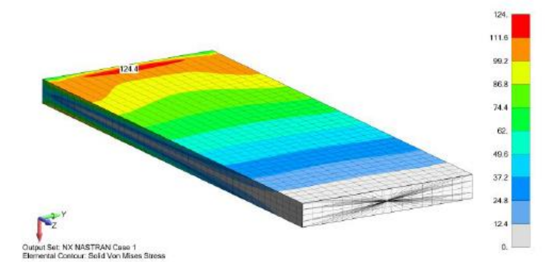
Results Comparison		
	Analytical	Nastran
Nautral Frequency (Hz)	110.6	113.0
Acceleration (in/s^2)	425	425
Velocity (in/s)	0.611	0.598
Displacement (in)	8.42e-4	8.80e-4
Stress (psi)	136	116

The error in the maximum stress of the FEA model is 12%, which seems a bit on the high side. Let's check a hand calculation of a 1G static load versus the finite element static load to explore the behavior. This beam with 1G static loading should see 120psi maximum stress based on the simple calculations.

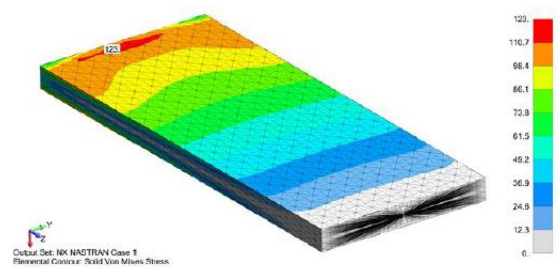
Maximum stress is 110 psi with a hex meshed solid element beam, which is 8% error from the hand calculations. This is similar to the error seen in the dynamic analysis.



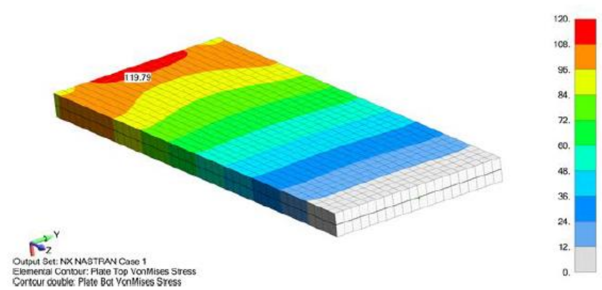
Maximum stress is 124 psi in a fine hex meshed solid element beam. This gives an error of just over 3% compared to the hand calculations.



Maximum stress is 123 psi with a 10-node tet meshed solid element beam. This gives an error of under 3% which is a large improvement.



Maximum stress is 119.8 psi with a quad meshed plate element beam. This matches with the hand calculations very closely with an error of 0.2%.



It is interesting to compare the accuracy with the # of nodes required for analysis. In a solid element model it seems that the coarse hex mesh does not have enough definition to capture the stresses and has a relatively large error. The fine hex mesh drops the error down to 3% with just under 4,000 nodes. Meshing with a fine 10-node tetrahedral mesh gives similar accuracy but requires 26,000 nodes, a significant increase! The plate meshed beam is the closest with 0.2% error and under 900 nodes.

A summary of the findings for a 1 G static load is shown in the table below.

	Max Stress	% Difference	# of Nodes
Hand Calculation	120 psi	n/a	n/a
Coarse Hex Mesh	110	8%	236
Fine Hex Mesh	124	3%	3,782
Fine 10-node Tet Mesh	123	3%	26,274
Fine Quad Plate Mesh	119.8	0.2%	884

16. APPENDIX

16.1 FLOW CHART FROM NX NASTRAN THEORETICAL MANUAL

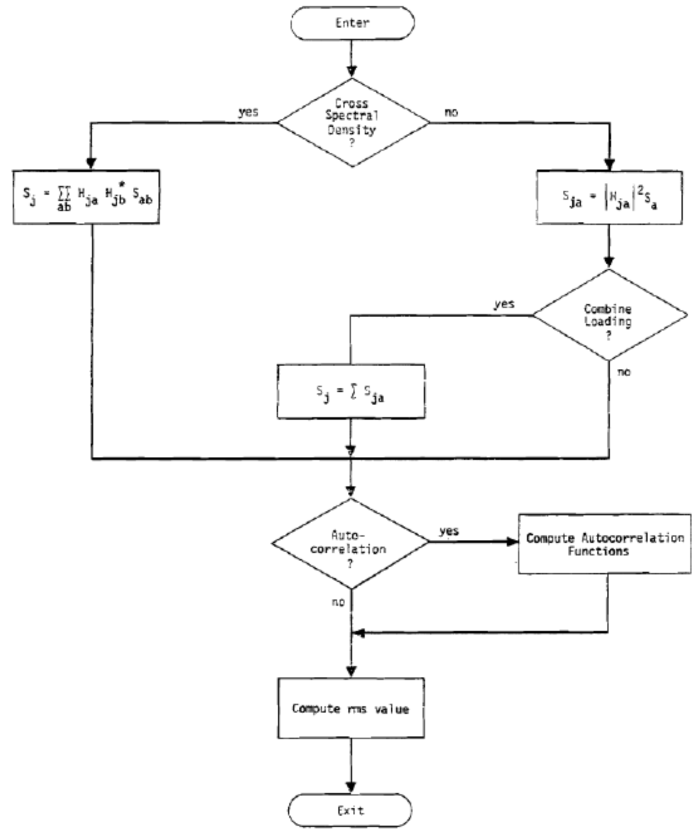


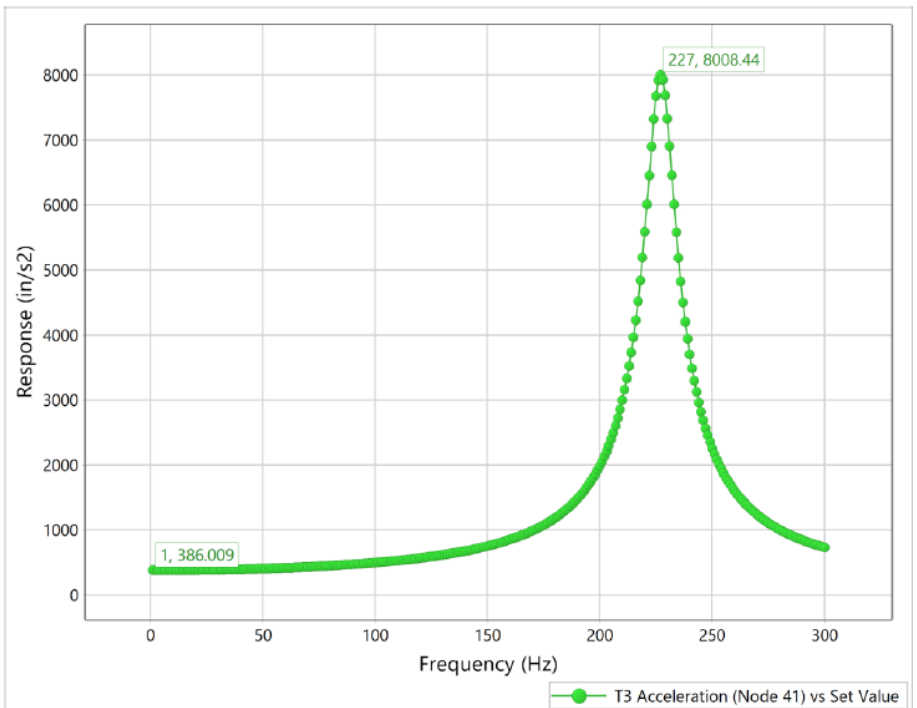
Figure 1. Flow diagram for random analysis module.

12.2-4

16.2 CREATING MODAL FREQUENCY TABLE

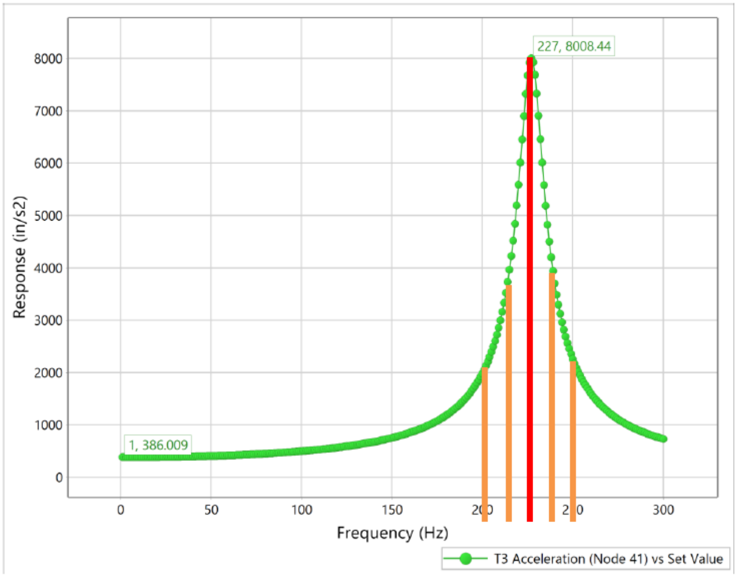
When Nastran calculates the RMS stress value for a PSD analysis it first calculates the frequency response at each value on the Modal Frequency Table, and uses that response to calculate the stress due to PSD excitation.

The chart below shows the frequency response for a simply supported beam with the first mode at 227 Hz, and the next mode above 300 Hz. The response acceleration ramps up near the natural frequency and gradually drops off as you move away from it.

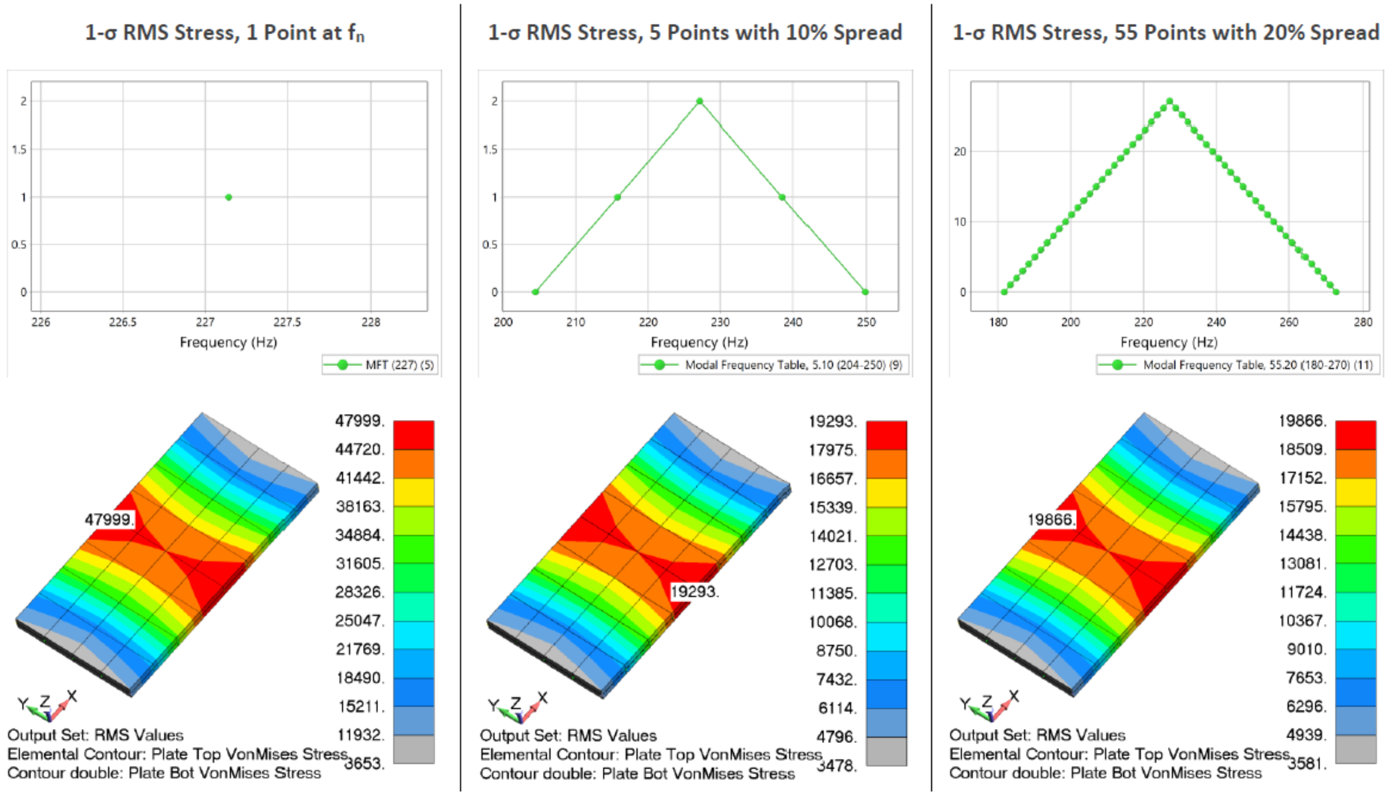


PSD analysis is statistical and the 1-sigma stress output is simply the stress the structure will likely see when subject to a specified acceleration spectrum. The random vibration solver doesn't calculate the stress at every frequency—it only solves for the stress at the frequencies specified in the Modal Frequency Table. The simplified process is that it solves for the stress at each value in the Modal Frequency Table, and then combines those stress results to give the RMS stress.

The red line in the image below shows the response at the natural frequency. The orange lines show the response at multiple points with a 10% spread from the natural frequency. As you can see, the response drops off as you move away from the natural frequency so adding more solve points, or a greater spread from the natural frequency does not improve accuracy of the results, but it does add significant computational cost.

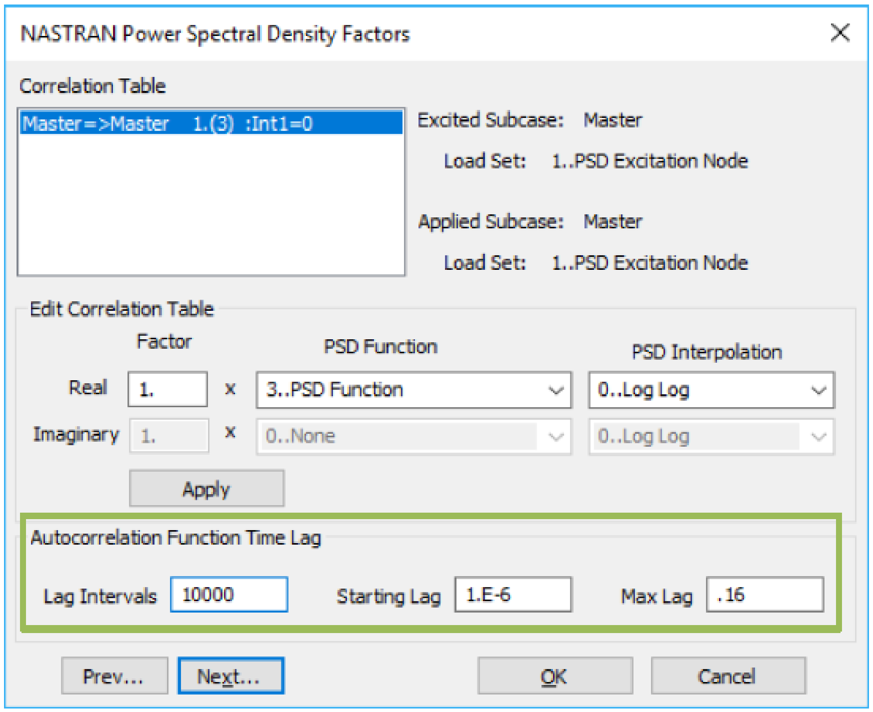


The chart below shows Modal Frequency Tables (X-axis is frequency, Y-axis is arbitrary) with varying number of solution points and the resultant RMS stress on a simply supported beam. With a single solve point at the natural frequency it significantly overestimates the RMS stress. The default 5 points with 10% spread gives a more reasonable result, and you can see even going up to an impractical 55 points with 20% spread gives a result within a few percent of the default table configuration.



16.3 AUTOCORRELATION FUNCTION

A newly supported feature in FEMAP 11.3 is the ability to generate Autocorrelation functions during random vibration analysis. The Autocorrelation Function Time Lag input is available in the *NASTRAN Power Spectral Density Factors* dialog as shown below:



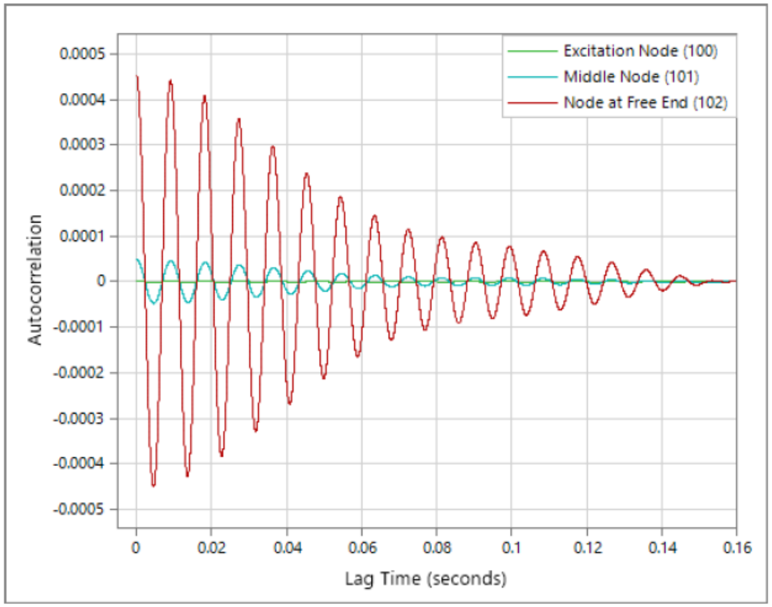
The three inputs available are Lag Intervals, Starting Lag, and Max Lag. These inputs do not change the way the random vibration analysis is conducted, it merely defines the autocorrelation function which will be generated in addition to the output.

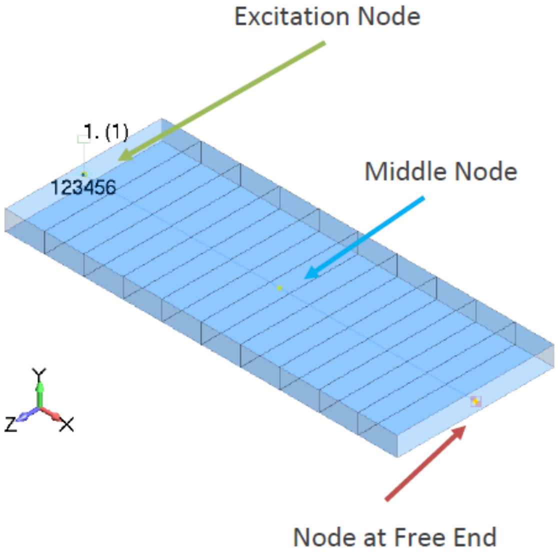
Lag Intervals: How many times to chop up the time band between the starting lag and maximum lag

Starting Lag: Starting time step

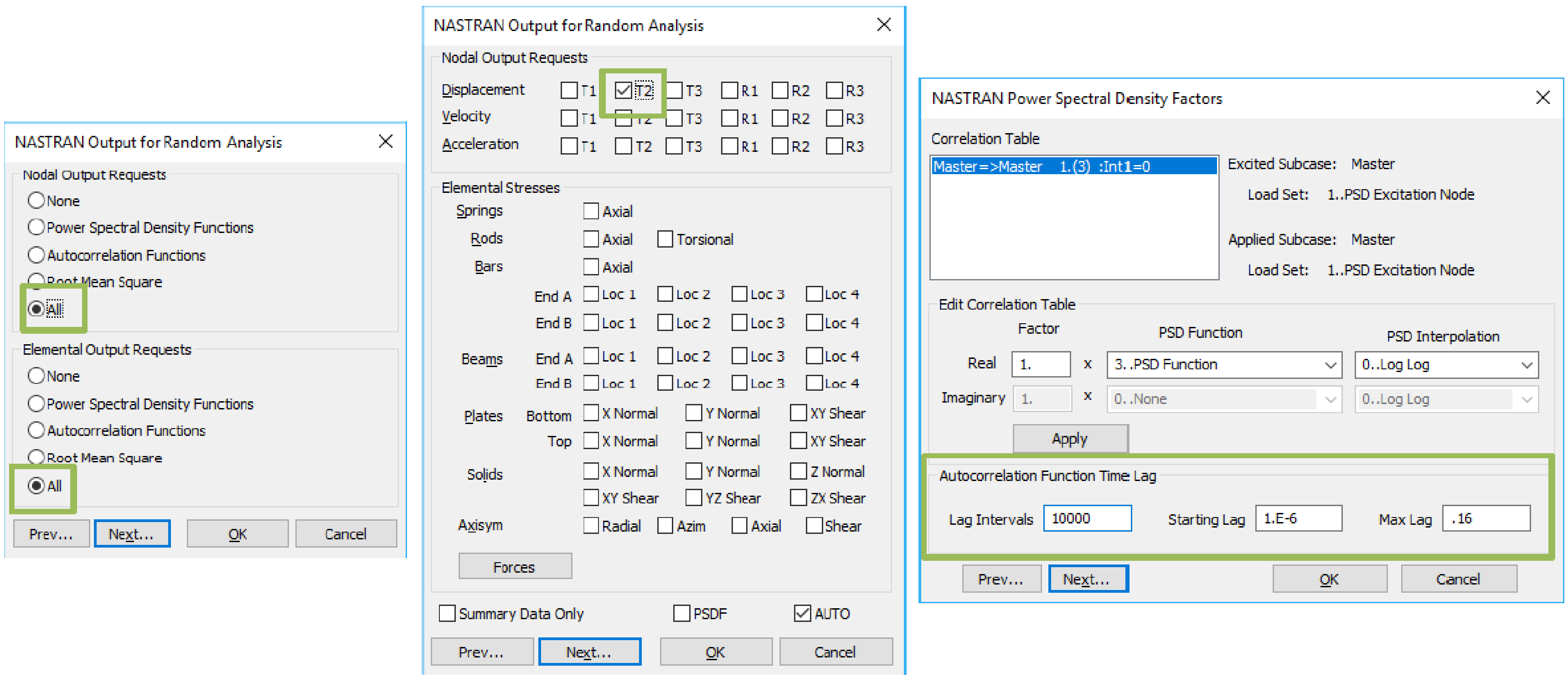
Max Lag: Final time step

If we take our beam example and plot the autocorrelation function for displacement on a couple of nodes we can get a more intuitive idea of what is going on. No matter the lag time, the autocorrelation at the excitation node is very close to zero. At lag = 0, the autocorrelation for the node at the end of the beam is high while the node at the middle of the beam is lower amplitude, and follows the same sinusoidal pattern. From this plot we can infer that a small lag time results in a high autocorrelation at the beam end, and it tapers off as you increase the lag time. It is worth noting that the period of the sinusoidal response shown here is 0.009 seconds, which matches the 110 Hz natural frequency of the beam.



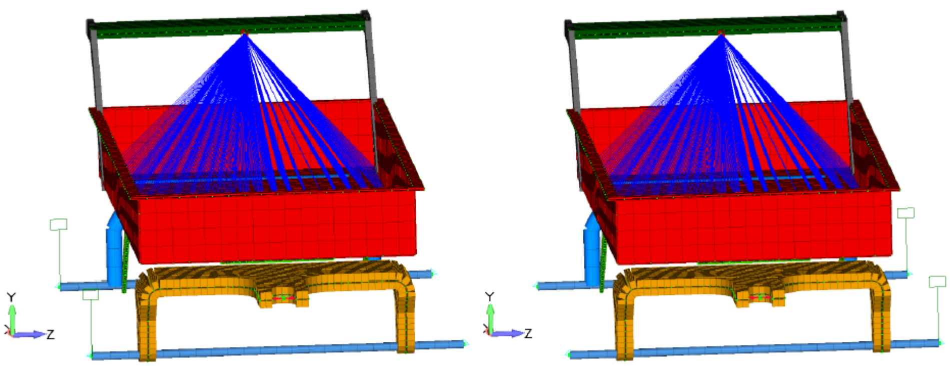


At this point you may be wondering how to generate the autocorrelation function in your analysis. In the first *NASTRAN Output* dialogs, select “Autocorrelation Functions” or “All”. Choose the desired nodal or elemental outputs to plot, and then enter the desired Autocorrelation Function Time Lag values.

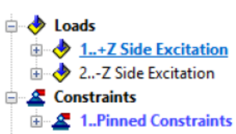


16.4 MULTIPLE EXCITATION SPECTRUMS

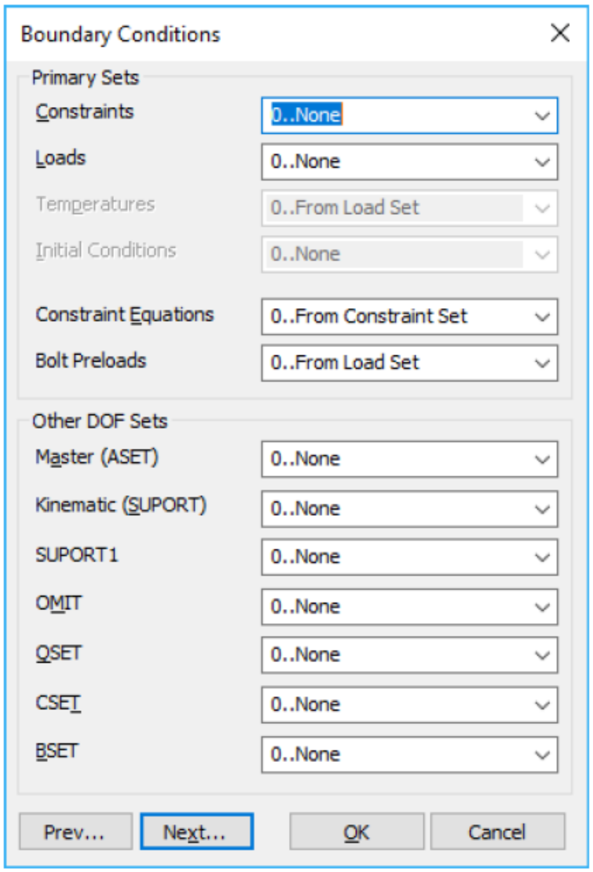
In some cases you may need to analyze a system with multiple excitation spectrums. This can be due to a difference in structure mounting points, wheels driving on different surfaces, or an array of other situations. In this example we will take a look at a hypothetical red wagon, where the wheels on the –Z side are on a smooth section of road, and the -Z side is rolling on over a rougher road.

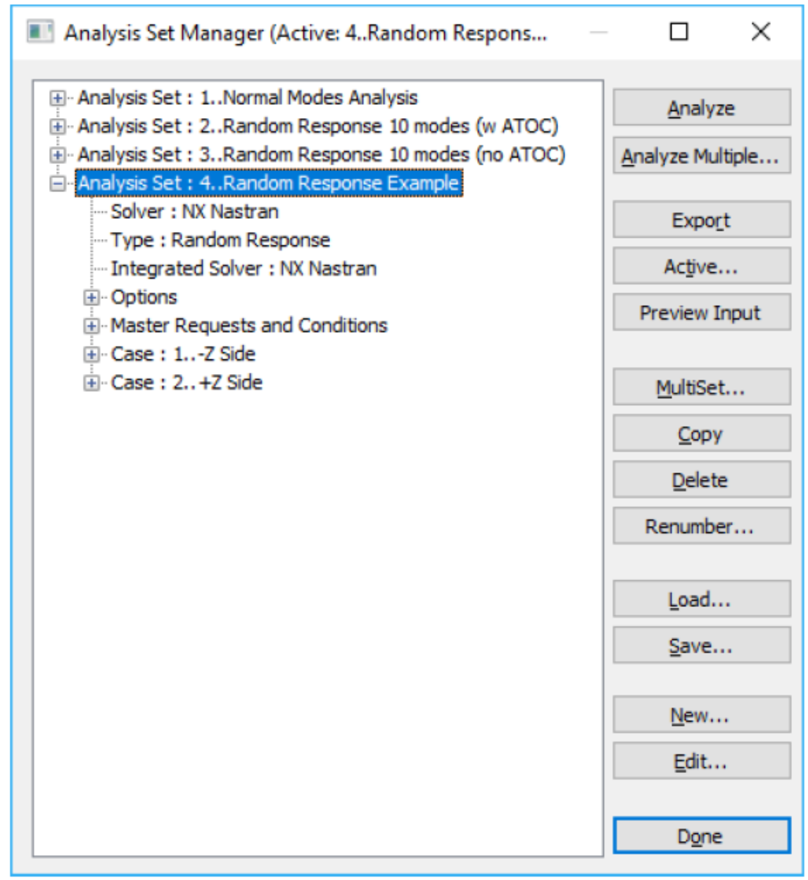


First the boundary conditions must be set up as shown—with your excitation points defined in separate load sets and all of the constraints in one Constraint Set.

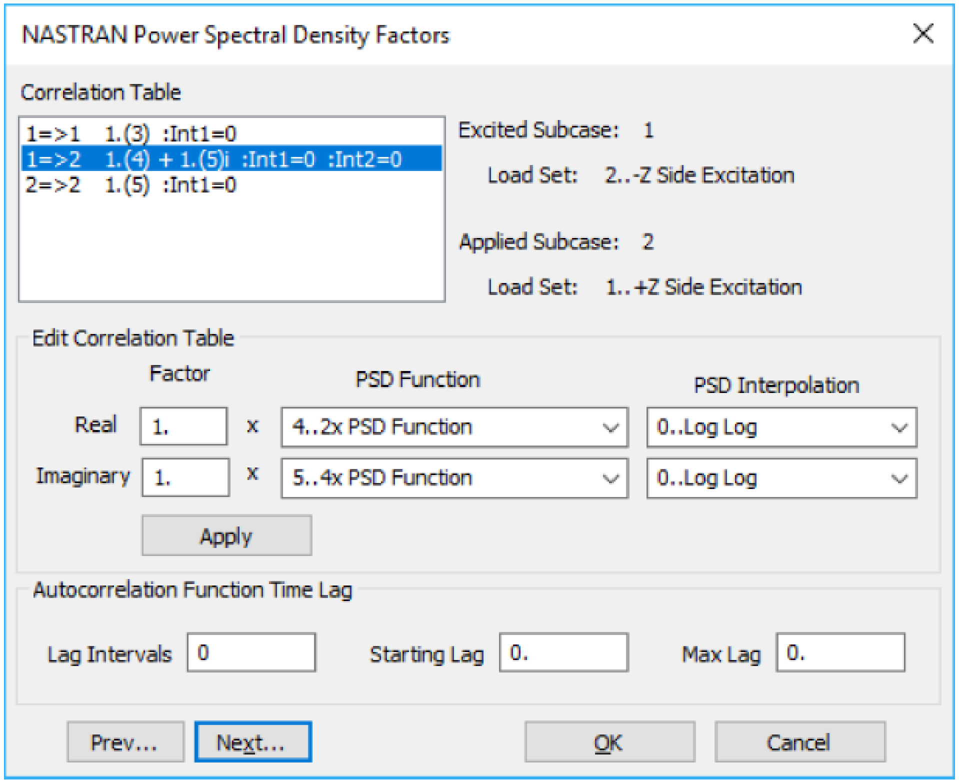


With the boundary conditions set up, prepare the random analysis in the usual manner, but do not specify boundary conditions. Instead, specify the boundary conditions as subcases in the Analysis Set Manager.





After that is set up, go back to edit the PSD Factors tab. Here you will notice it looks a little different than the previous method. You can now choose PSD Functions for each subcase, and you can correlate the two sub cases for coupled analysis if desired. If you do not wish to correlate the sub cases, leave the settings at their default values.



16.5 WHY WE DO A PSD ANALYSIS

Dynamics are tricky. Structures that seem sturdy intuitively may have unexpected responses when excited dynamically. The images below show a circuit board which was subjected to an intense PSD spectrum where the assembly was expected to see accelerations near 700 g's! In this case, the circuit board was not designed for that level of excitation and the chips removed themselves from the board during testing.