FINITE ELEMENT ANALYSIS
Predictive Engineering

Random Vibration Analysis FEMAP 10.3.1

An Introduction to The How's and Why's

Table of Contents

Introduction

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

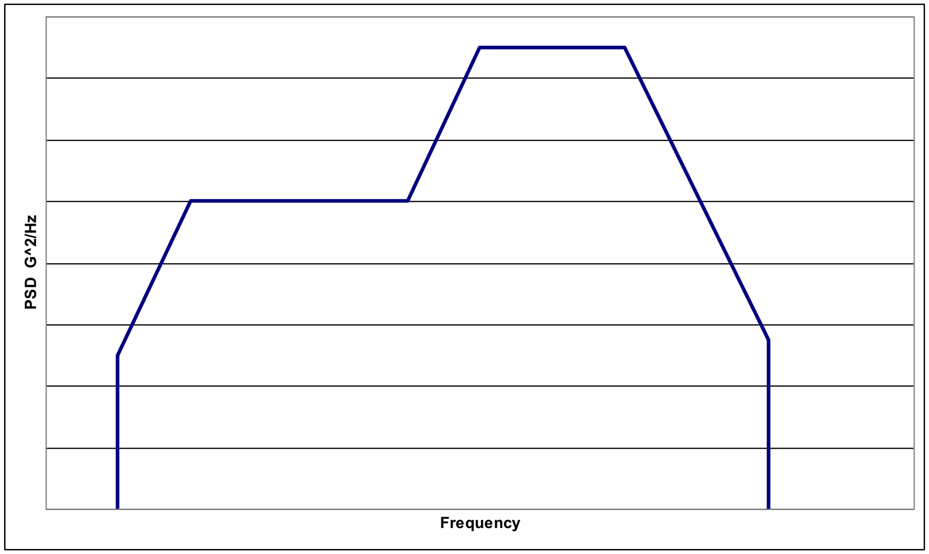
Random vibration problems arise due to earthquakes, tsunamis, acoustic excitation (e.g., rocket launches), wind fluctuations, or any loading which is inherently random. Often random noise due to operating or transporting conditions can also be considered. These random vibrations are usually described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

The PSD Function

The input for random vibration problems is usually in the form of a power spectral density (PSD) function. The PSD function is dimensionless. The acceleration is divided by the acceleration of gravity:

$$G = \frac{a}{g} = \frac{acceleration}{gravity}, \qquad \text{dimensionless}$$

An acceleration of 10 G means that the acceleration has a magnitude that is 10 times greater than the acceleration of gravity. A sample PSD curve is shown on the right. For a given frequency, the PSD input in given.



The NX Nastran Method

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

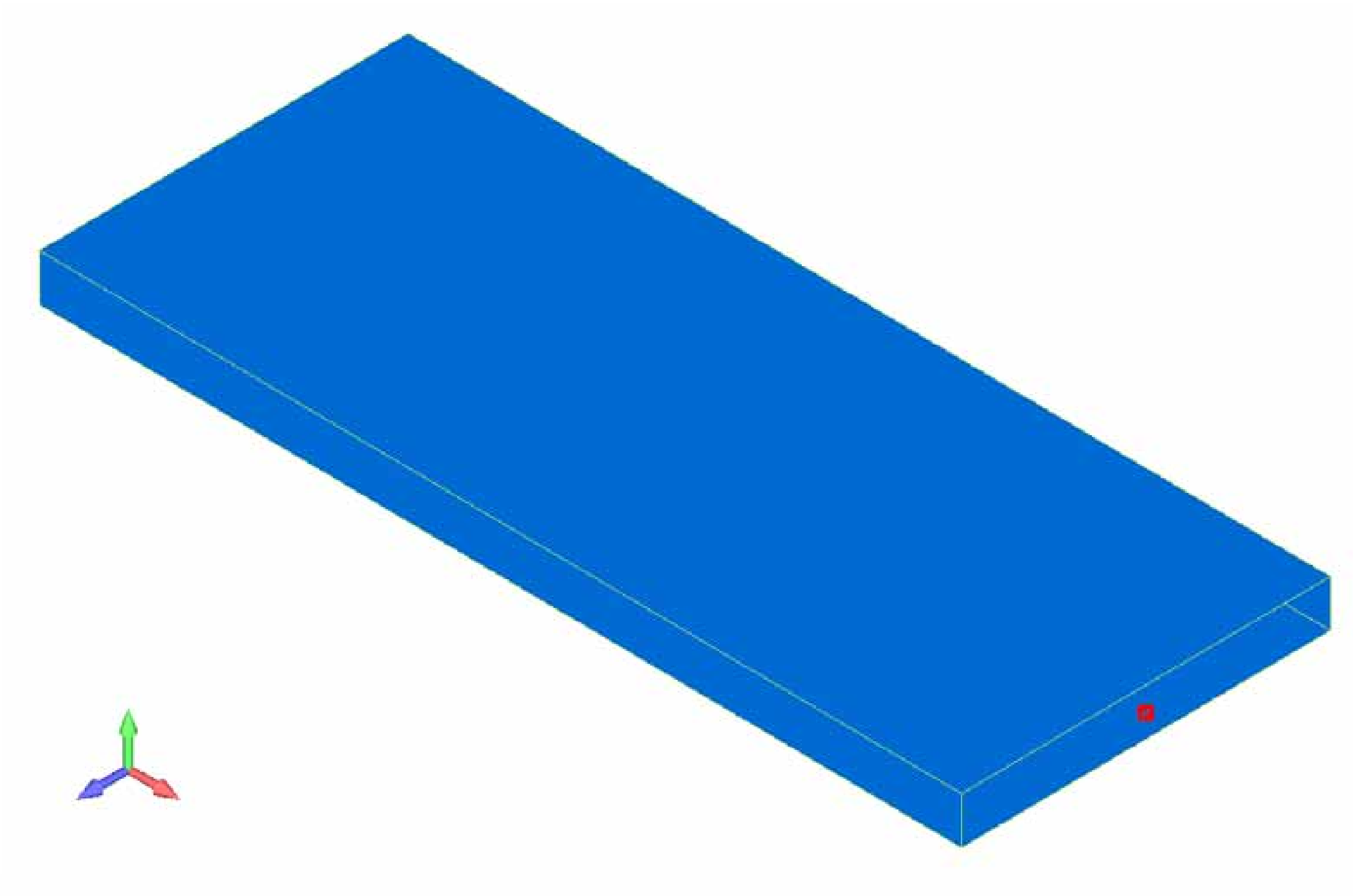
$$PSD_{Out} = |g(\omega)|^2 \, PSD_{In}$$

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

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

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

Example: Cantilever Beam



Problem Definition

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

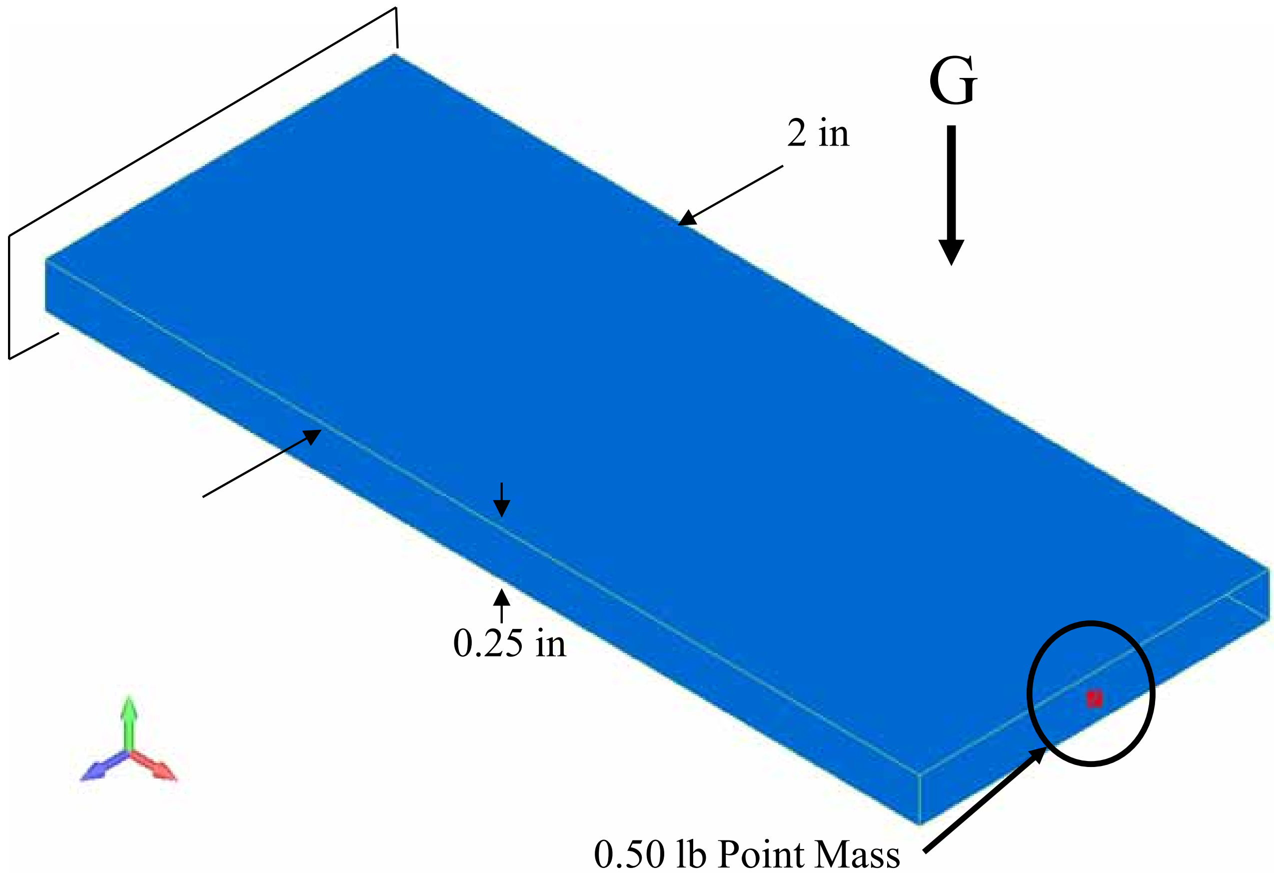
The FEA model is a single beam element. A picture of the beam element, with its cross section displayed is shown on the right.

We will compare the FEA results to an analytical solution[Ψ]. The PSD input (PSD_{IN}) function used by Steinberg was

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

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

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



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

Analytical Solution

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

$$Y_{St} = \frac{WL}{3EI} = 8.01E-4 \text{ in.}$$

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

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

For a beam, the transmissibility can be approximated as:

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

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

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

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:

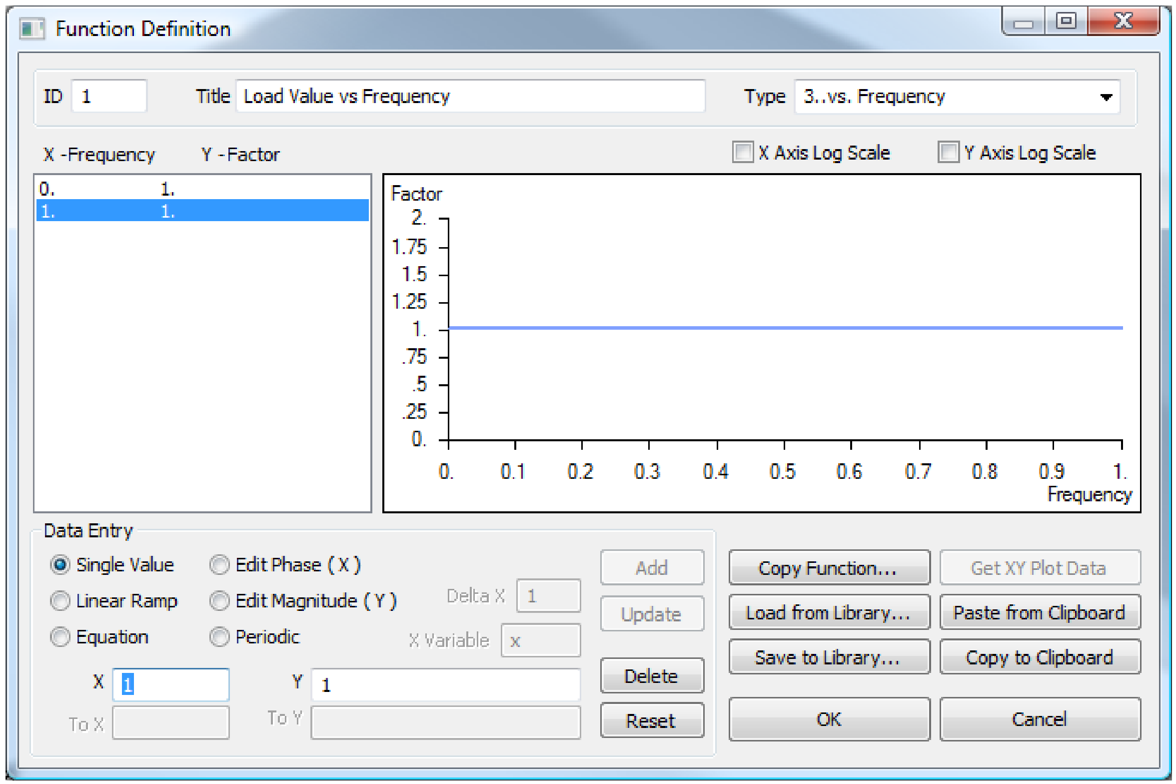
$$\mathrm{PSD_{out}} = \mathrm{Q}^2 \cdot \mathrm{PSD_{in}} = 21^2 \cdot (0.2 \cdot \mathrm{G}^2) \text{ where } \mathrm{G} = 1\mathrm{g} \text{ or } 386 \mathrm{in/s^2}$$

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

Step 1: Creating the Load Function

4 functions must be created in defining the analysis. The first that we will create is the load function.

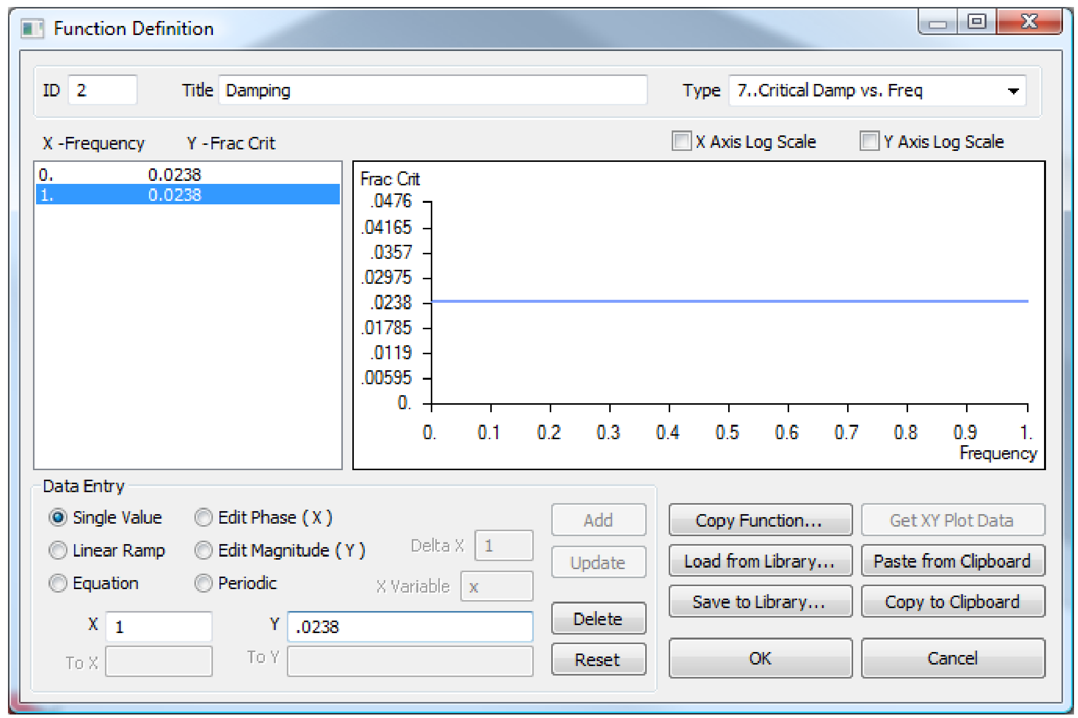
The Load function consists of two points: (0,1) & (1,1). This function essentially defines a constant load across all frequencies.



Step 2: 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.



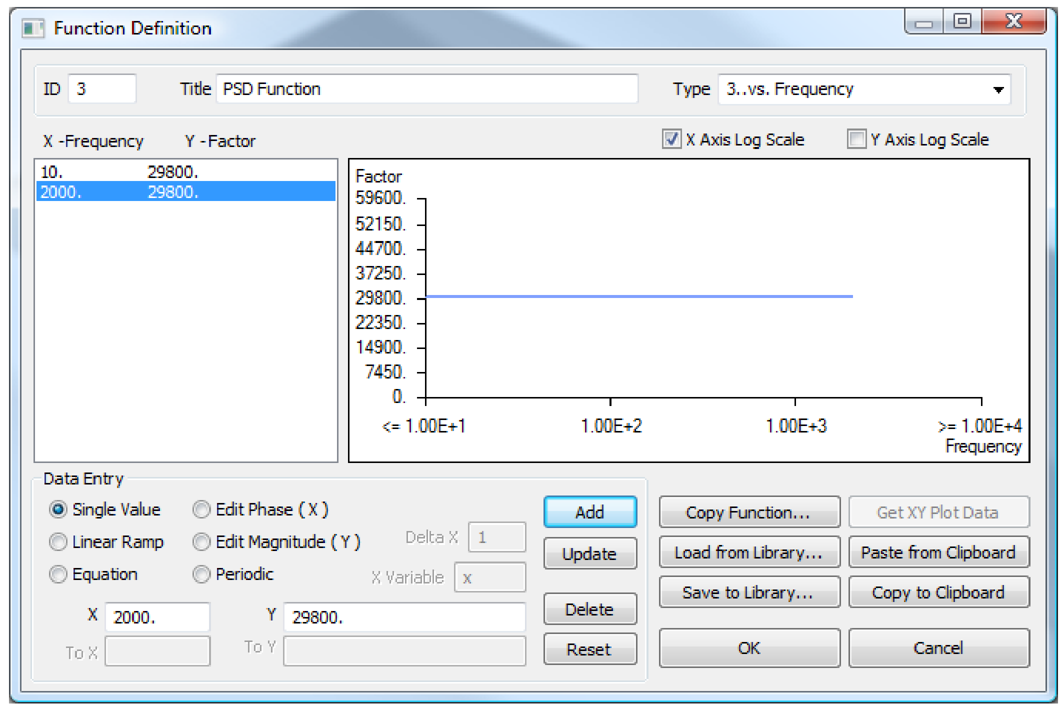
A good approximation of the transmissibility of the beam is Q = 21. This value yields a critical damping ratio of 2.38%; this is what we will use.

Step 3: Creating the PSD Function

The input to the cantilever beam is a white-noise vibration with a PSD input of 0.20 G^2/Hz from 20 to 2000 Hz.

Entering the PSD as G^2/Hz will cause all our output to be in G, including stress. Most analysts prefer their stresses in psi or Pa. For output to be in psi, we need to scale the PSD function so that it is in consistent units, instead of G. We will enter a value of $0.20*(386)^2$. Since the input is now in $(in/s^2)^2$/Hz, all of our output will be in inches, psi and in/s^2.

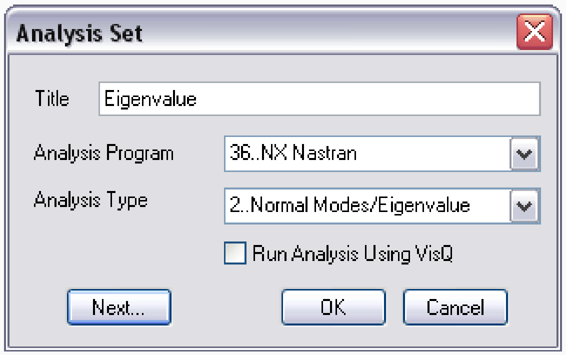
Scaling either the acceleration load, or the acceleration load curve will produce the same results, but it is more generally accepted to scale the PSD curve as described above.



Step 4: Creating the Modal Frequency Table / Setting up the Load Set Options for Dynamic Analysis

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

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

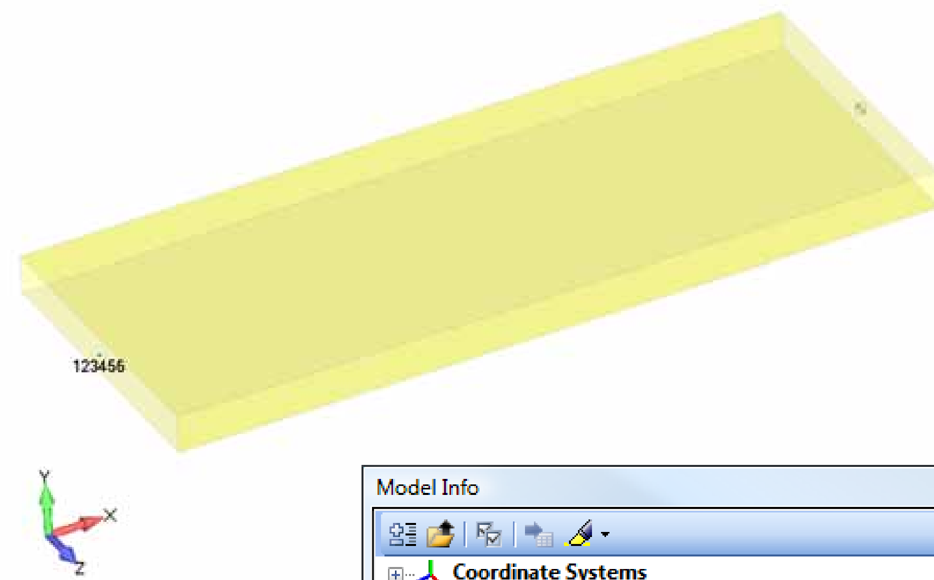


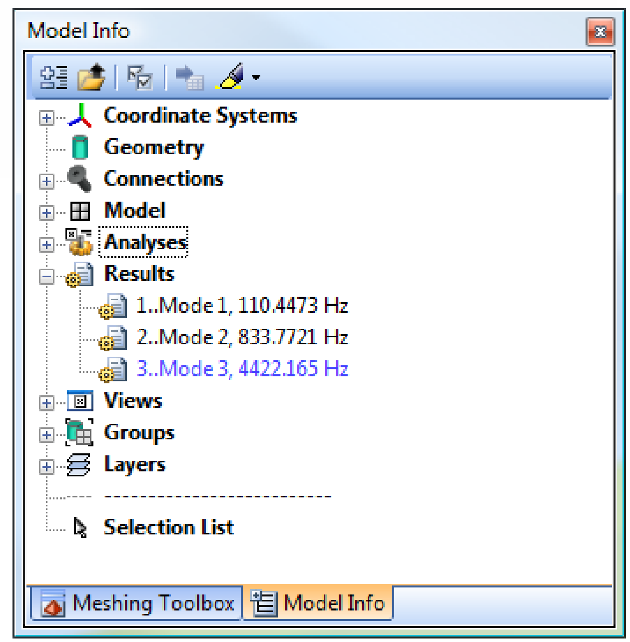
The normal modes will be used to define the solution frequencies of the Random Analysis. Think of it as guiding the Random Analysis such that only frequencies of interest (significant frequencies) are processed. This greatly limits the amount of post-processing that is required for the Random Analysis.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

Step 4: (Continued)

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

After the analysis has finished running, you should have three modes. In Section 9 we will show you how these Normal Modes are used to generate the Solution Frequencies for the Random Analysis.

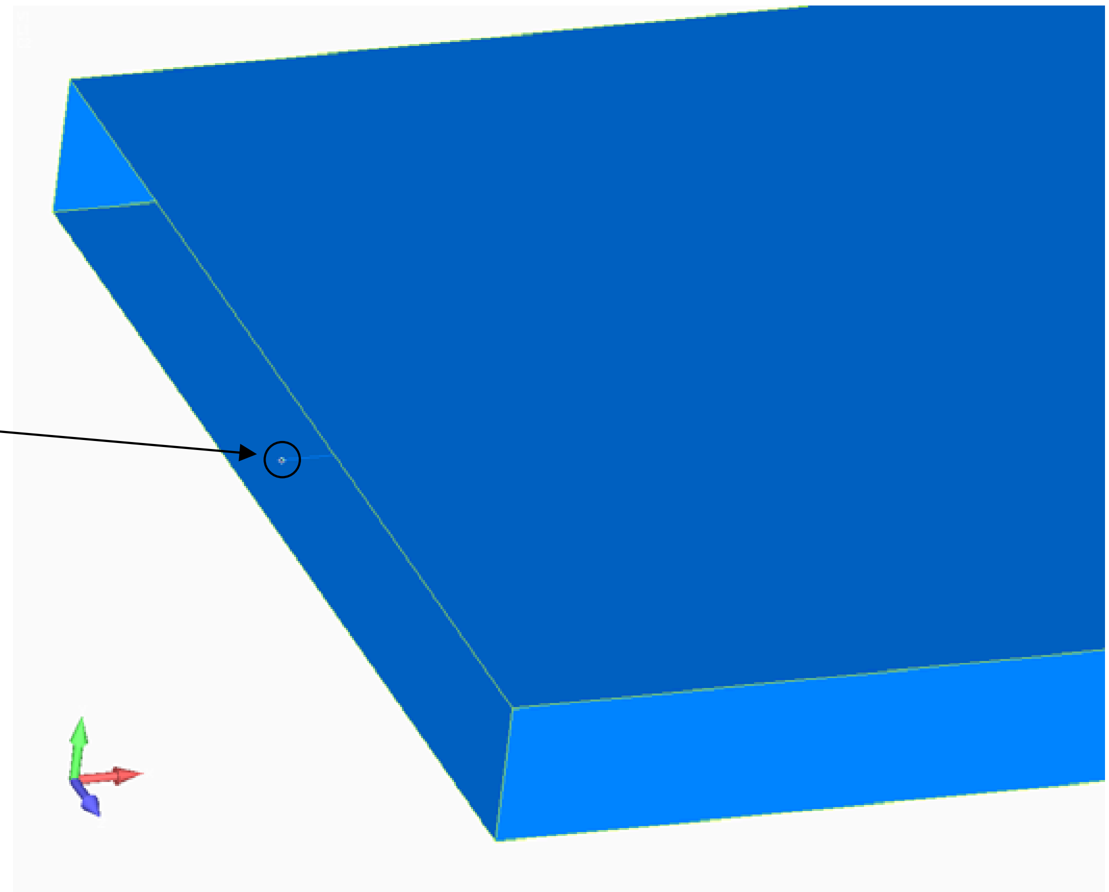




Step 5: Creating the Excitation Node and Tying it to the Model

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

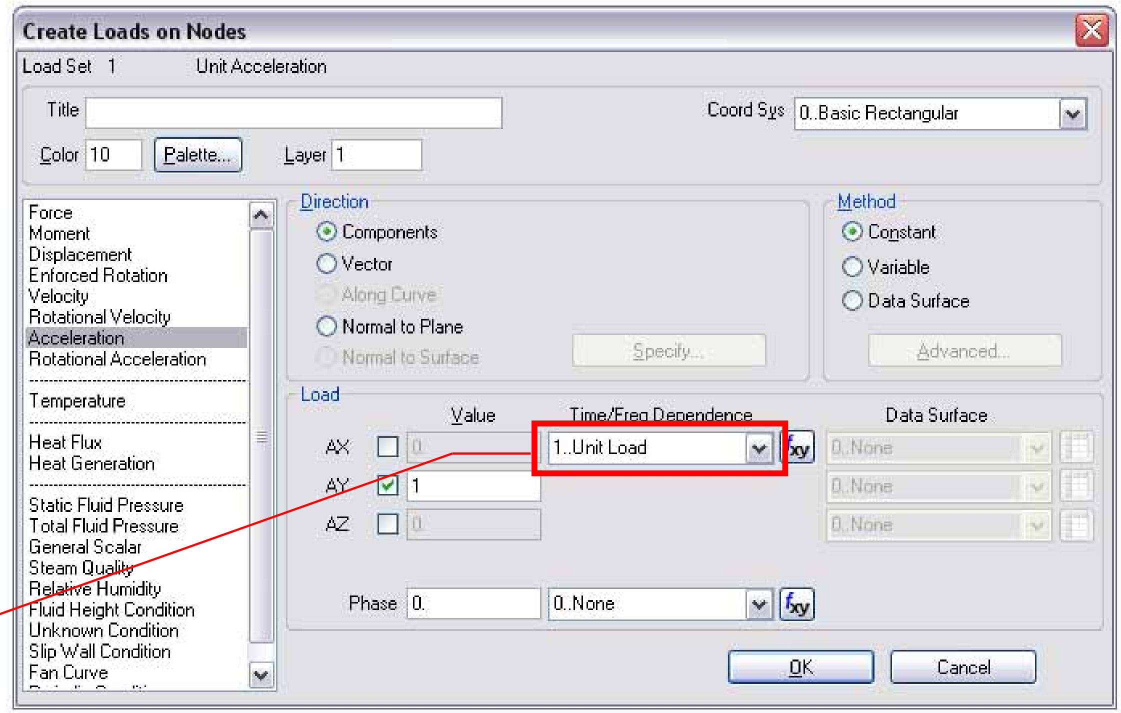
Since this is a base excitation problem, and the base of the structure consists of one node, it is that node to which we will apply our acceleration. In the case where the base of the structure is not one node, a rigid link approach can be used.



Step 6: Loading the Model

An acceleration load must be given to the base node in the direction of the excitation. Since the PSD is given in $(in/s^2)^2/Hz$, we can enter a unit acceleration (1.0 in/s^2). Be sure to ***uncheck*** the directions in which the acceleration will not act.

The Loading Function created in step 1 must also be specified in the **Time/Freq Dependence** field.



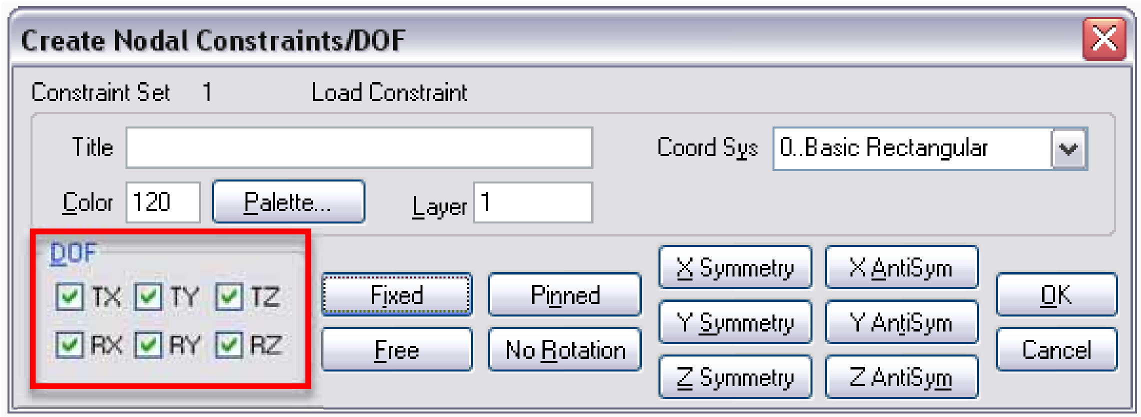
Step 7: Constraining the Model

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

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

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

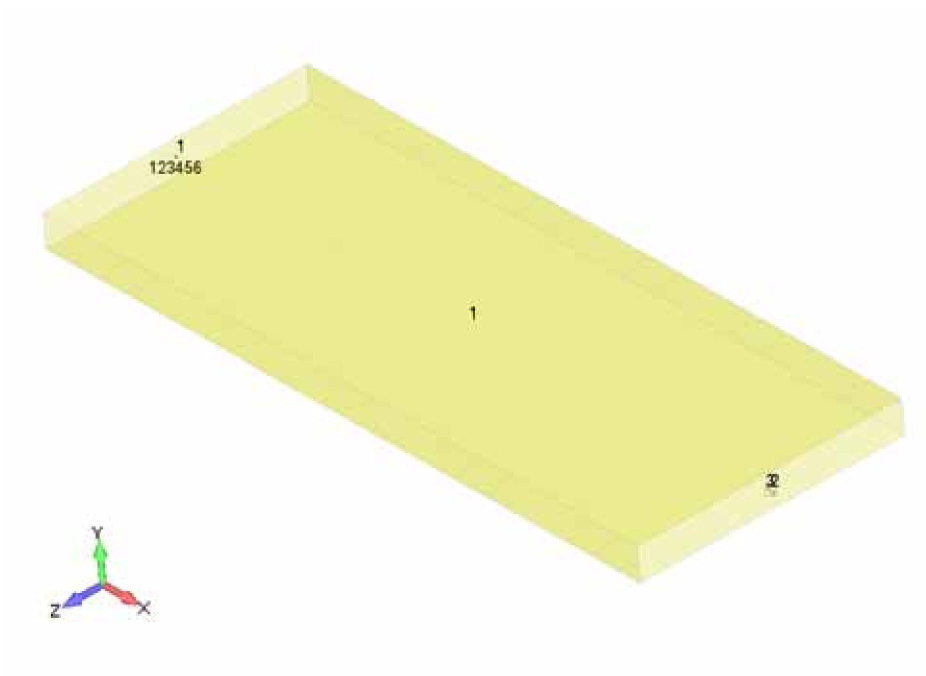
The idealization concept is that the base is fixed in the TX, TZ, RX, RY, RZ while the structure is excited in the Y-direction (i.e., there is displacement in the Y-direction).



Step 8: Specifying Groups for Nodal and Elemental Output

In a PSD analysis, output is not given for all elements and nodes by default. The user must specify, through the creation of a group or groups, which nodes and elements output data will be created for. Because our model consists of 1 element and 2 nodes, we will 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.



List a Group

Group 1 - Node and Element

Clipping ____

Rules ____

Node IDs

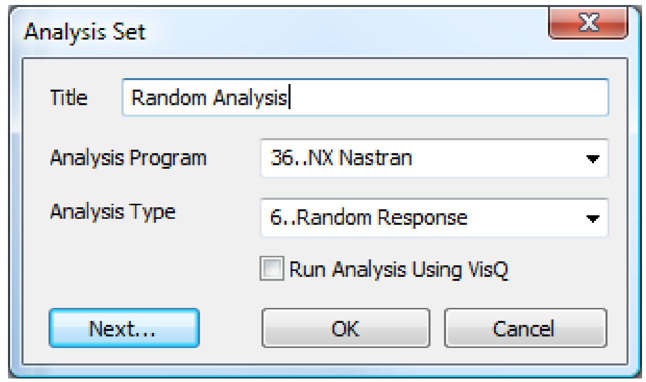
Option	Start	Stop	Increment
Add	1	0	1
Add	2	0	1

Element IDs

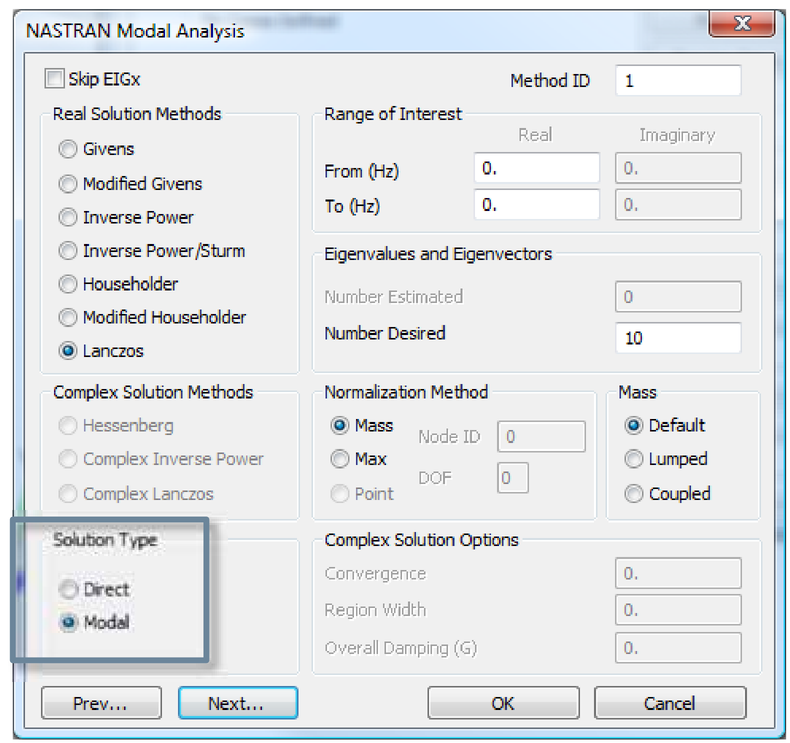
Option	Start	Stop	Increment
Add	1	0	1

Step 9: Setting up the Analysis in the Analysis Manager and Random Output Requests

To set up the analysis in the Analysis Manager, choose to create a new Analysis Set, and enter ***6..Random Response*** in the drop down menu. Then press the ***Next…*** button a few times until you arrive at this screen shown at the bottom right.

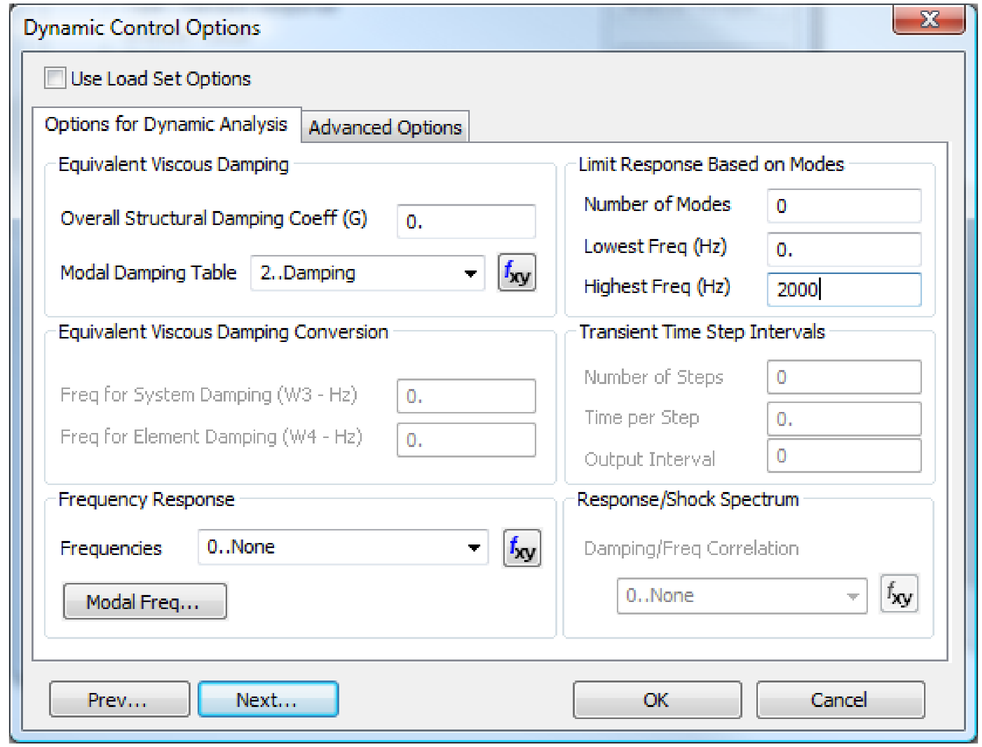


This is one of the first major decisions that one gets to make in a Random Analysis – whether to generated the Solution Frequencies using the Direct or the Modal method. If you have questions on which method is best suited for your model – take a look at the NX Nastran Basic Dynamic Analysis User's Guide, Chapter 5.4 ***Modal Versus Direct Frequency Response***

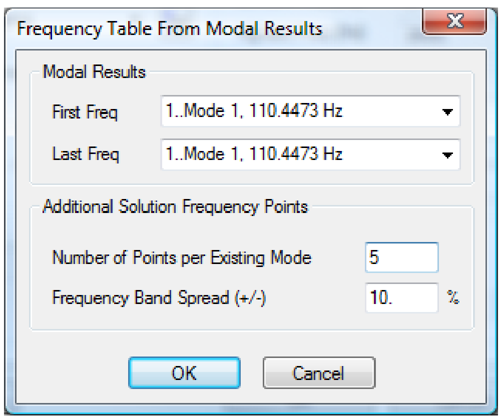


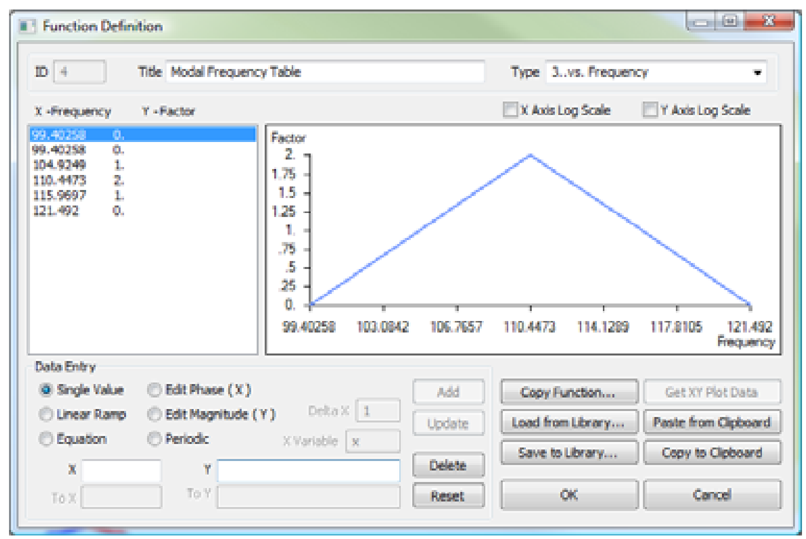
Step 9: (Continued)

The next dialog box to work with is the Dynamic Control Options. It is within this dialog box where one specifies Modal Damping, the number of modes to use in generating the Solution Frequencies (Limit Response Based on Modes), and the actual Frequency Response or Solution Frequencies.



When you push the Modal Freq… button you will get the screen shown below. We are only using the first mode to match the analytical solution.





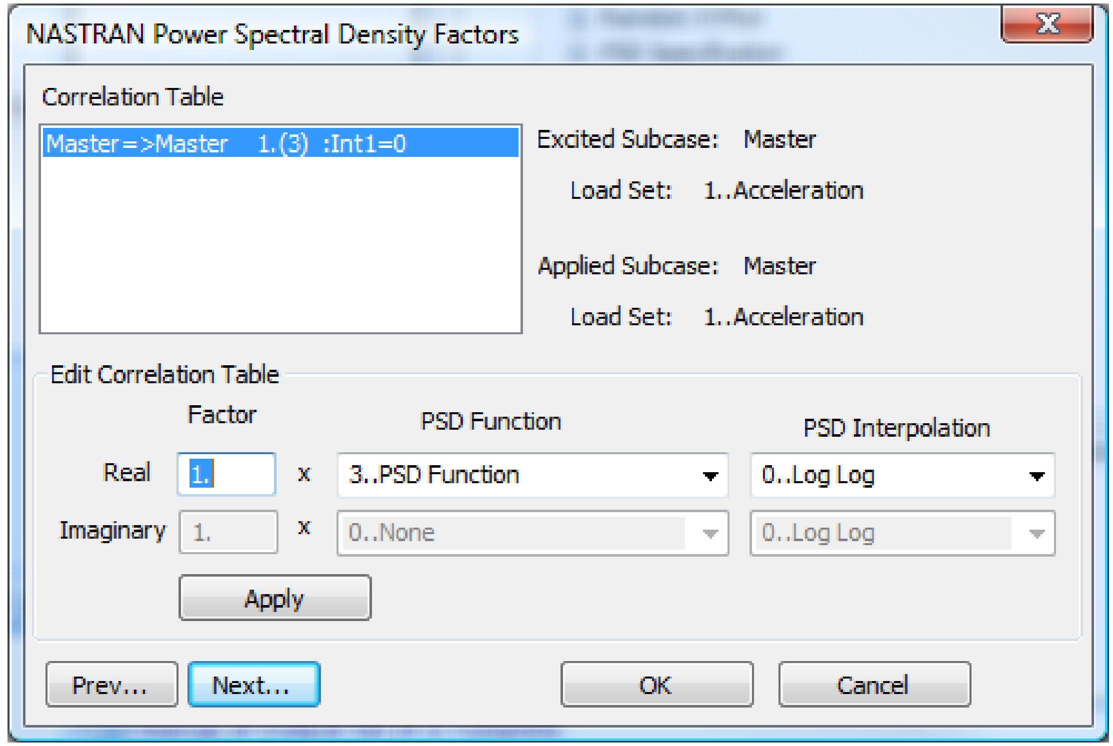
After hitting OK, it will generate a new Function that will require six Solutions to be used in the calculation of the PSD response.

Step 9: (Continued)

This dialog box is buried at the end of the Output Requests. Since the subsequent section of Step 9 is dedicated to just managing PSD Output Requests, this little box is discussed at this juncture.

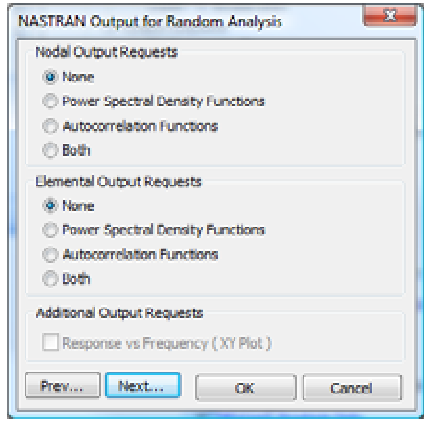
This is where you apply the PSD Function to the analysis. For convenience, one can directly scale the PSD Function.

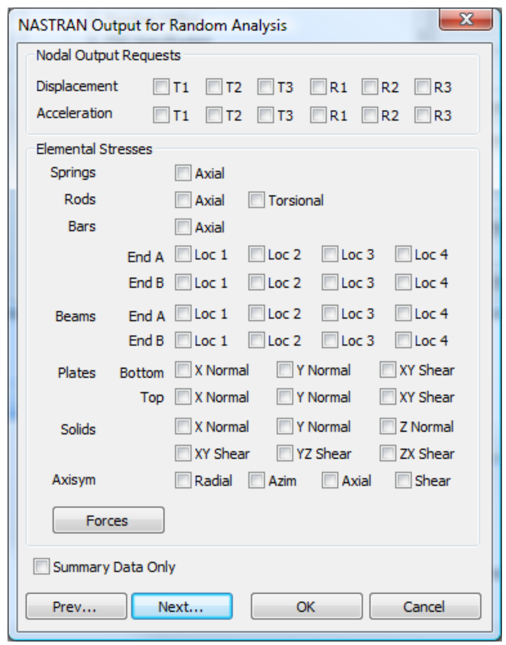
These settings are used for all the subsequent Output Requests.

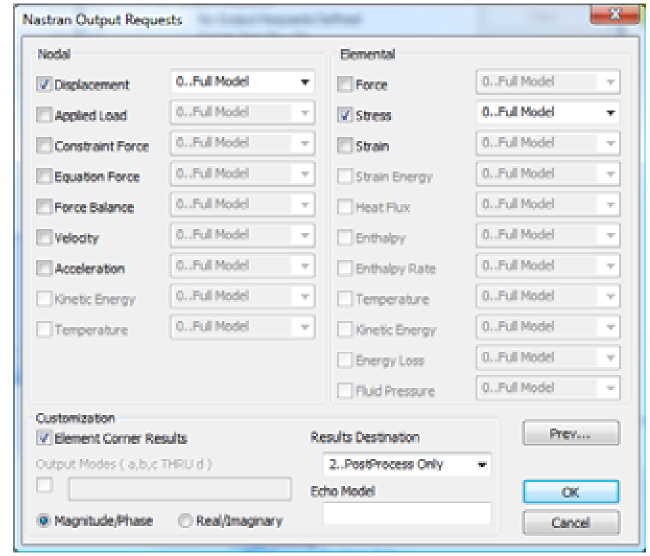


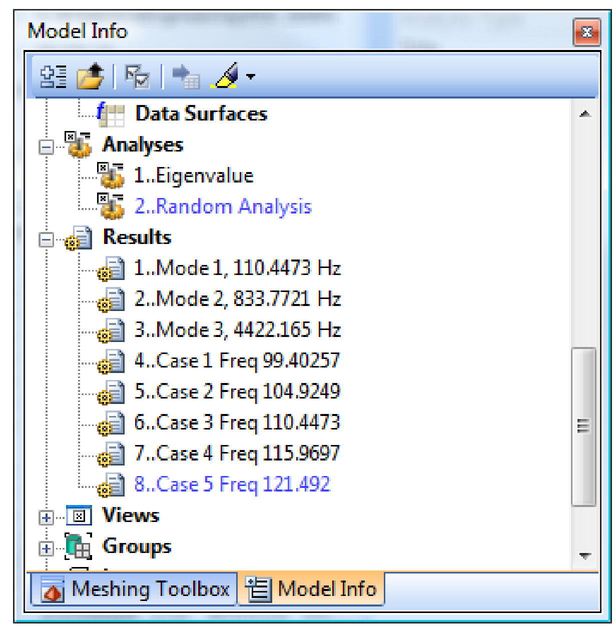
Step 9: (Random Output Requests)

Every PSD analysis is based on the post-processing of Modal Frequency results. When you submit a Random Analysis to NX Nastran, it calculates the Normal Modes and then performs a Modal Frequency Analysis at each of the requested Solution Frequencies using the applied acceleration load. As a default, no matter what you request, NX Nastran will dump out these Modal Frequency Solutions when ever you request some sort of output. This can be a bit of a problem if you have large model. There are two options: (i) Use the same PSD Node/Element group(s) under the Nastran Ouput Request dialog screen (upper right-hand) or (ii) Use the XDB Results Destination (more later…).







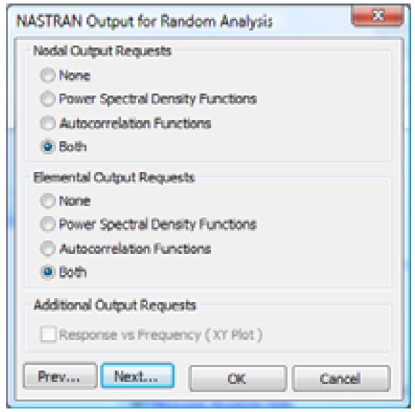


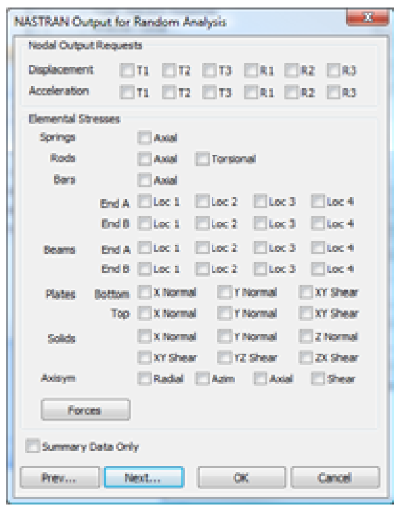
Step 9: (Random Output Requests)

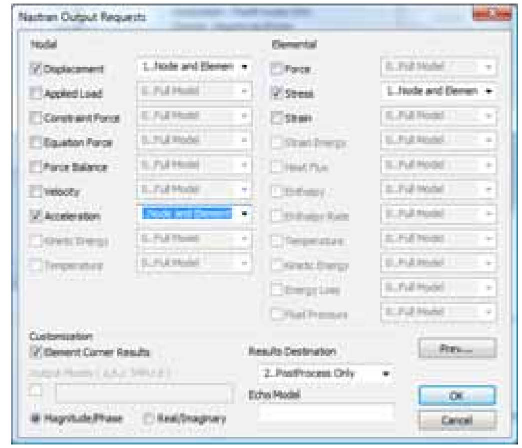
Here is one way to request Random Ouput. The first screen (upper left-hand) you request Both under Nodal and Elemental Output Requests and nothing under the next screen. On the last Nastran Ouput Request screen (upper right-hand), you make your global requests for data (in this case, I'm requesting Displacement, Acceleration and Stress data).

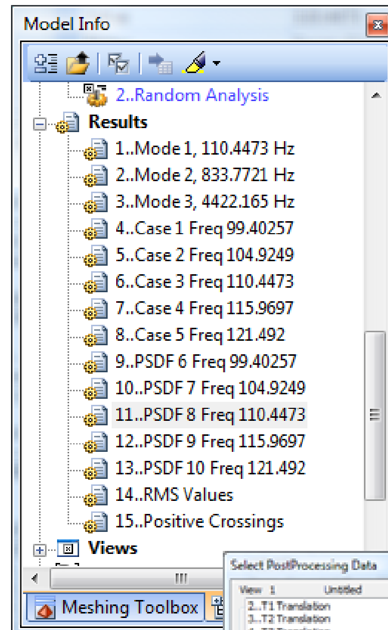
Your output are the Modal Frequency Solutions and then the PSD data. You also get the final summary data for RMS values of displacement, acceleration and stress and then the Positive Crossing (RMS frequency of the above request items).

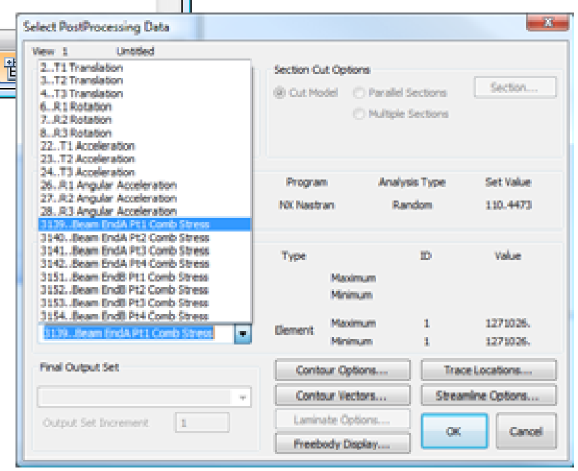
Very useful but you get ALL the displacement data (T1, T2,T3, etc) and likewise all the other data. This is the shotgun approach and again, when dealing with a large model could be a problem.







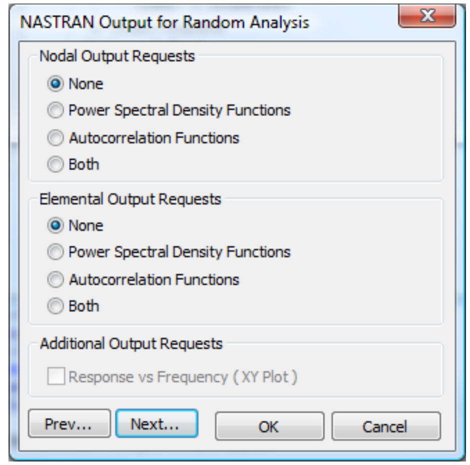


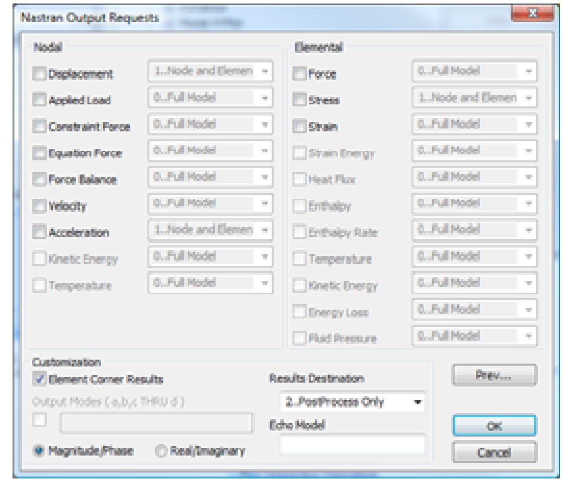


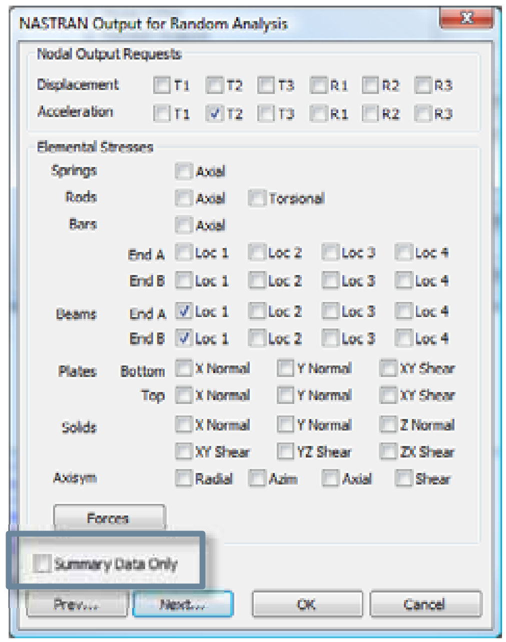
Step 9: (Random Output Requests)

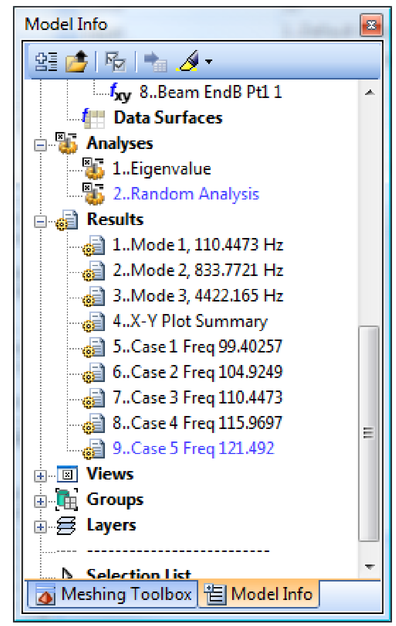
This is close to one of the most efficient ways to request output data. Of course, you are still stuck with the Modal Frequency Solutions, but your data set is smaller.

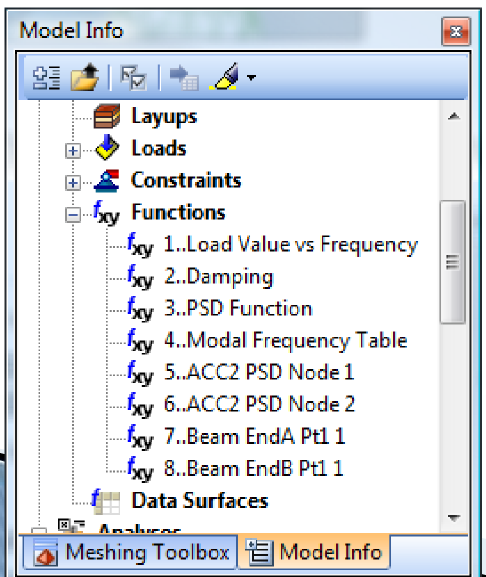
When you don't check the Summary Data Only, NX Nastran creates Functions that allow you to plot the PSD quantities as a function of frequency. This will be used later on to show how to interpret or to check your results.

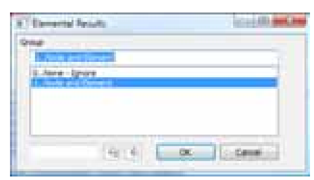












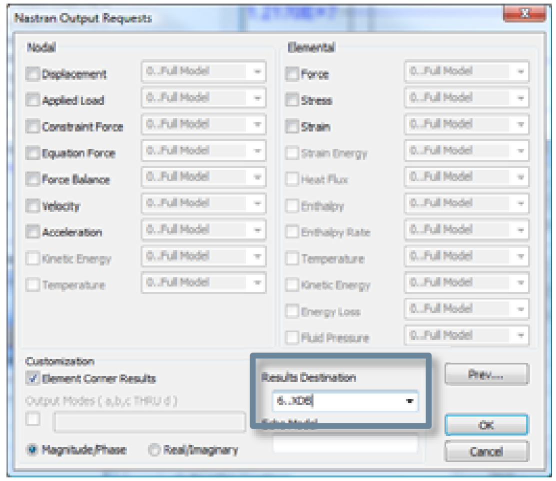
Step 9: (Random Output Requests)

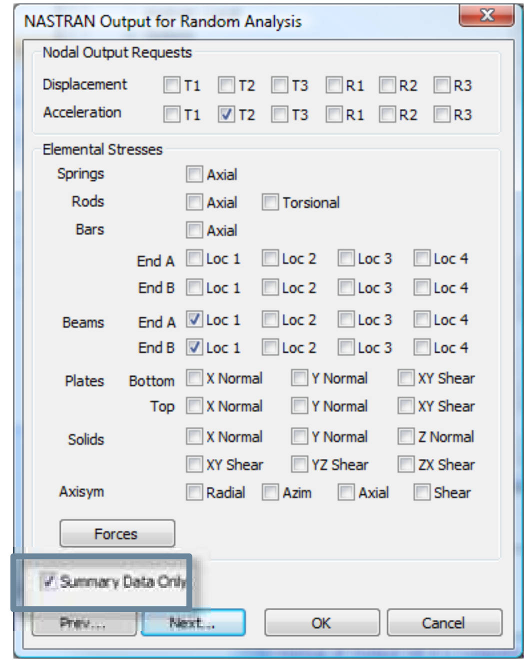
Okay, this is the ultimate in Random Output Request efficiency. We will use the NX Nastran XDB option to only pull in the final X-Y Plot Summary Results for only the exact results that we want. This is what I use for my +500,000 node analysis models.

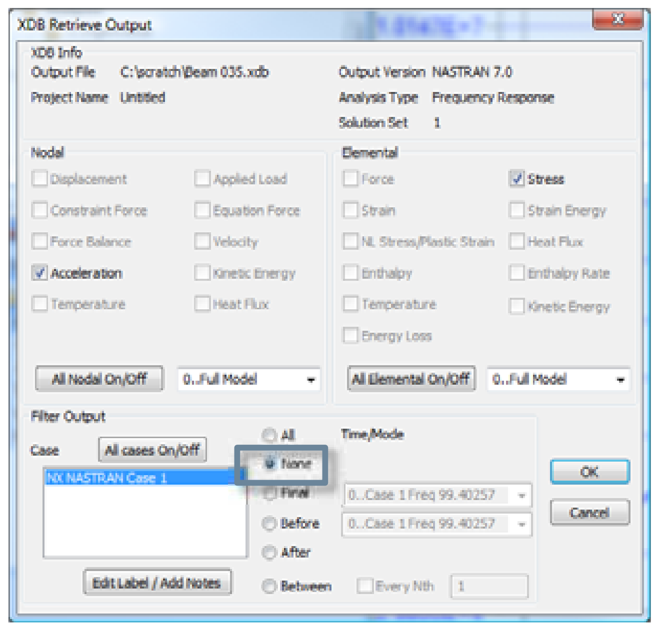
All the settings are the same from the prior setup with two exceptions: (i) Check the Summary Data Only and (ii) Under Results Destination, select Item 6..XDB.

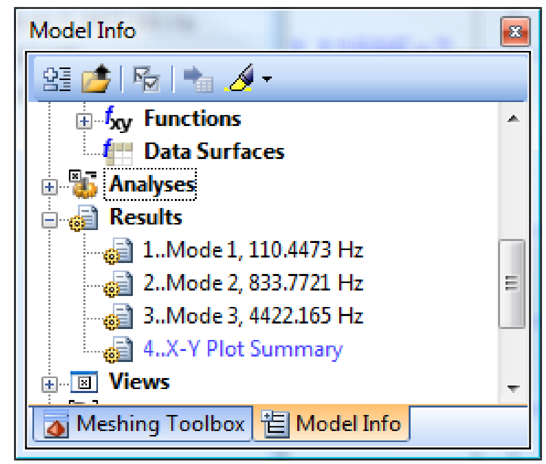
When the analysis finishes the run, you select None and Femap will import only the X-Y Plot Summary into the model.

This creates the ultimate in minimal data.







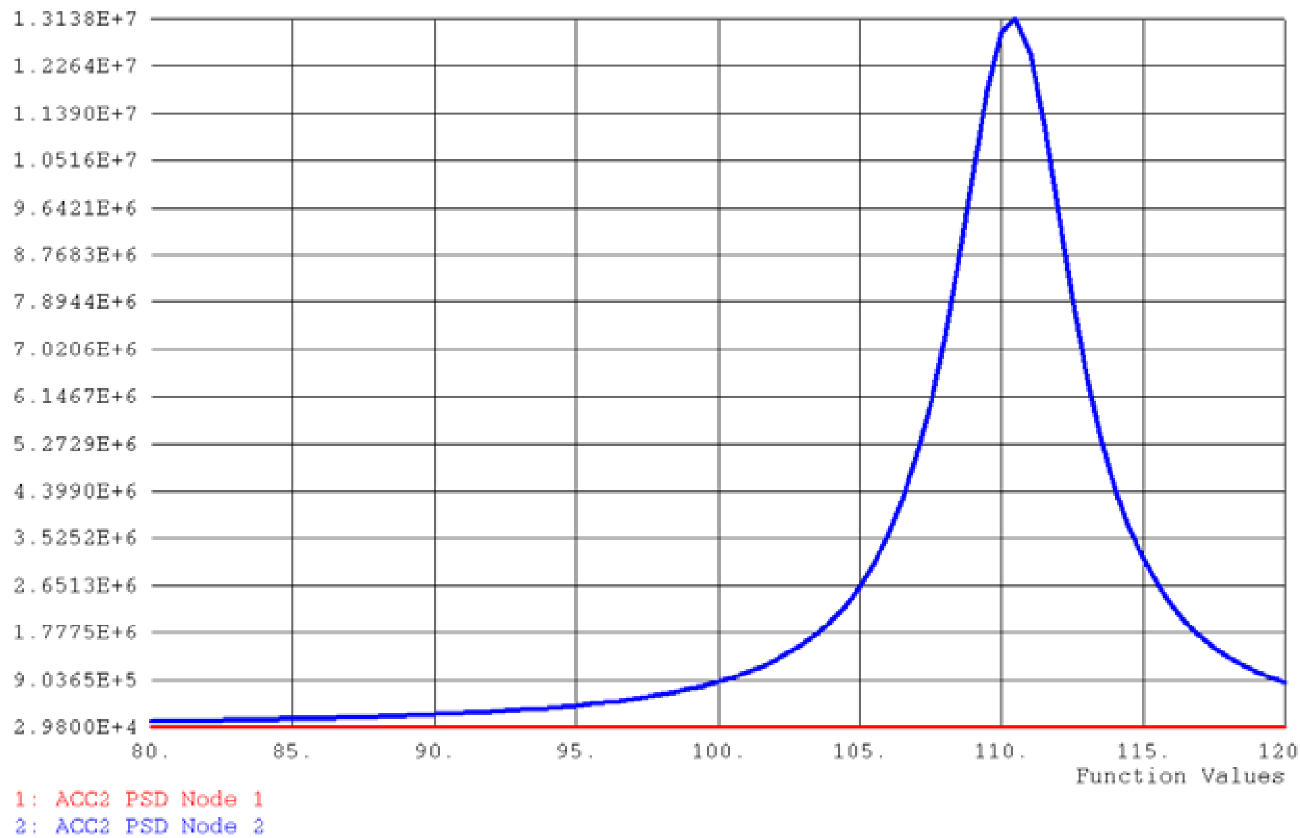


Interpreting the Output – *Analysis Check*

A plot of the Output PSD is shown on the right as well as nodal data for the end of the beam.

It can be seen that the FEA results match the analytical results on page 8 very closely.

Additionally as a check, the base node output PSD matches the input PSD as it should, with a value of 29,800 $(in/s^2)^2/Hz$.




```
Node 2
 Output Set 82 - X-Y Plot Summary
  Output Vector 1571    - T2 Acceleration RMS      = 9876.03
  Output Vector 1572    - T2 Acceleration +Cross   = 109.223
  Output Vector 1573    - T2 Acceleration Xmin     = 80.
  Output Vector 1574    - T2 Acceleration Xmax     = 120.
  Output Vector 1575    - T2 Acceleration Ymin     = 131100.
  Output Vector 1576    - T2 Acceleration Ymax     = 13140000.
  Output Vector 1577    - T2 Acceleration Ymin X   = 80.
  Output Vector 1578    - T2 Acceleration Ymax X   = 110.5
```

Interpreting the Output – *RMS Stress Check*

Output for the beam element at End A (fixed end), point 1 (top corner) of the beam is shown on the right. Since the beam is a rectangle, the stress recovered at all four corners are the same. This output is found in the X-Y Plot Summary.

This output shows the RMS stress as 3,003 psi. This RMS stress represents how much stress the beam will experience 68.3% of the time. This is well below the yield stress for aluminum. Because this is a vibration problem, we are also concerned about fatigue.

```
Element 1
 Output Set 82 - X-Y Plot Summary
   Output Vector 87241    - Beam EndA Pt1 RMS          = 3003.39
   Output Vector 87242    - Beam EndA Pt1 +Cross       = 110.375
   Output Vector 87243    - Beam EndA Pt1 Xmin         = 80.
   Output Vector 87244    - Beam EndA Pt1 Xmax         = 120.
   Output Vector 87245    - Beam EndA Pt1 Ymin         = 353.9
   Output Vector 87246    - Beam EndA Pt1 Ymax         = 1283000.
   Output Vector 87247    - Beam EndA Pt1 Ymin X       = 80.
   Output Vector 87248    - Beam EndA Pt1 Ymax X       = 110.5
```

Interpreting the Output – *Fatigue Analysis using RMS +Cross*

We will now use the number of positive crossings (or **Beam EndA Pt1 +Cross**) to calculate fatigue life.

The **Beam EndA Pt1 +Cross** output vector gives us a value of 110.4 Hz. The number of positive crossings is a measure of the apparent frequency of the response. Given a white noise PSD input of 0.2 G^2/Hz, the beam will experience a fully reversible stress of 3,003 psi at a frequency of 110.4 Hz.

Statistically speaking, this stress value represents the 1σ value and will be experienced 68.3% of the time. A 2σ of 2*3,003 or 6,006 psi will be experienced 27.1% of the time and a 3σ value of 9,009 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.

All three σ level stresses fall into the infinite life range on a fatigue curve for aluminum. To demonstrate how to treat the problem if this is not the case, let us assume that there is a small hole in the beam which causes a stress concentration factor of 3. This would put the 1σ stress level at 9,009 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:

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

Interpreting the Output – *Fatigue Analysis using RMS +Cross*

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.

Point A	1σ	2σ	3σ
Stress	9,009 psi	18,018 psi	27,027 psi
No. of Cycles to Fail	infinite	6.0E5 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 1.3E6 cycles. If the beam is vibrating at a frequency (number of positive crossings) of 110.4 Hz, then it will take the beam approximately 11,775 seconds or a bit over 3 hours to fail.

$$1=\frac{0.6831\cdot n}{\infty}+\frac{0.271\cdot n}{6.0E5}+\frac{0.0433\cdot n}{14.0E4}$$

As long as the beam is exposed to the while noise vibration for under 3 hours, it should not fail.

Conclusions

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 and 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."

George Laird, Ph.D., P.E.
Email: George.Laird@PredictiveEngineering.com
Phone: 503.206.5571

FINITE ELEMENT ANALYSIS
Predictive Engineering

Random Vibration Analysis
Application Note

Introduction

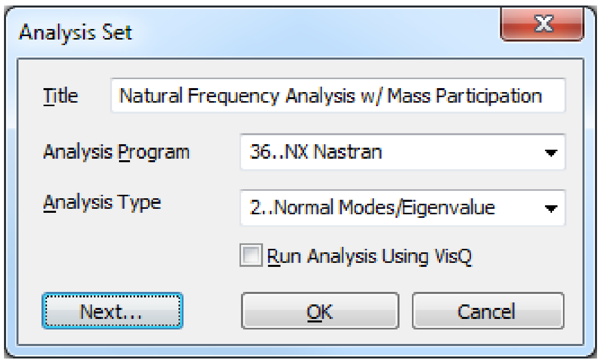
Random vibration analysis can be very complex and time consuming. A solid foundation in the theory of the subject cannot be replaced by any recipe or cookie cutter workflow; however, there are a few tips which can help guide the process and save time.

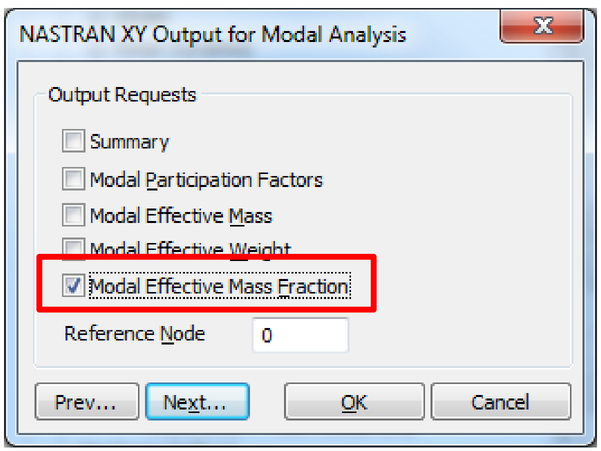
This application note will build upon the previous tutorial* by giving a real world example. The electronics enclosure shown here will be analyzed. It contains a housing and multiple printed circuit boards connected together.

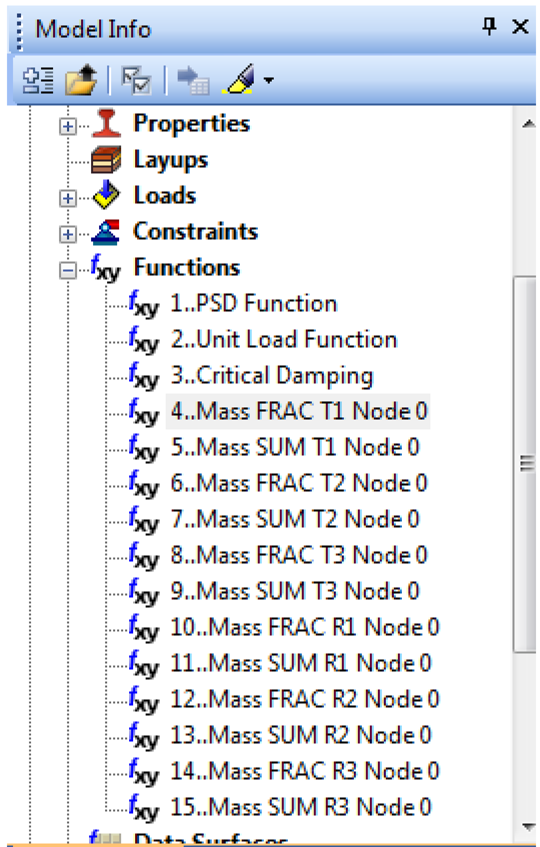
Significant time can be saved by inspecting how much of the mass is participating in the normal modes.

Setting up the model for random vibration analysis is covered in the previous tutorial and is not covered in detail here.

* http://predictiveengineering.com/downloads/Tutorials/PSD_Tutorial_2012.pdf





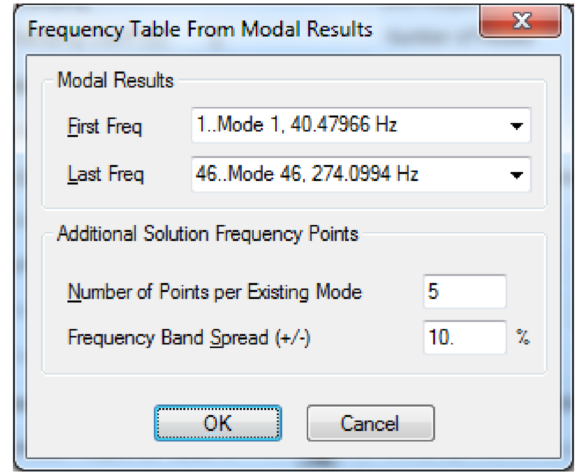


Just like the previous tutorial a *Normal Modes/Eigenvalue* analysis is run; however, this time the *Modal Effective Mass Fraction* output is requested. With this output request, when the analysis run is complete, Femap will add 12 functions. These functions will give insight into how much of the mass is participating in each direction for each of the modes.

The *Mass SUM* is a running total of the mass participation and the *Mass FRAC* is the fraction of the mass that participates at the given mode.

In random vibration analysis a typical range of interest is from 20 or 2,000 Hz. This enclosure, with all it’s printed circuit boards, has 386 normal modes in this range.

Inspecting the functions will indicate which modes are of interest and which can be ignored.

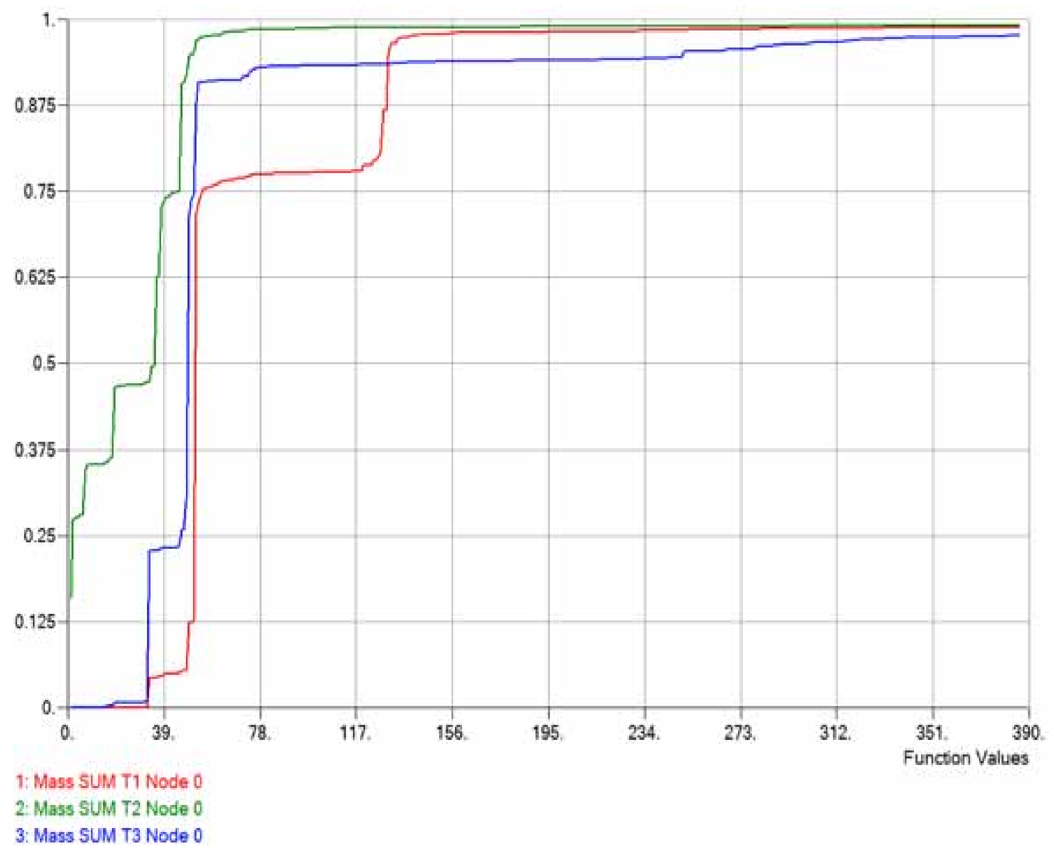


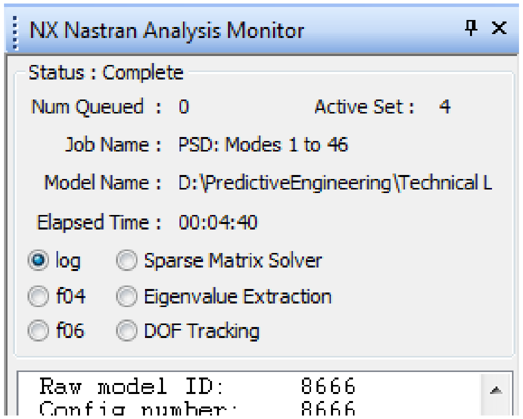
Remember, given an input PSD Function, an output response can be calculated by using the systems transfer function:

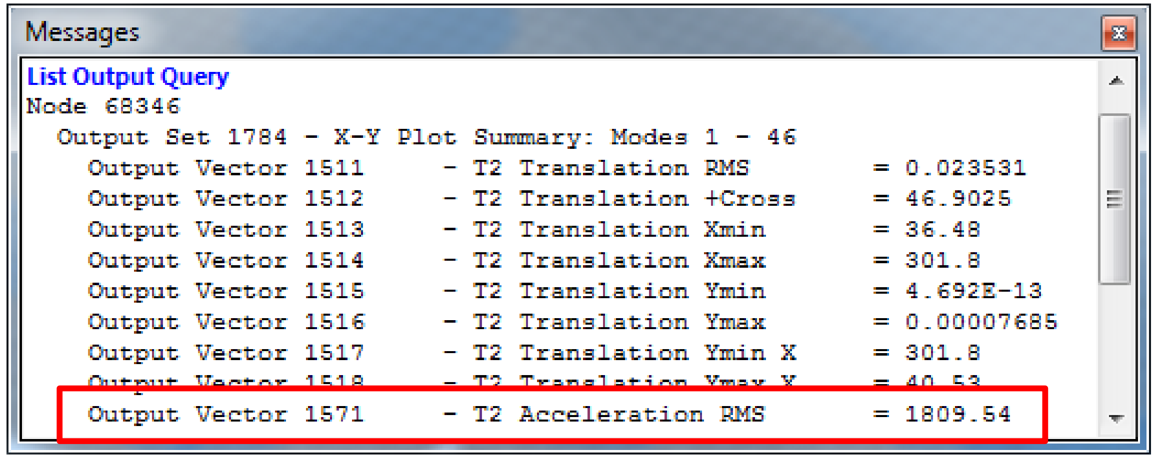
$$PSD_{out} = |g(\omega)|^2 PSD_{in}$$

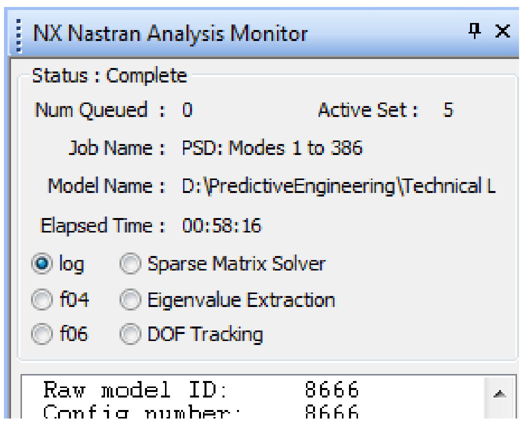
And that NX Nastran performs a frequency response analysis based on the table you create to obtain the system transfer function, $g(\omega)$.

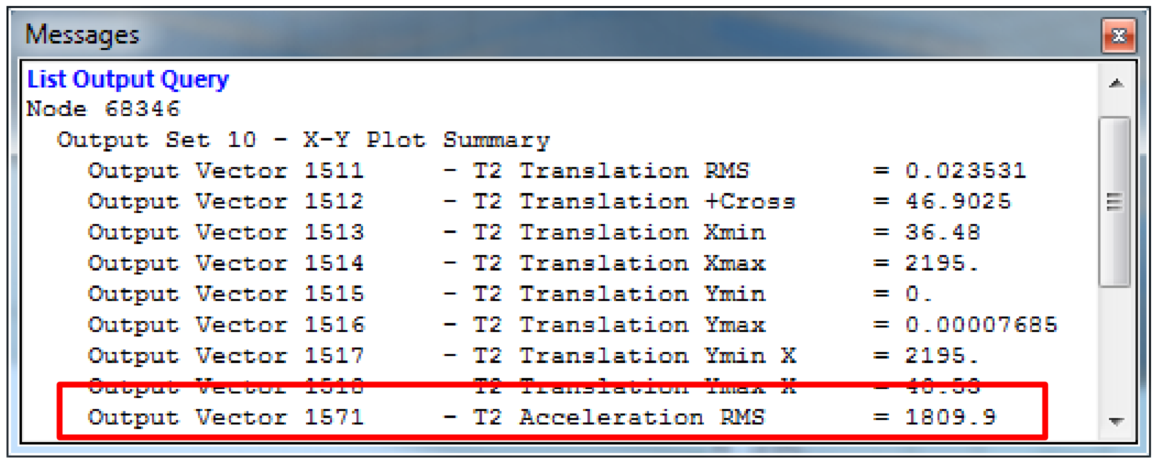
From the *Mass SUM,* it can be seen that the majority of the mass is already participating in the first 46 modes. All the modes after 46 have little mass involved and output for those modes can be safely ignored. Including only the first 46 modes in the Frequency table will greatly reduce the time required to run the analysis because the frequency response will only need to be performed at 46 frequencies not all 386.



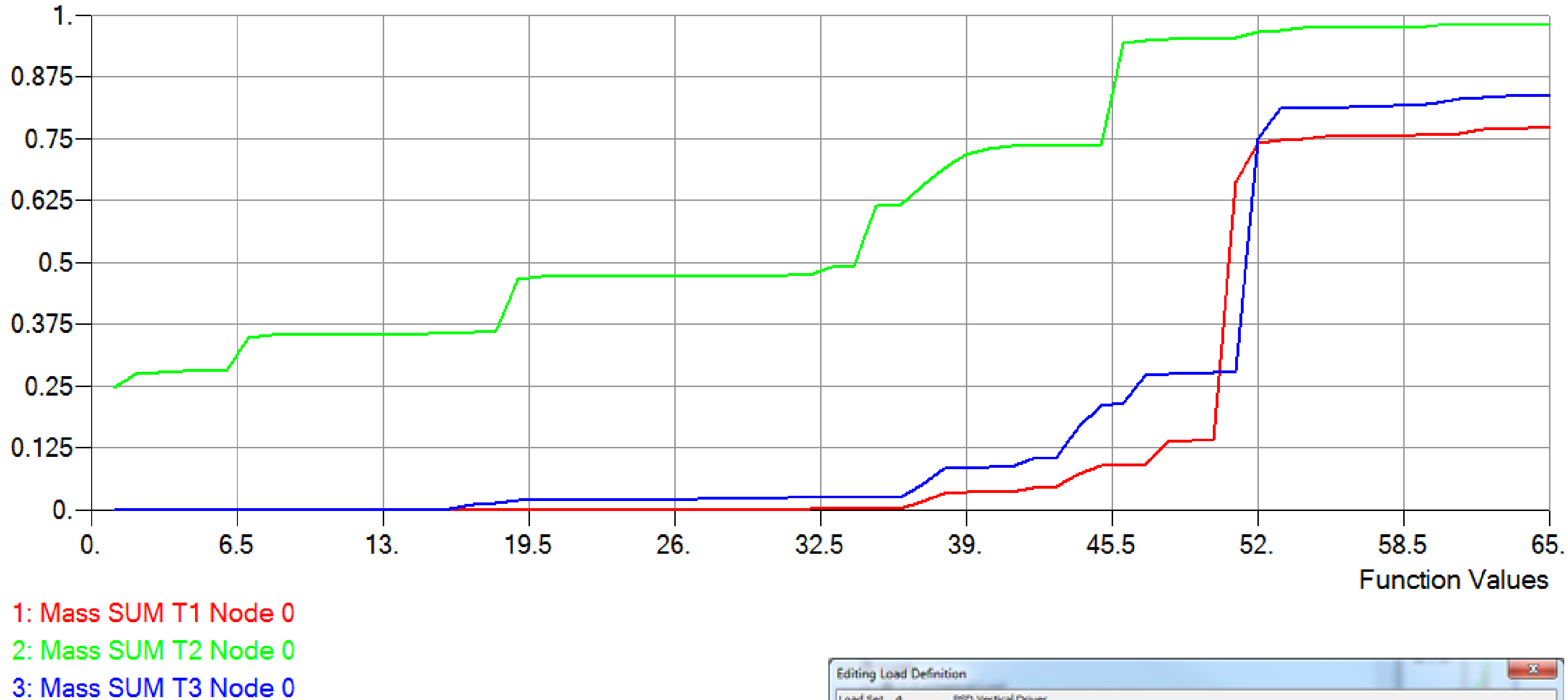




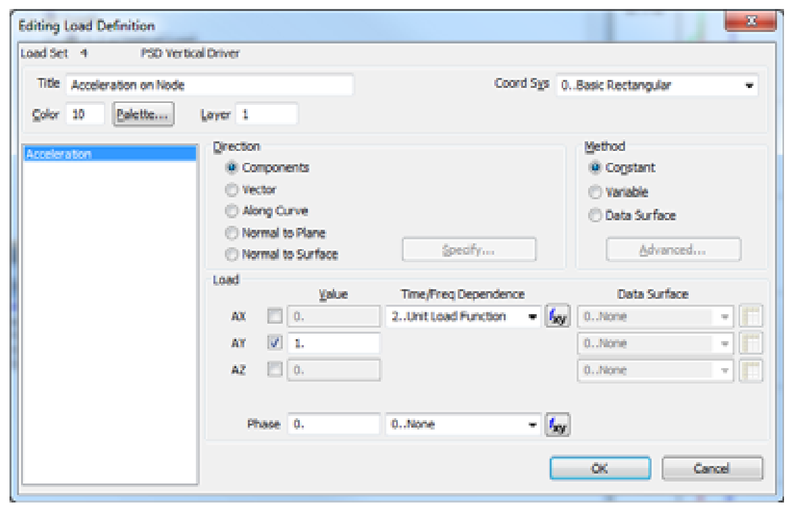




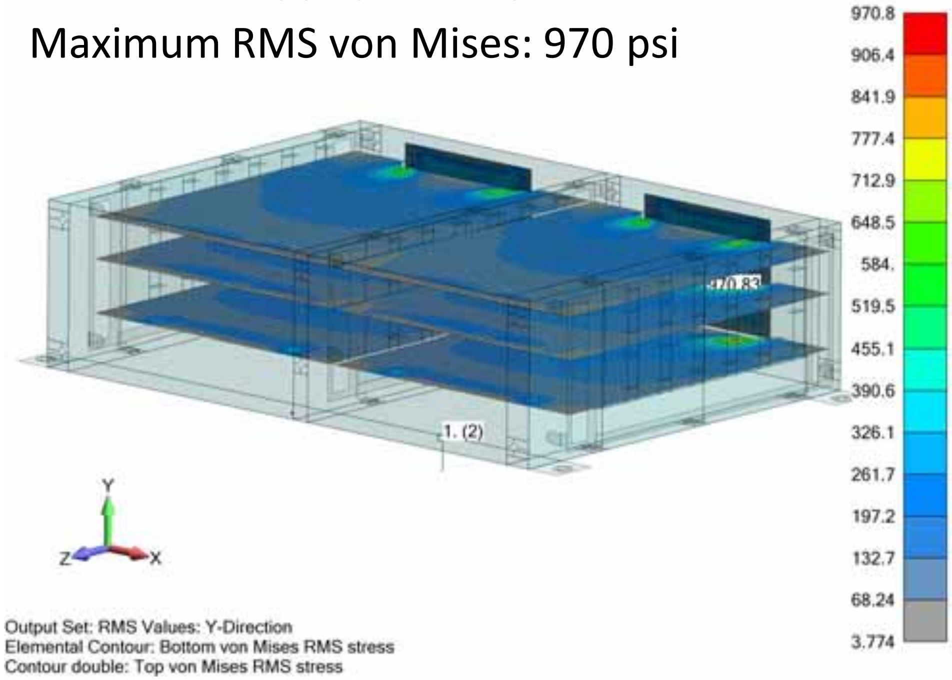
Two runs where preformed to show the time savings . One with the frequency response table set from modes 1 to 46 and the other from modes 1 to 386. The RMS acceleration response is basically the same in both runs but the time required is significantly lower for the first run – 5 vs. 58 minutes.



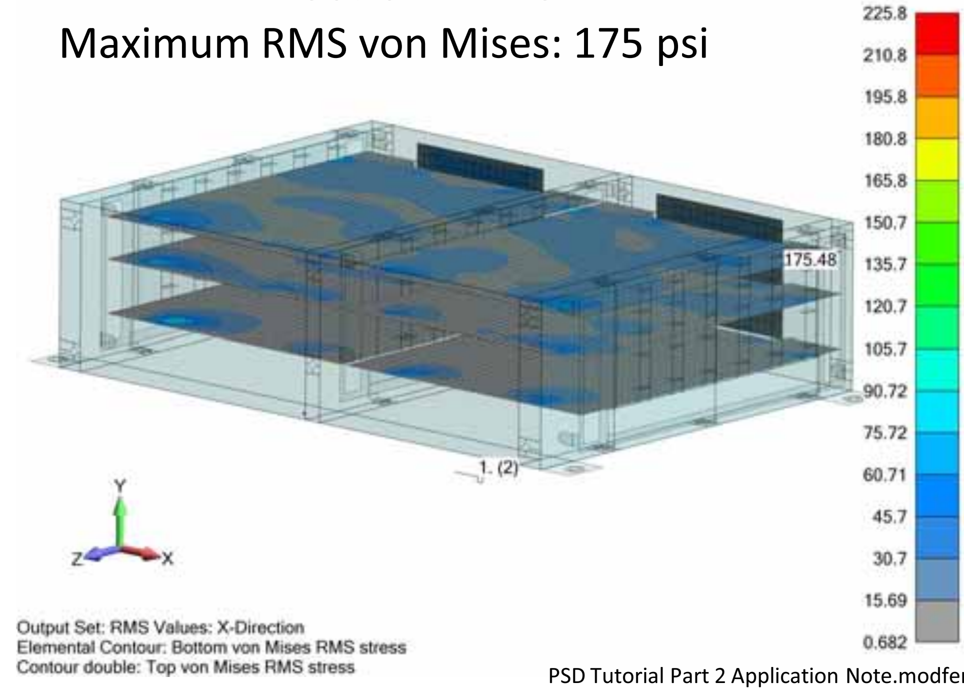
Taking a closer look at the first 65 modes shows that the majority of the mass is participating in the y-direction. This indicates that the other two directions are of little interest.



Y-Direction Driver
Maximum RMS von Mises: 970 psi



X-Direction Driver
Maximum RMS von Mises: 175 psi



PSD Tutorial Part 2 Application Note.modfem

To illustrate this point, the model was run twice: with the driver in the y-direction, and the driver in the x-direction. Shown are contour plots of the RMS von Mises stress**. The maximum stress is 175 and 970 psi in the x and y directions, respectively. Clearly, the y-direction is of greater interest.

** The RMS von Mises stress is calculated with a custom api. Contact Predictive Engineering to learn more.

FINITE ELEMENT ANALYSIS

Predictive Engineering

www.PredictiveEngineering.com