Random Vibration Analysis in FEMAP

An Introduction to the Hows and Whys

Adrian Jensen, PE | Senior Staff Mechanical Engineer
Kyle Hamilton | Staff Mechanical Engineer

FINITE ELEMENT ANALYSIS
Predictive Engineering

LS-DYNA Sales, Support & Consulting

www.PredictiveEngineering.com

Applied CAx

Siemens PLM Software Sales & Support
CAD I CAM I CAE I Teamcenter

www.AppliedCAx.com

Table of Contents

1. 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.

2.   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 G^2/Hz.
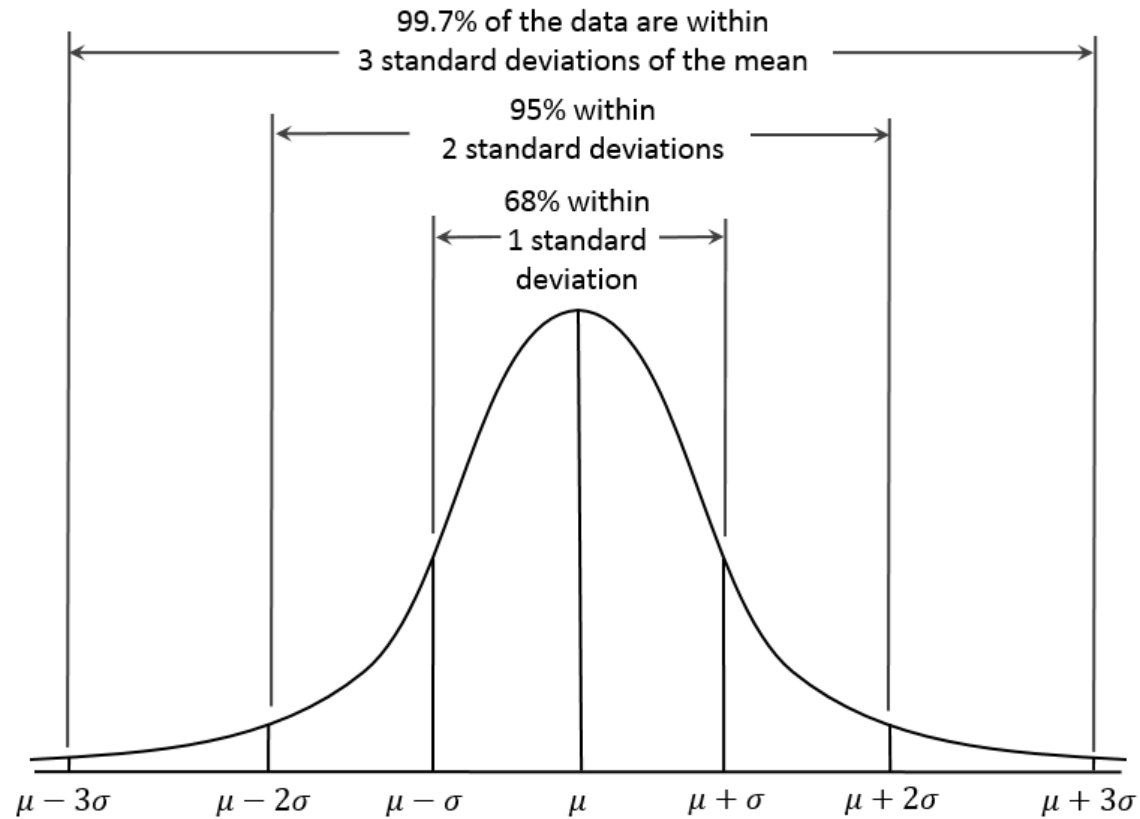
A typical power spectral density is shown below:

Frequency (Hz)	PSD (G^2/Hz)
20	0.01
100	0.05
900	0.05
1,100	0.01
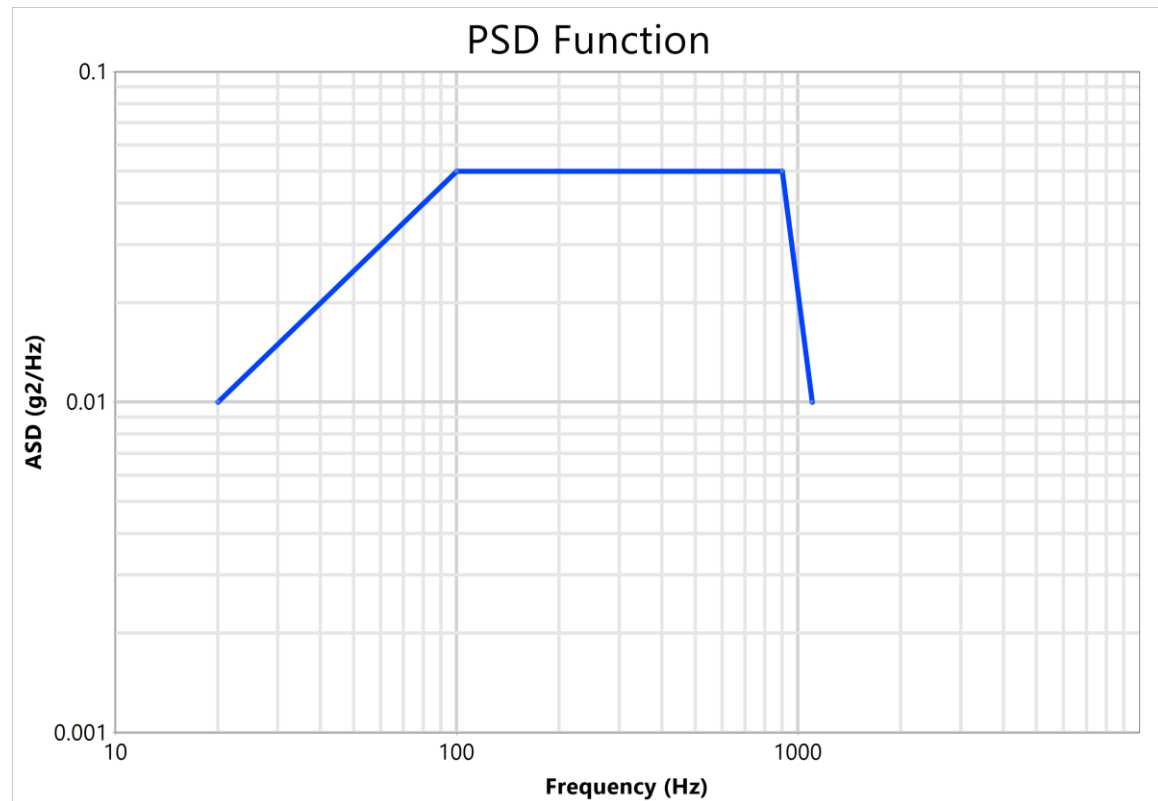
For more of the nitty gritty math details see NASA's webpage on random vibration here: https://femci.gsfc.nasa.gov/random/

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):

Image By Dan Kernler (Own work) [CC BY-SA 4.0 (http://creativecommons.org/licenses/by-sa/4.0)], via Wikimedia Commons

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.

3. 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 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. 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

4. EXAMPLE 1: CANTILEVER BEAM

4.1 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 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.

E = 10E6 psi

2 in

G

0.25 in.

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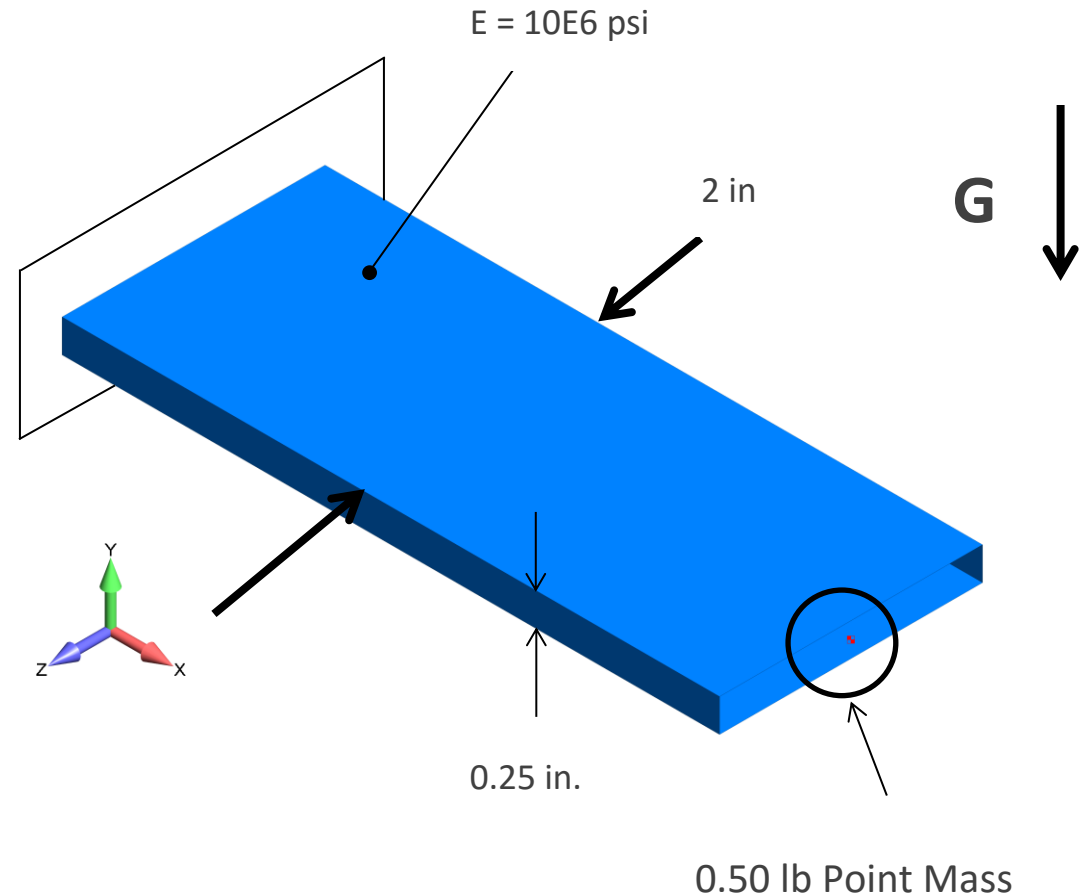
[ψ] Steinberg, Dave S. Vibration Analysis for Electronic Equipment. 2nd ed. New York: John Wiley & Sons, 1988. 226-231.

4.2 ANALYTICAL SOLUTION

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

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

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

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

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:

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

This output 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 in2/s4. This can also be verified against the FE Model.

Note: When approximating transmissibility (Q), the square root of the natural frequency should be scaled by 0.5 – 2 per Steinberg. In this case there is 100% mass participation for the first mode, thus 2 is appropriate.

4.3   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.
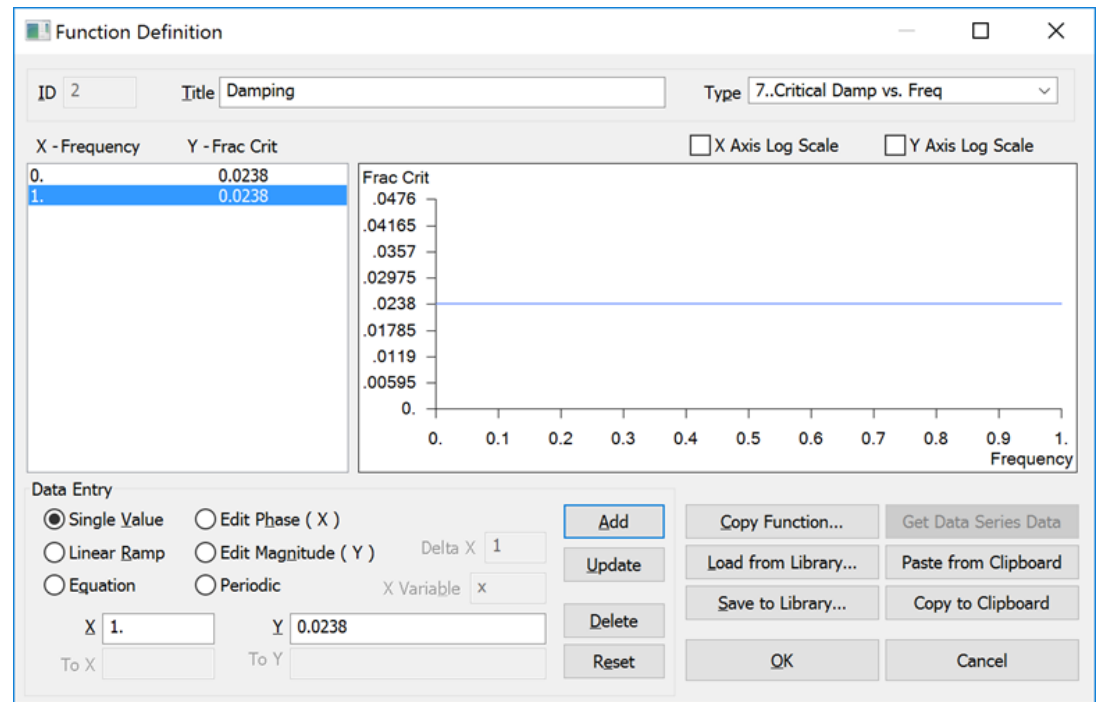
An approximation of the transmissibility of the beam is Q = 21. This value yields a critical damping ratio of 2.38%; this is what we will use.

4.4 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.

This is entered directly with no scaling. It will be scaled for the desired unit system in the Load Definition dialog (Section 4.7).

4.5 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….
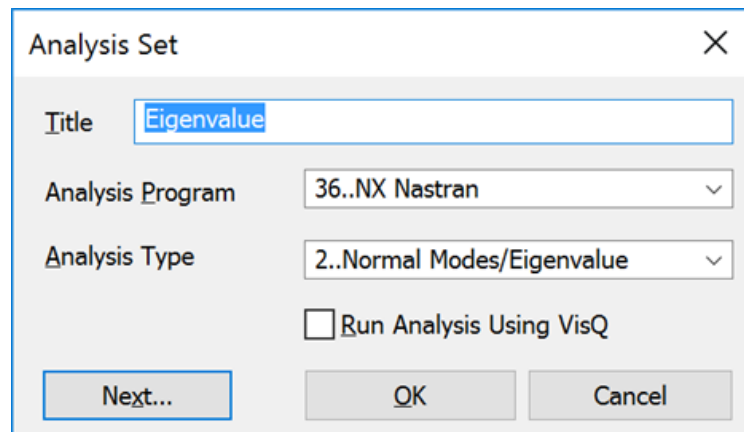
It is good practice to run the normal modes analysis first to see how the structure will behave. In this simple beam model, we have fixed the end of the beam in all six DOF. The beam is also mass-less (material density of 0.0). This was done to allow us to exactly match the analytical solution.

After the analysis has finished running, you should have three modes. In Section 4.10 we will show you how these Normal Modes are used to generate the Solution Frequencies for the Random Analysis.
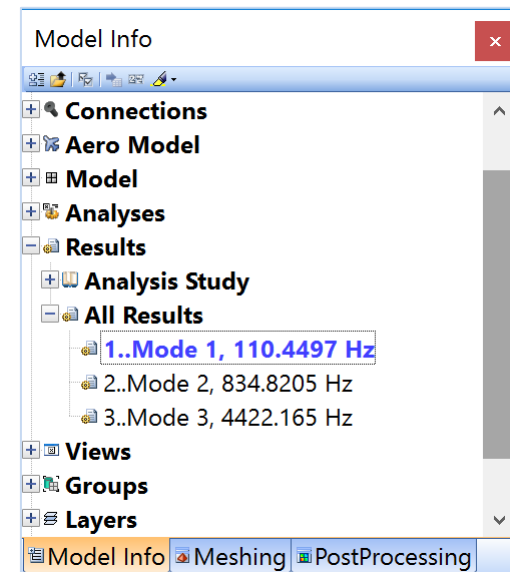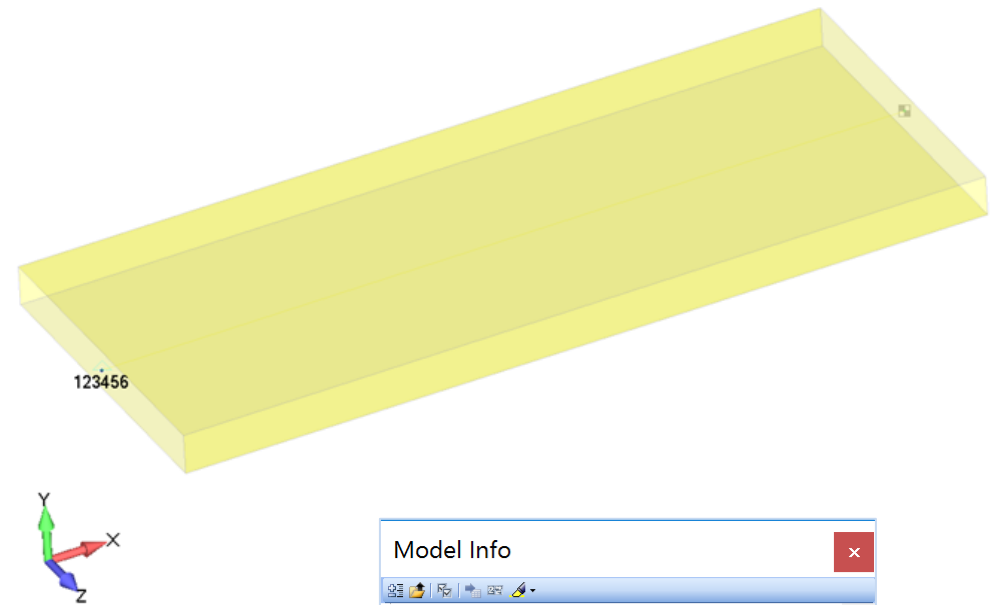
4.6 CREATING THE EXCITATION NODE AND TYING IT TO THE MODEL

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.
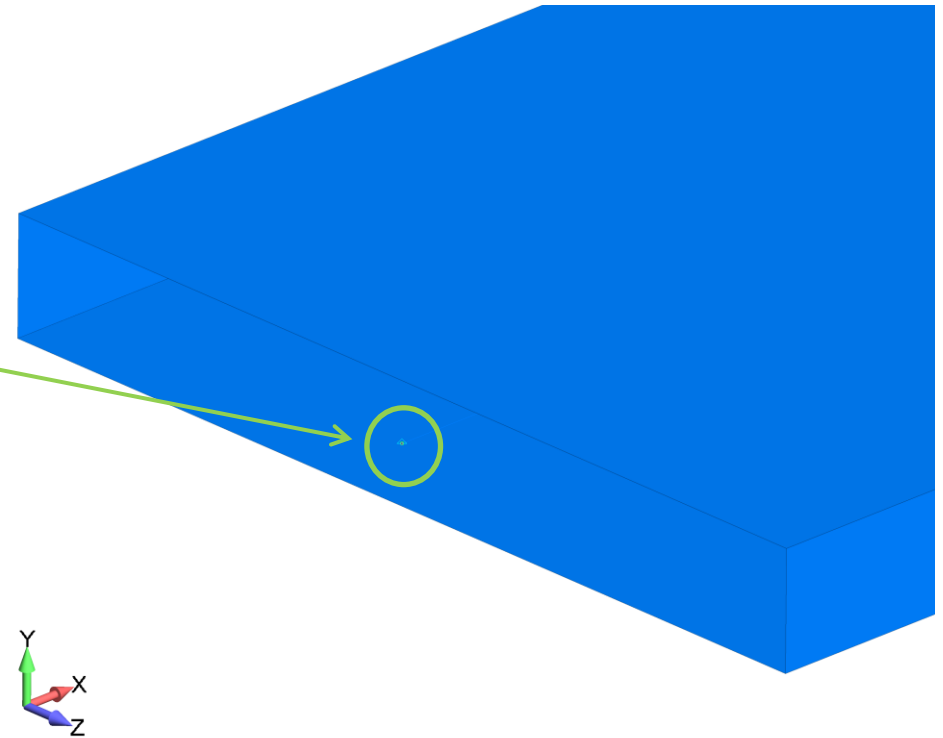
4.7   LOADING THE MODEL

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

4.8   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).
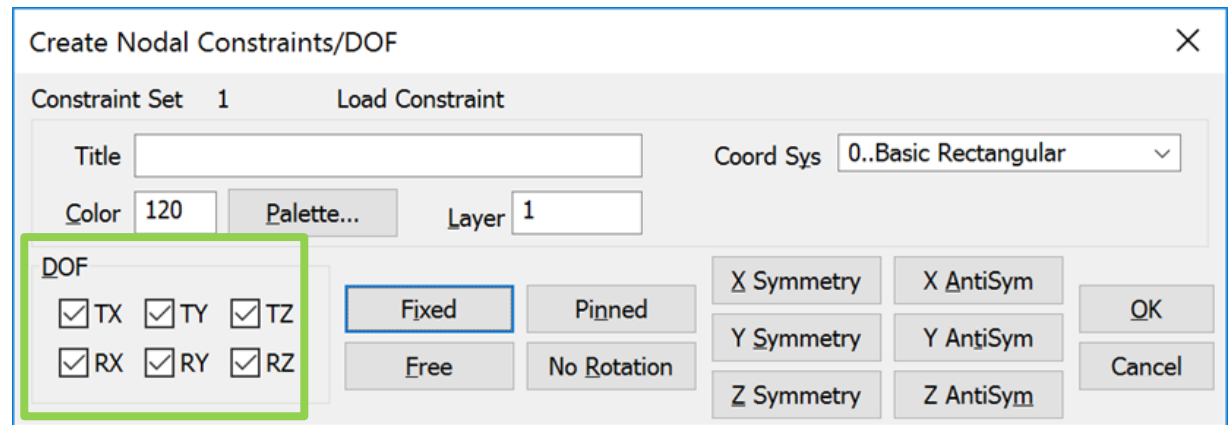
4.9   SPECIFYING GROUPS FOR NODAL AND ELEMENTAL OUTPUT

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.
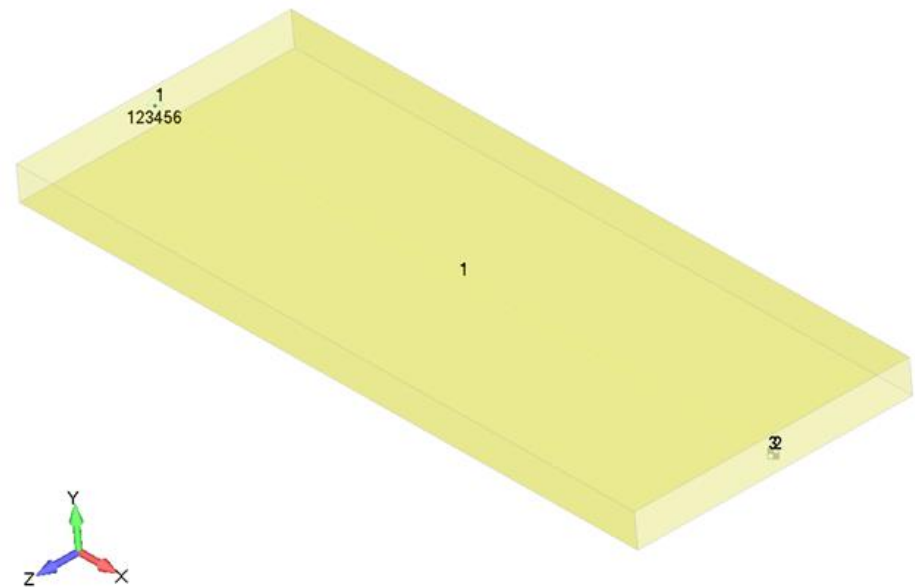
4.10 CREATING AN ANALYSIS SET – SIMPLE PSD

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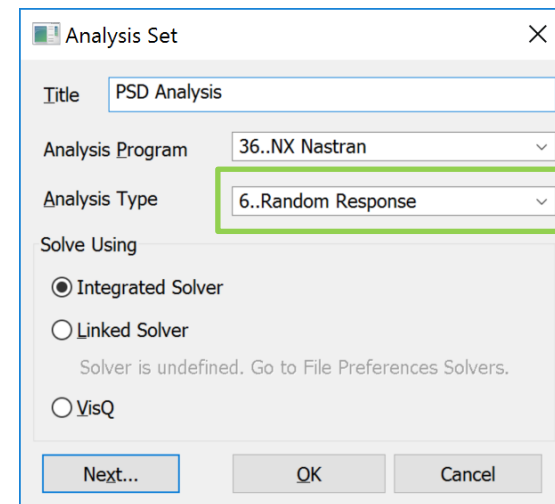
Keep pressing **Next...** until you arrive at the Modal Analysis window.

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 Response bounds (# of modes, or Lowest and Highest Frequency).
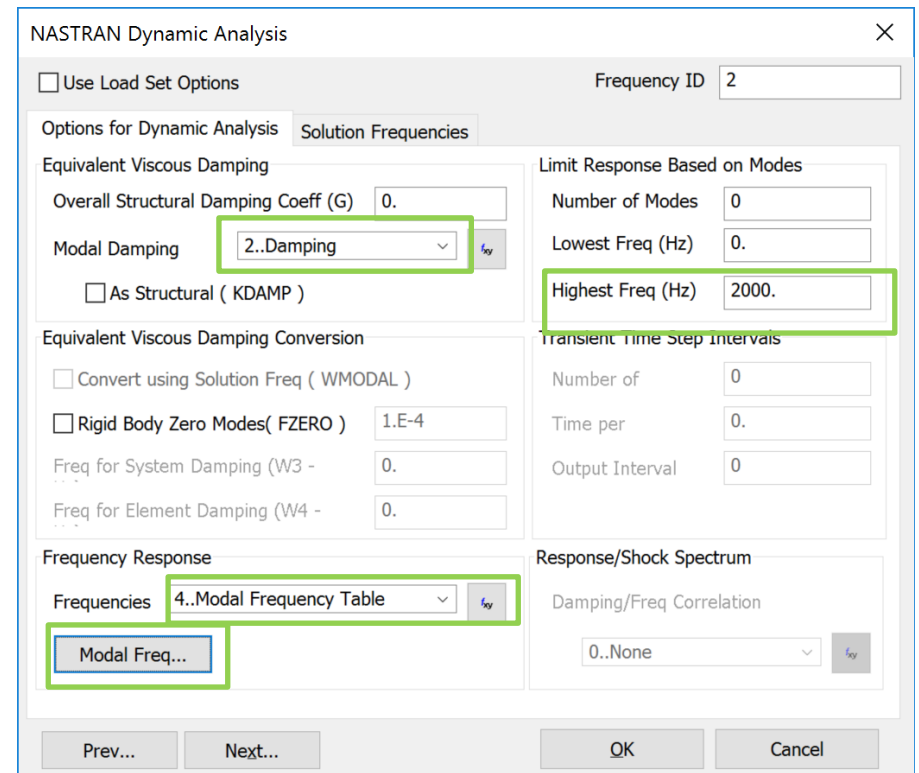
For Frequency Response, Select the "**Modal Freq…**" button, and then choose the modes you would like to create a modal frequency table from. For this analysis only the first mode will be selected to match up with the analytical solution.
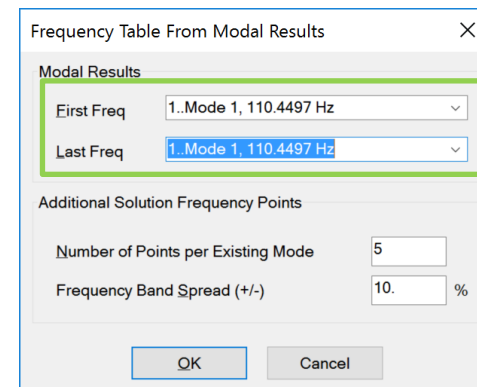
In the XY Output window, you can leave all of the options un-checked. This information can be gathered when you run a standard modal analysis so there is no need to request it here.
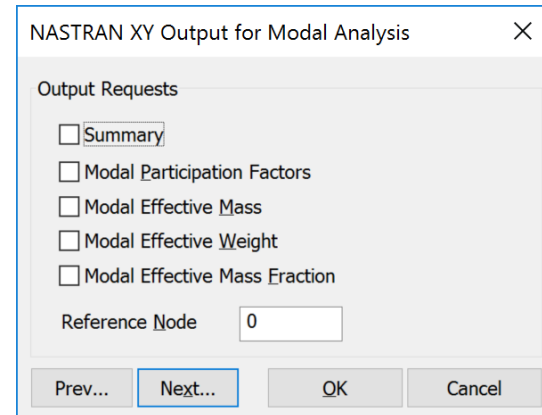
In the *Output for Random Analysis* window, select none for nodal and elemental output.
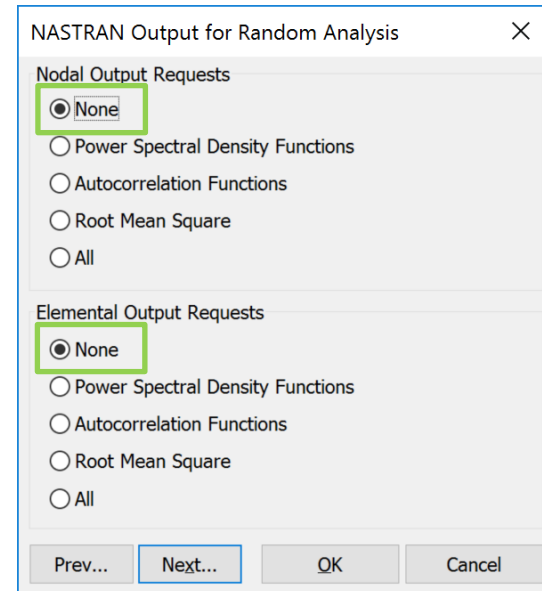
PSD Functions: Generates 'PSDF' output set for each frequency in the modal frequency table

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 4.9. If you select PSDF it will generate a function with the acceleration vs frequency for a group.

FINITE ELEMENT ANALYSIS
Predictive Engineering

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

Boundary Conditions   ✕

Primary Sets

Constraints	1..Fixed
Loads	1..PSD Excitation Node
Temperatures	0..From Load Set
Initial Conditions	0..None
Constraint Equations	0..From Constraint Set
Bolt Preloads	0..From Load Set

Other DOF Sets

Master (ASET)	0..None
Kinematic (SUPORT)	0..None
SUPORT1	0..None
OMIT	0..None
QSET	0..None
CSET	0..None
BSET	0..None

Prev...   Next...   OK   Cancel

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

4.11 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 a duration of excitation.
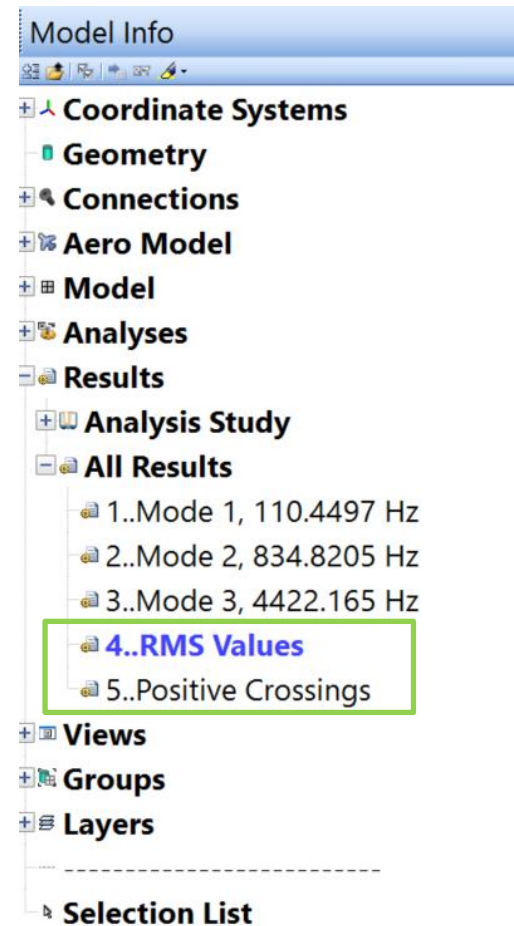
4.12 RMS VALUES

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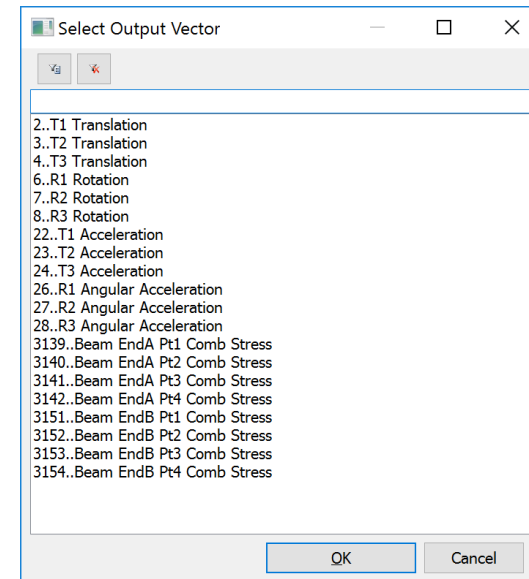
Beam EndA Pt1 Comb Stress is shown contoured over the beam. This output shown is the RMS Stress, and is also known as the 1-σ PSD stress value. This represents how much stress the beam will experience 68.3% of the time.
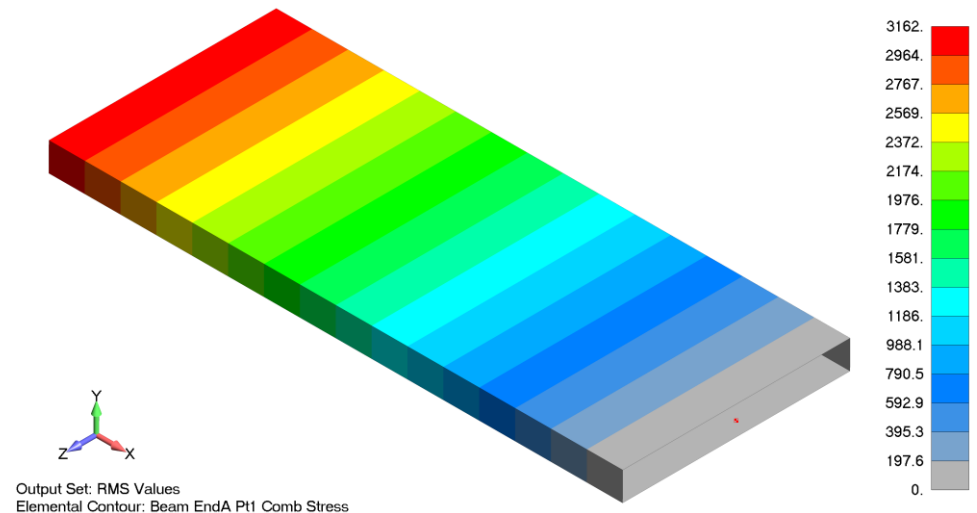
4.13   POSITIVE CROSSINGS

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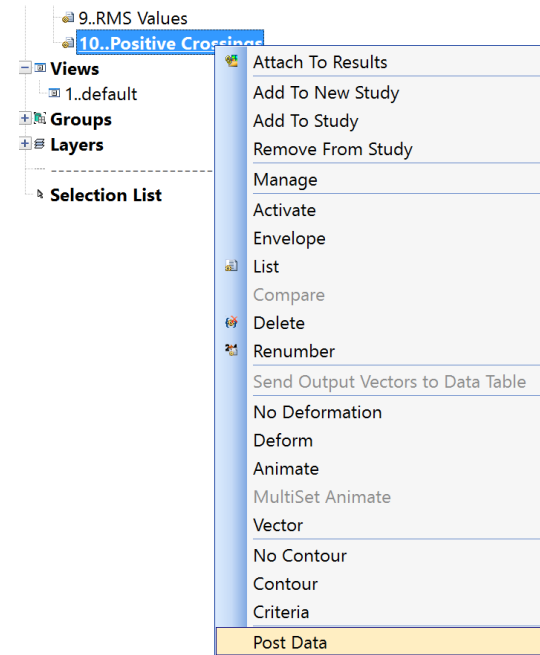
In the Post Data toolbar select the Dynamic Max/Min box in the upper right

Select the output vector for the positive crossing frequencies desired. In this model, all stress recovery points on Beam EndA show the same frequency.
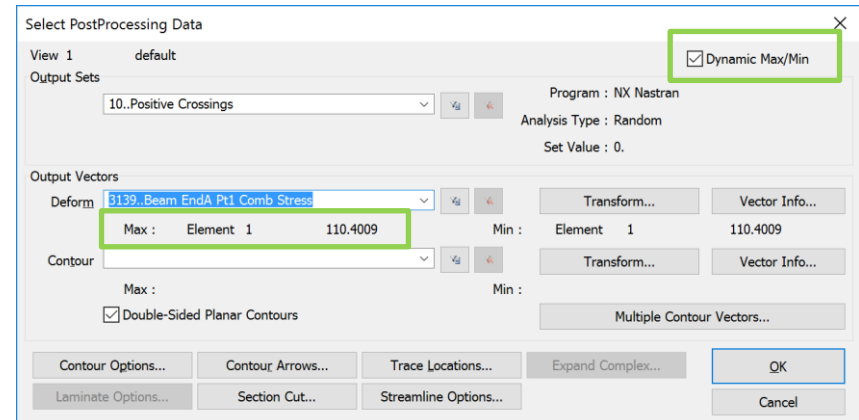
Positive Crossings can also be contoured over the model. This can help the user understand how the positive crossing frequency changes throughout the model.

4.14 FATIGUE ANALYSIS USING RMS STRESS AND POSITIVE CROSSINGS

We can see that **Beam EndA Pt1 Comb Stress** vector gives a positive crossings frequency of 110.4 Hz. This means that given the white noise PSD input of 0.2 G^2/Hz, the beam will experience a fully reversible stress of 3,162 psi at a frequency of 110.4 Hz.

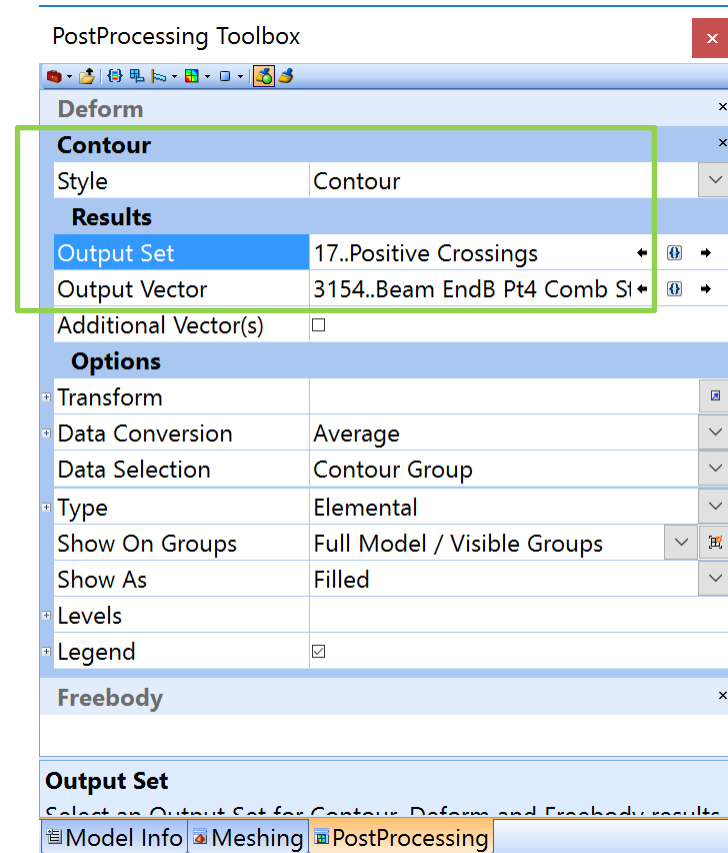Statistically speaking, this stress value represents the 1σ value and will be experienced 68.3% of the time. A 2σ of 2*3,162 or 6,324 psi will be experienced 27.1% of the time and a 3σ value of 9,486 psi will be experienced 4.33% of the time. These values represent 99.73% of the stresses the beam will 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 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 9,486 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.
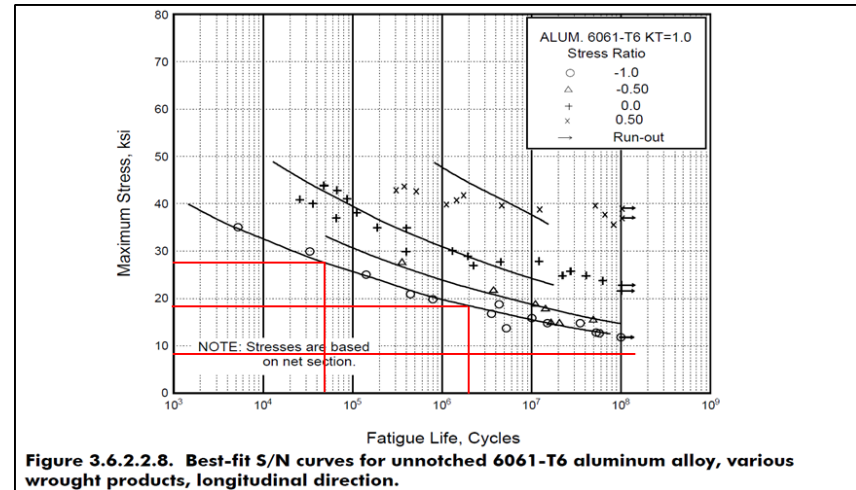
4.15 FATIGUE ANALYSIS – TIME TO FAILURE

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.

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.

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

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

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

5.   EXAMPLE 2: 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.
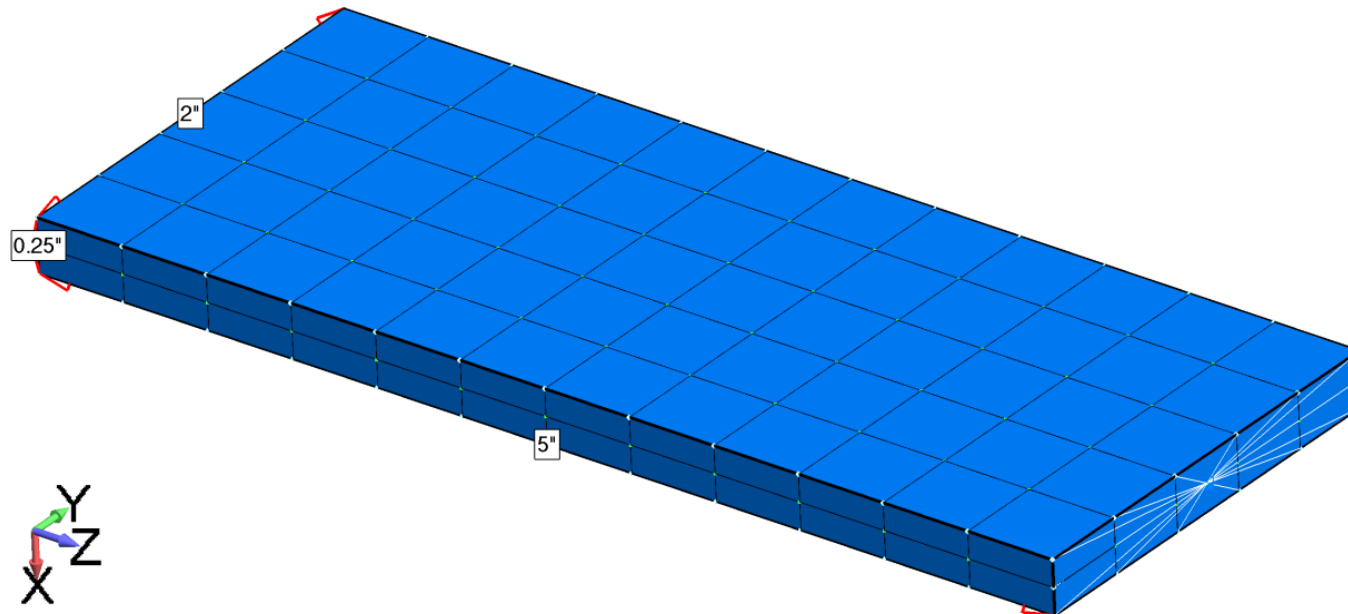
The beam properties are shown below:

$w = 2$ in

$T = 0.25$ in

$L_{beam} = 5$ in

$W = 0.5$ lbf

$E = 10e6$ psi

5.1 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}

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

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

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

We can then approximate the transmissibility:

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

Utilizing Miles Equation to estimate G_{rms} we see that G_{rms} is approximately 27 Gs:

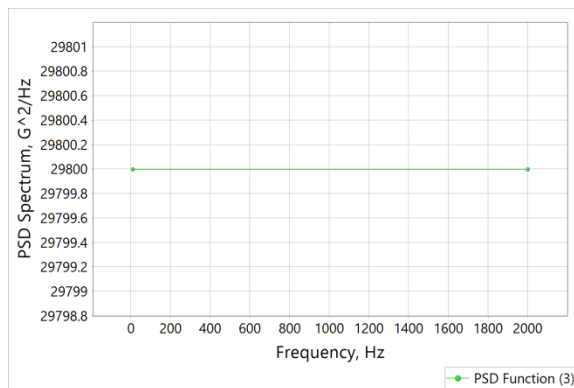$$G_{outRMS} = \sqrt{\frac{\pi}{2}PSD_{in}\,f_n\,Q} = 27G's$$

5.2 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.
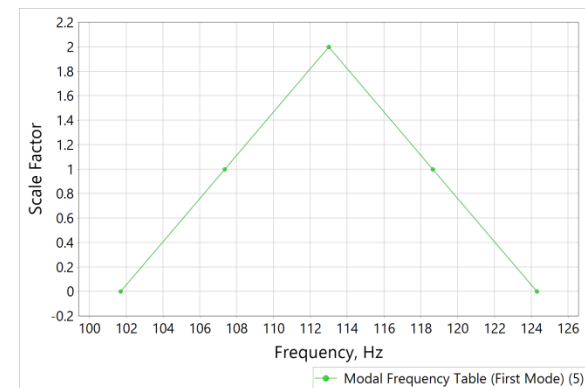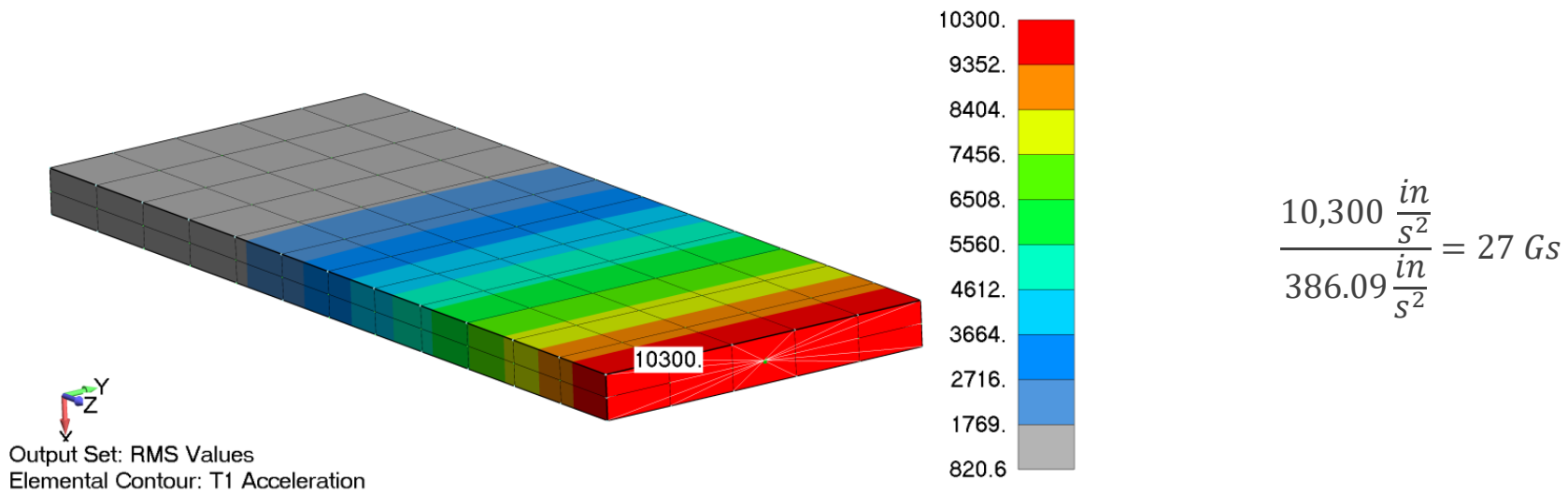
Damping Function
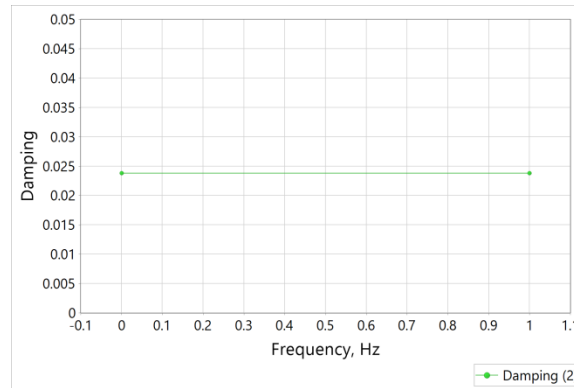
PSD Spectrum

Modal frequency table

5.3 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 Mile's equation prediction. Mile's equation predicted 27 G's for the maximum acceleration. The results show 10,300 in/s^2 acceleration which matches up with the Mile's equation prediction.

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

Output Set: RMS Values
Elemental Contour: T1 Acceleration

5.4 COMPARING MILE'S APPROXIMATION AND PSD RESULTS

An additional verification is done by comparing the PSD stress results to the static analysis with the acceleration given by Mile's equation. The below results 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.
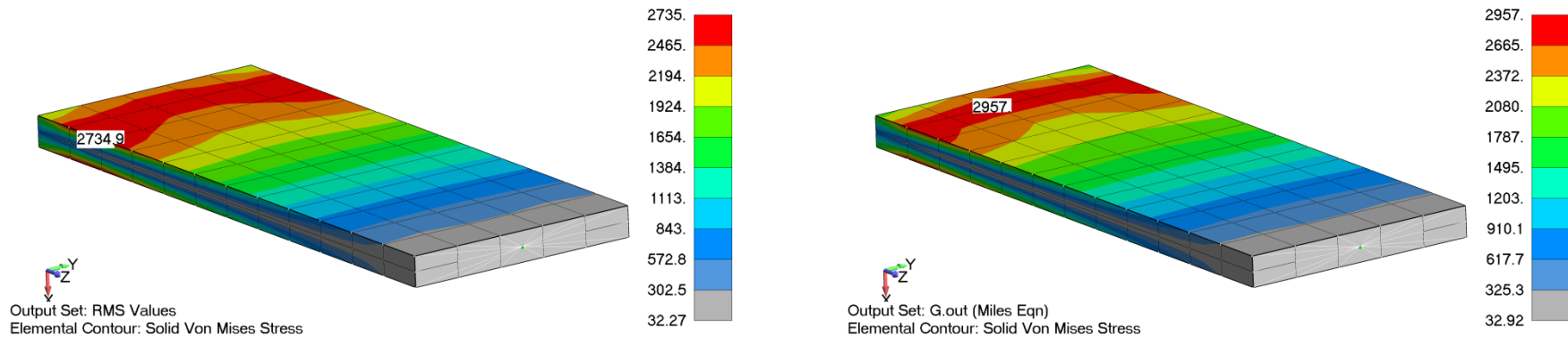
Hand Calculations:

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

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

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

6. 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."