Principles of Vibration Analysis: Normal Modes to PSD to Direct Transient

Technical Seminar for Femap and NX Nastran Users

George Laird, PhD, PE, Principal Mechanical Engineer

Adrian Jensen, PE, Senior Staff Mechanical Engineer

FEA, CFD & LS-DYNA Training, Support and Consulting

Siemens NX CAD, CAM, CAE, Teamcenter and Femap and NX Nastran Sales

WHAT WE ARE GOING TO DISCUSS

Most engineers are pretty familiar with the general concepts of vibration analysis but maybe just need a few pointers on the fundamentals and perhaps some suggested readings. This white paper provides the simulation engineer with all the necessary background to perform the following analysis sequences with Femap and NX Nastran.

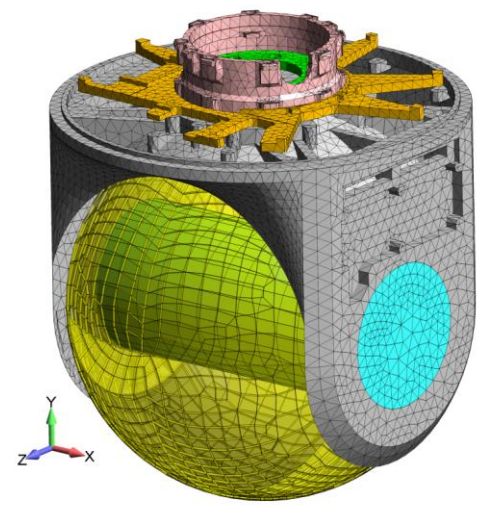
- Normal Modes Analysis
    - Natural Frequencies and Mode Shapes (why they are only shapes and not magnitudes)
    - Utility and Significance of Mass Participation
    - Mode Strain Energy
- Modal Frequency Analysis
    - Theory of Modal Frequency (all loads are sinusoidal)
    - Shaper Table Sine Sweep Analysis
    - Expanding Complex Results
- PSD Analysis
    - Modal Frequency Analysis using Statistics
    - One Sigma versus Three Sigma von Mises Results
    - Fatigue Analysis with PSD
- Direct Transient Analysis (it is easier than you think)

1. INTRODUCTION TO LINEAR DYNAMICS (NX NASTRAN)

Vibration analysis is a huge topic and is easily the second most common type of FEA analysis after the basic static stress analysis. Within the field of vibration analysis, the most common type of analysis is that based on the linear behavior of the structure or system during its operation. That is, its stress/strain response is linear and when a load is removed, the structure returns to its original position in a stress/strain free condition. Although this might sound a bit restrictive, it actually covers a huge swath of structures from automobiles, planes, ships, satellites, electrical circuit boards and consumer goods. If one needs to consider a nonlinear response of the structure during operation, there exist codes such as LS-DYNA that can solve for the complete nonlinear vibration response. But that is not simple or basic and is left for another seminar sometime in the future.

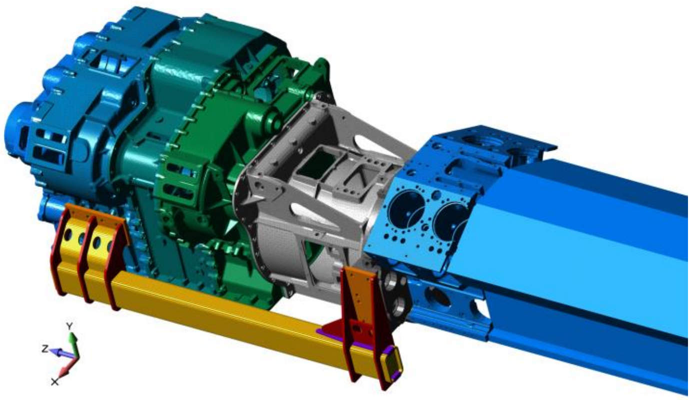
Vibration Rich Environment

Linear FEA Model (courtesy Predictive Engineering)

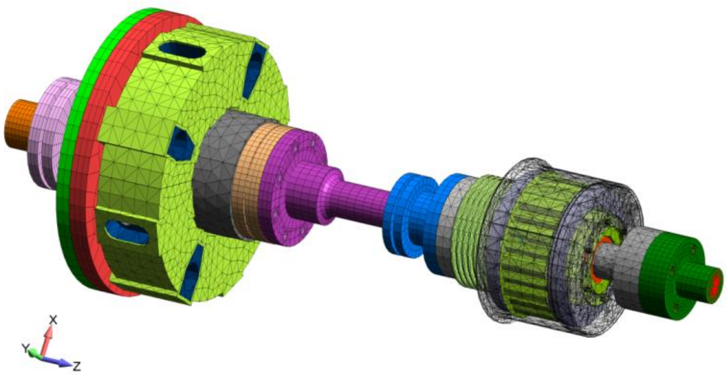


1.1 SOME EXAMPLES OF LINEAR VIBRATION ANALYSIS (PREDICTIVE)

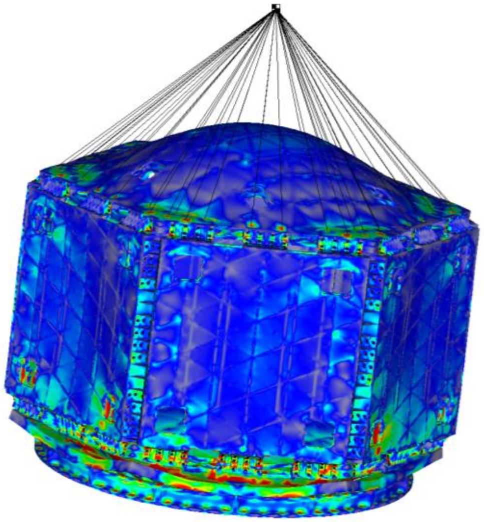
3,000 HP Transmission



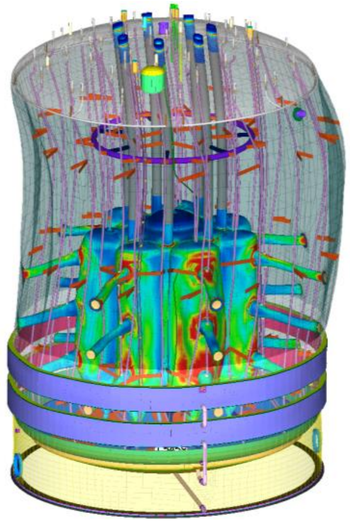
Drive Train Coupling



Satellite PSD Analysis



Seismic Analysis of Large Vessels



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

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

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

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

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:

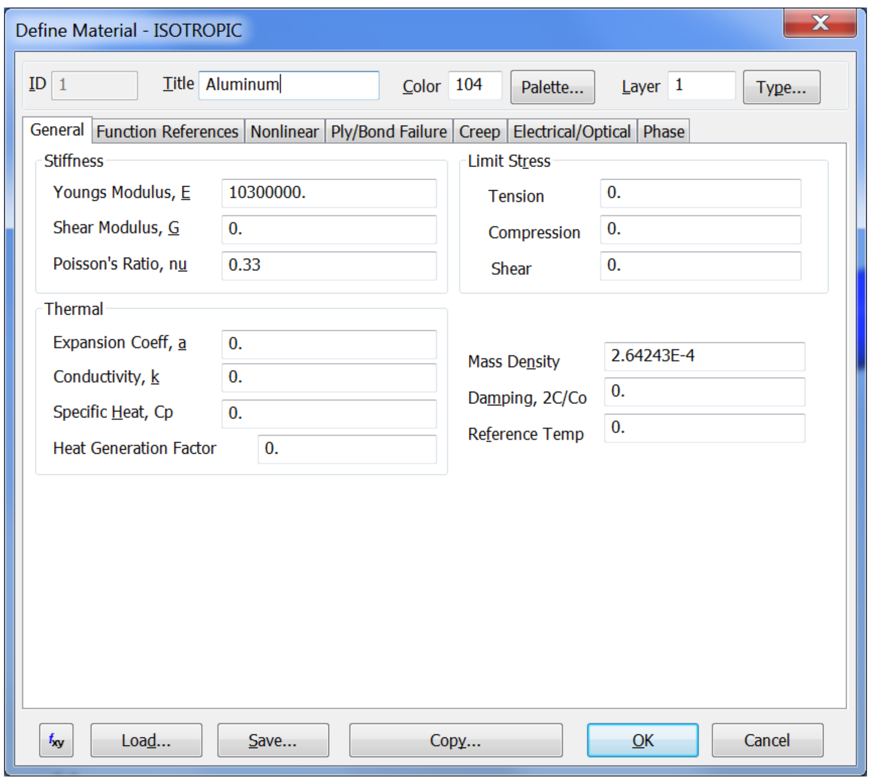
$$Ku = F$$

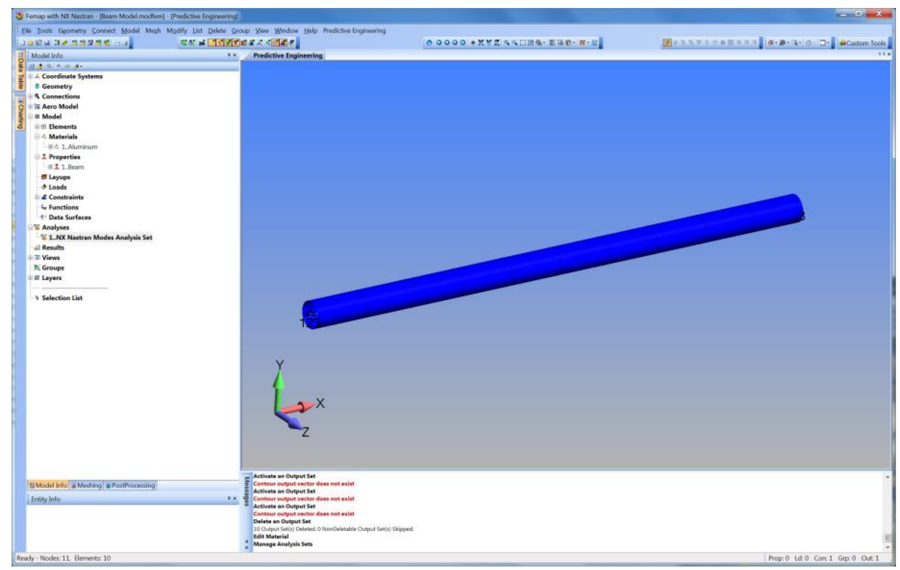
2. FOUNDATION OF FREQUENCY ANALYSIS

2.1 BABY'S FIRST BEAM MODEL

Normal modes only needs three material properties and some FEA lash up that will create a stiffness / mass relationship. A constraint set is optional.

Elastic Properties and Mass Density (Snails)

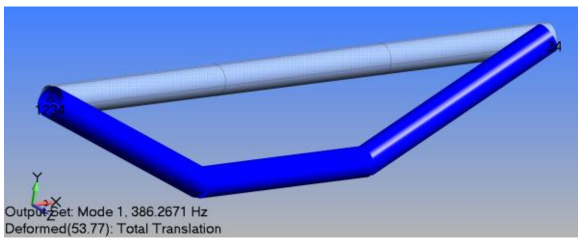


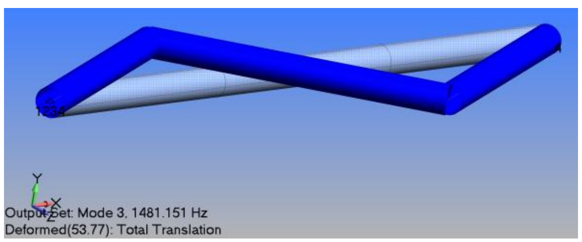


2.1.1 HERE'S YOUR MODAL ANALYSIS CHECKLIST

- 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/s^2)
- 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

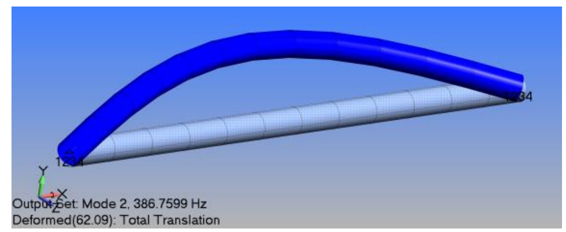
Three Element Mesh (Coarse)

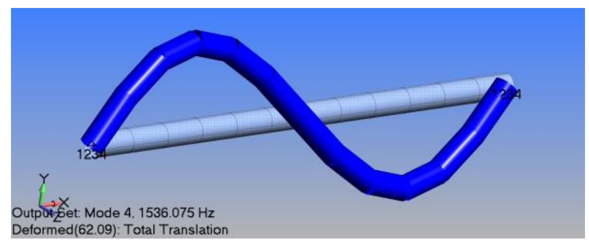


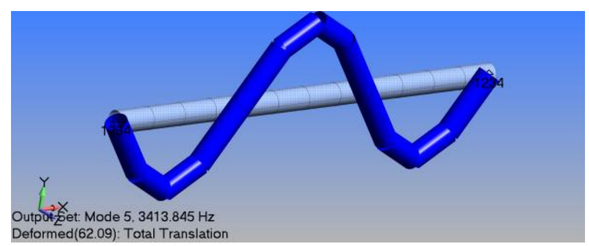


Doesn't Exist

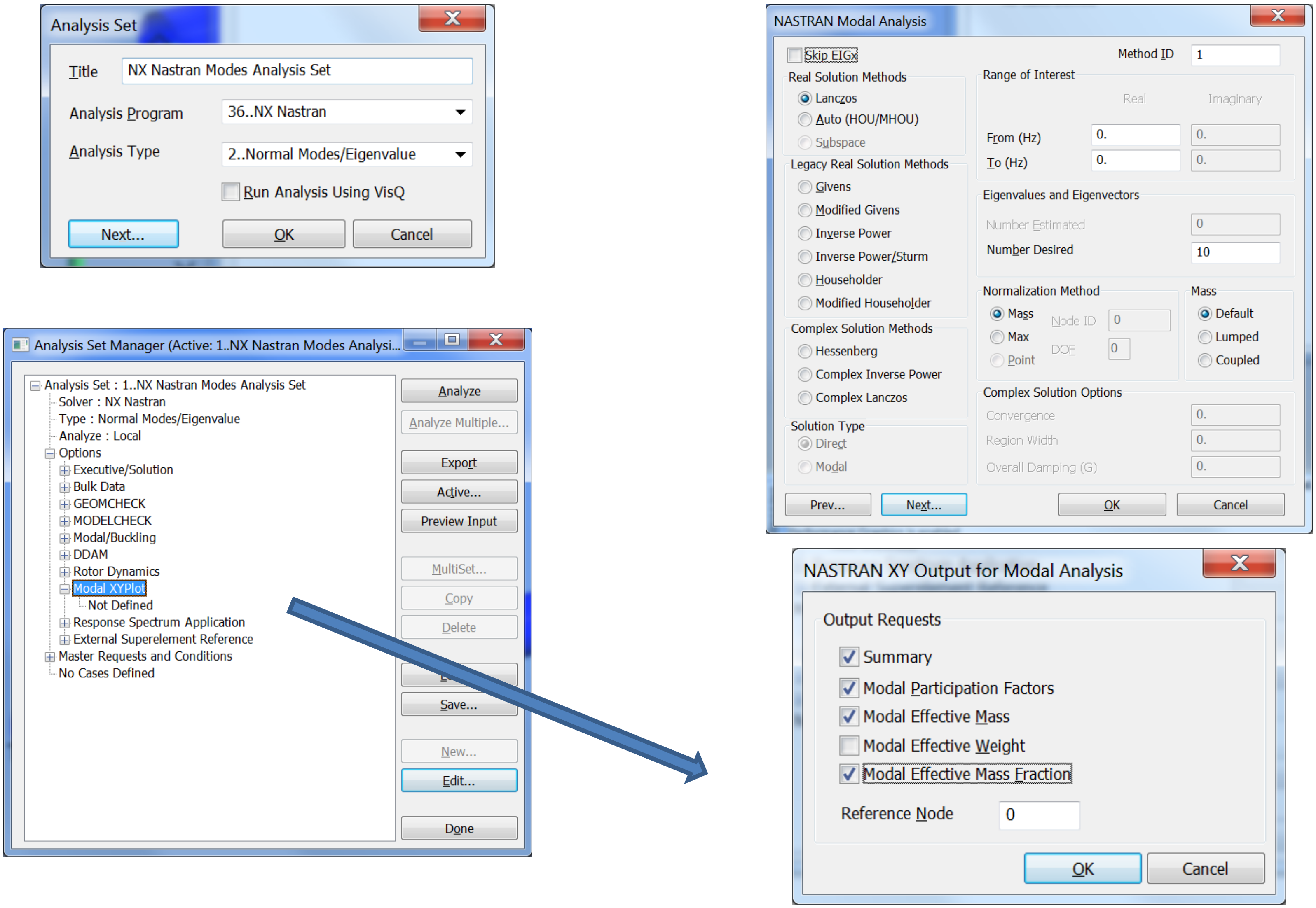
Twelve Element Mesh (Fine)







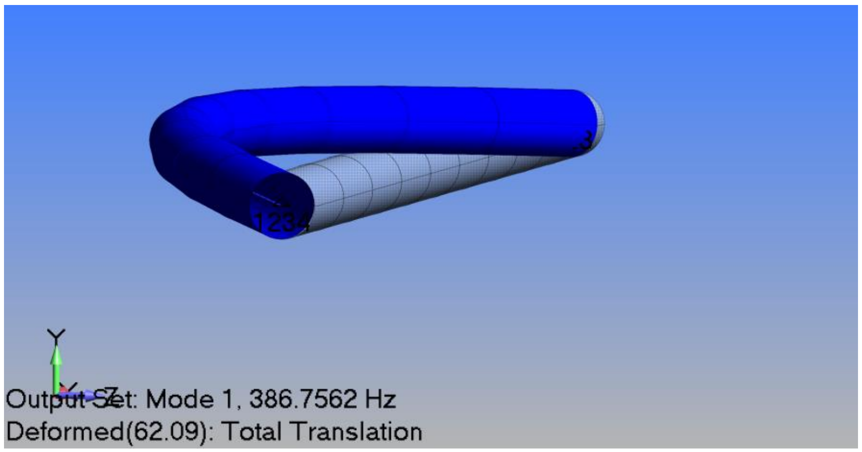
2.1.2 SETTING UP THE MODEL FOR NORMAL MODES WITH MASS PARTICIPATION



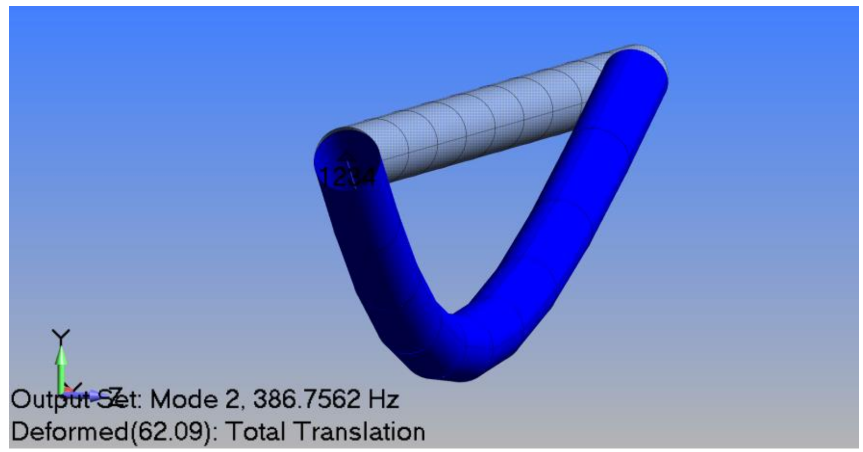
2.1.3 INTERPRETING RESULTS BASED ON ORTHOGONALITY AND MASS PARTICIPATION

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.

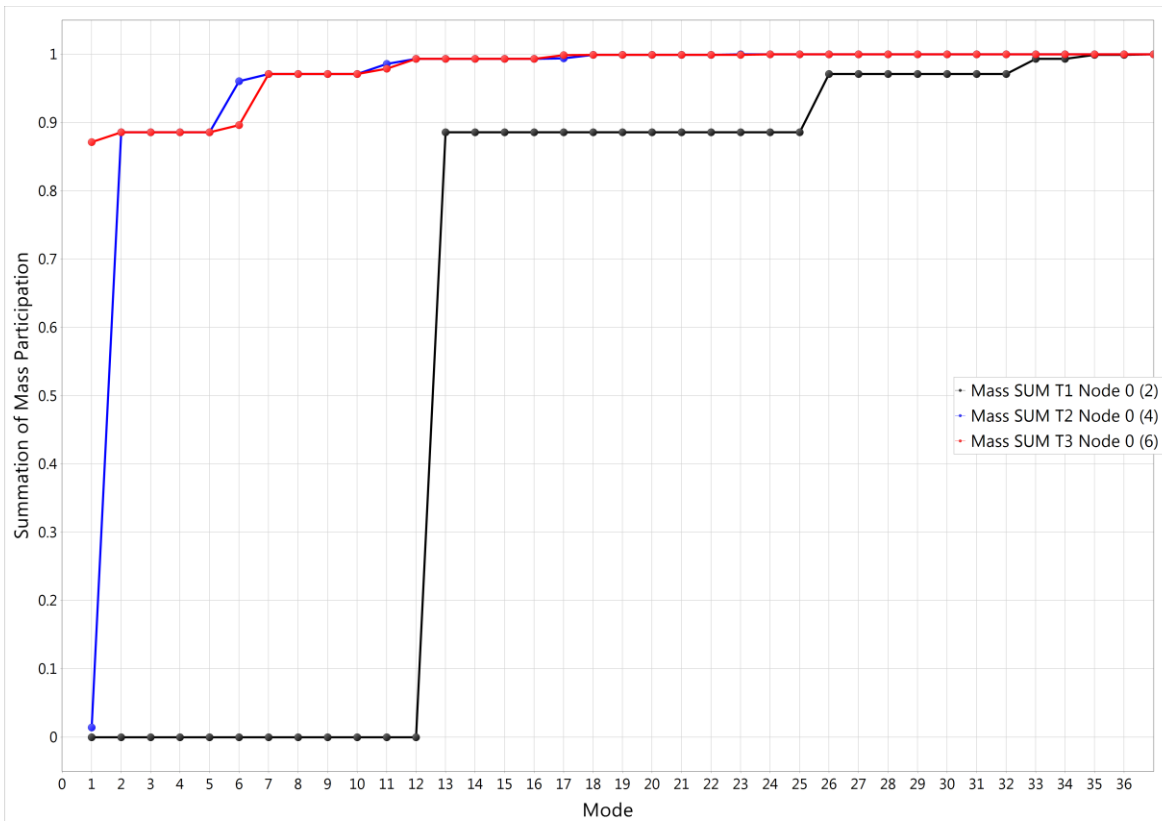
Frist Frequency 386.8 Hz



Second Frequency 386.8 Hz



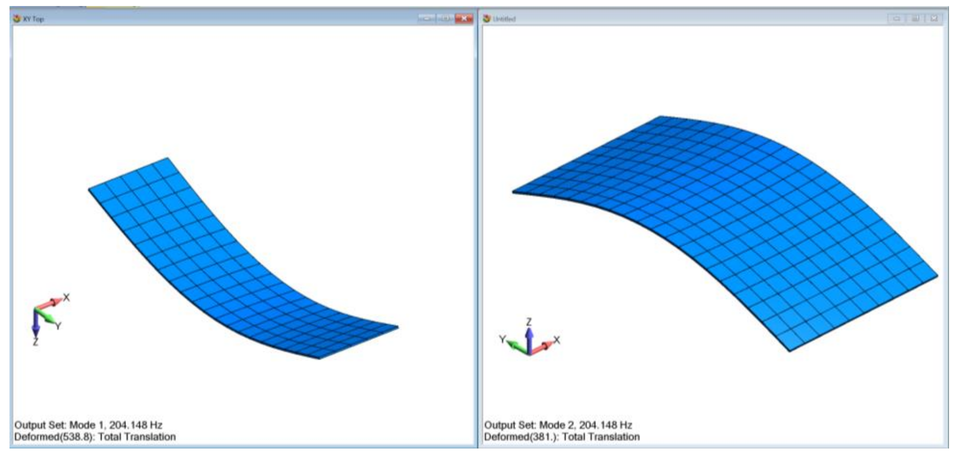
Mass participation tells you which modes have the “umph” and how many frequencies you need (modes) to accurately capture the dynamic response of the structure. On this later subject, a modal frequency analysis (e.g., PSD) formulates its response based on the number of modes chosen for the analysis. To ensure that you have captured the dynamic response of the structure, you’ll want to use enough modes that you have at least 90% of the mass of the structure covered. What does this mean? Take a look at this screen shot showing the Mass Participation versus Number of Modes for the simple rod model. The bending modes capture 90% of the mass after 6 modes while to get the axial mass, it takes 26 modes.



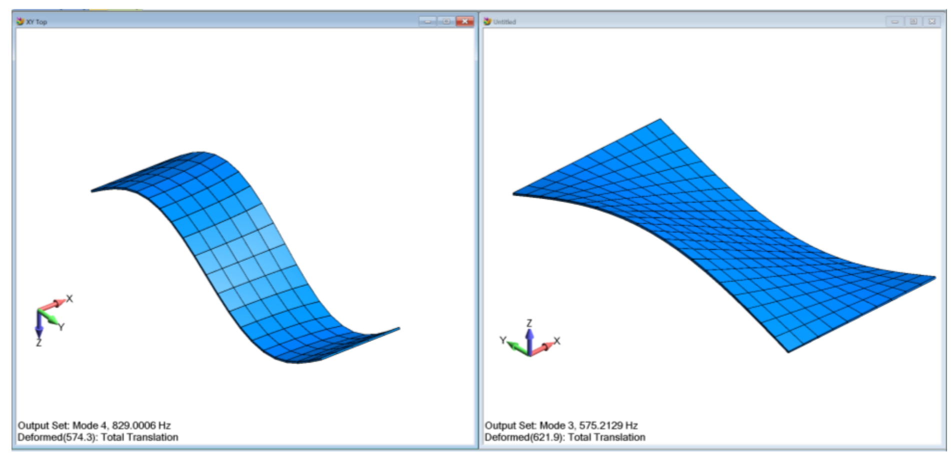
2.1.4 SYMMETRY AND FREQUENCY ANALYSIS

This is just a little note to remind everyone that you can rarely use symmetry in a frequency analysis since the mode shapes are rarely symmetric. It sounds off but the higher frequency mode shapes are not symmetric. One might be able to use symmetry if you are only interested in the most basic mode shape.

At the first mode, we have the same natural frequency.



At higher frequencies, things get lost.

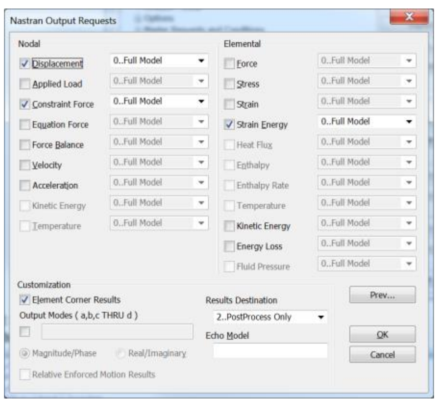


2.1.5 SIGNIFICANCE OF STRAIN ENERGY FOR FREQUENCY ANALYSIS

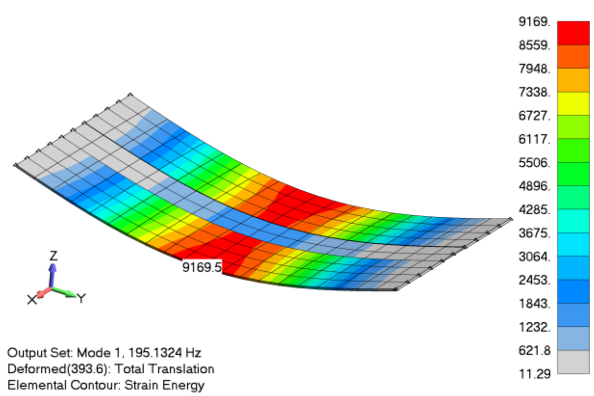
If one wants to move your natural frequencies up or down, sometimes intuition is good enough but it never hurts to have a quantitative tool. When a structure flexes or vibrates, there will be regions within the structure that are deforming more and other regions less. Since a natural frequency analysis provides you with the mode shape (dimensionless deformation); it can also easily provide you with a contour plot of the relative strains within that structure. It sound simple but can be tricky. Just to make sure that we understand this concept, we'll use a very simple model to explain this concept.

A center strip of the model has been thinned. This allows us to clearly see the effect of how strain energy plots can show us how to modify the structure to increase or decrease its natural frequencies.

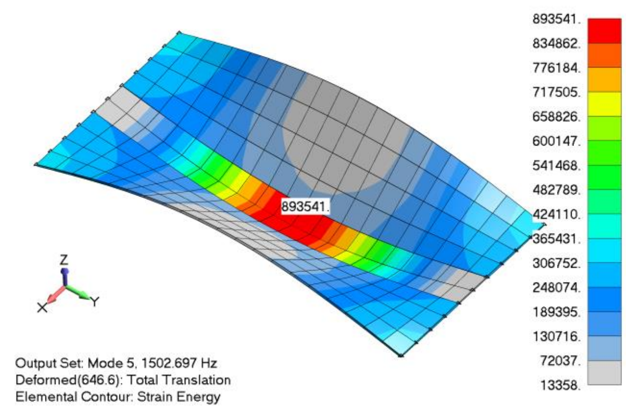
The Default Setup



Follow the Red – Increase the thickness of the outer strips



To Increase the Fifth Mode – Increase the thickness of the middle section

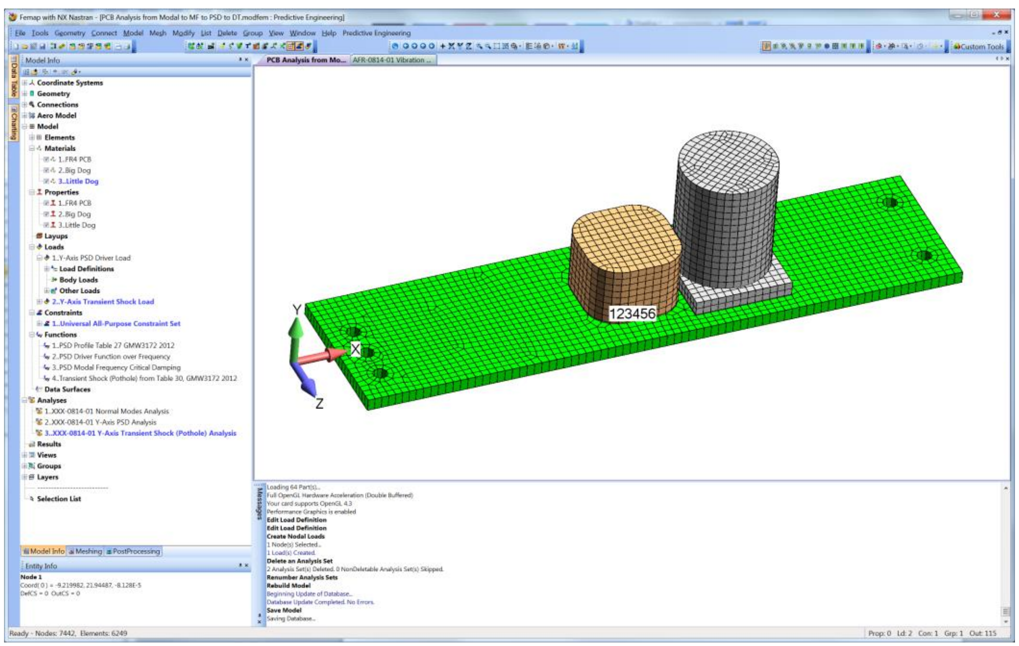


3. STANDARD NORMAL MODES ANALYIS

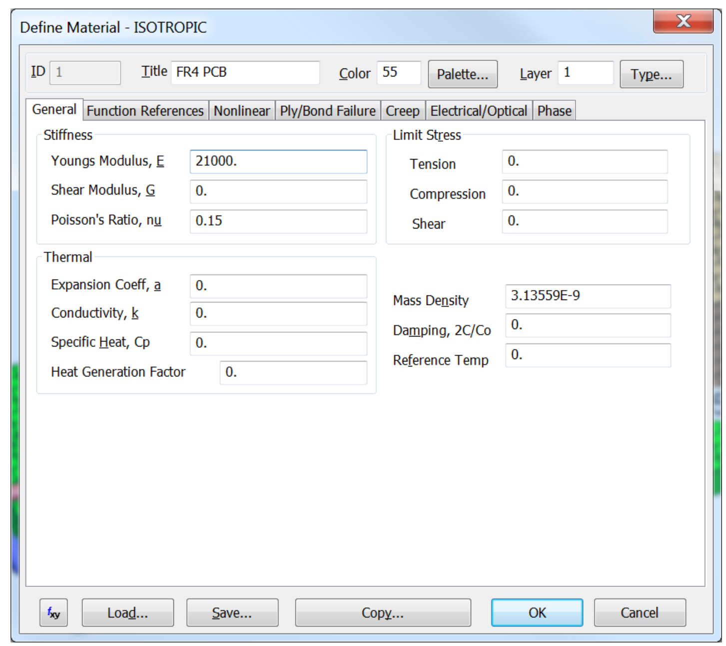
To see how this is applied in practice, we will run through an analysis project from start to finish (Normal Modes, Modal Frequency, PSD and Direct Transient). The model has been tweaked to protect the innocent.

We are starting with a PCB with two heavy electrical components. The PCB is a plate structure and the electrical components are modeled with solid elements. The PCB is screwed into a heavy component at the ends. The client must demonstrate that their PCB component can survive GM's vibration, PSD and Direct Transient (pothole) specifications (but that has been modified to confuse any automotive spies).

Femap v11.1.2A Model



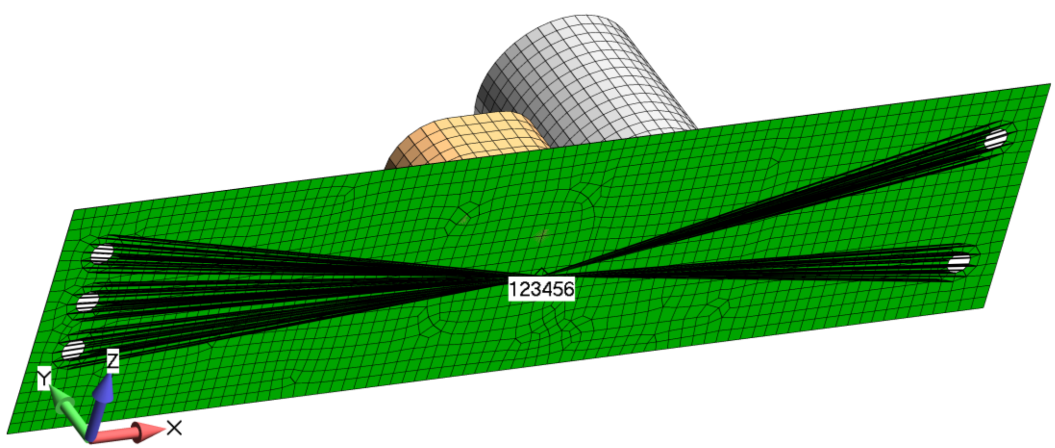
Units: N, mm, Tonne, s



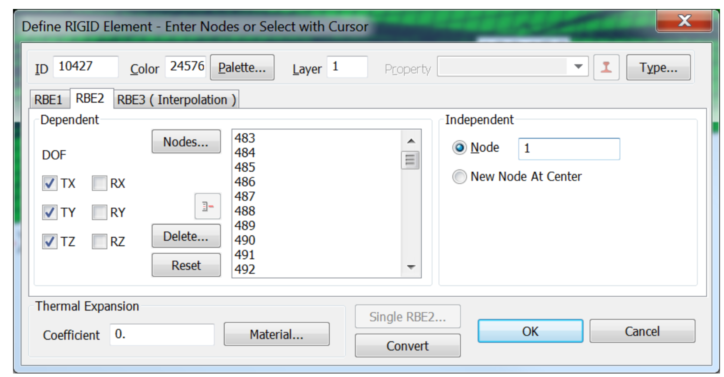
3.1 MODEL SETUP

Since we know in advance that we will be doing more advanced frequency analyses, we can set up the constraints such that we don't have to mess with them in downstream analyses. The RBE2 element is setup to mimic a pinned connection at each of the PCB mounting holes. This is done by releasing the dependent DOF's of the RBE2. If you are not up-to-speed on multi-point-constraint (MPC) theory, take a look at our Seminar "Connections 2013: RBE2, RBE3 and CBUSH Elements".

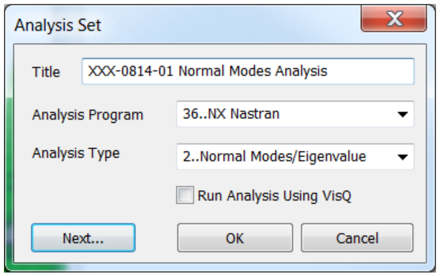
RBE2 Element to Common Node



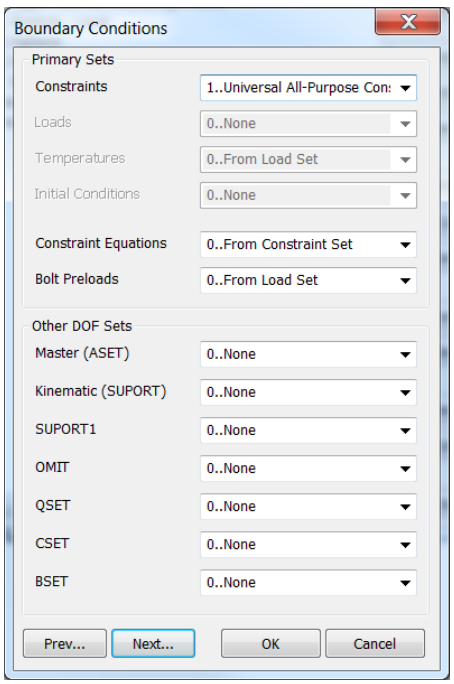
RBE2 with 3-DOF Dependent Nodes

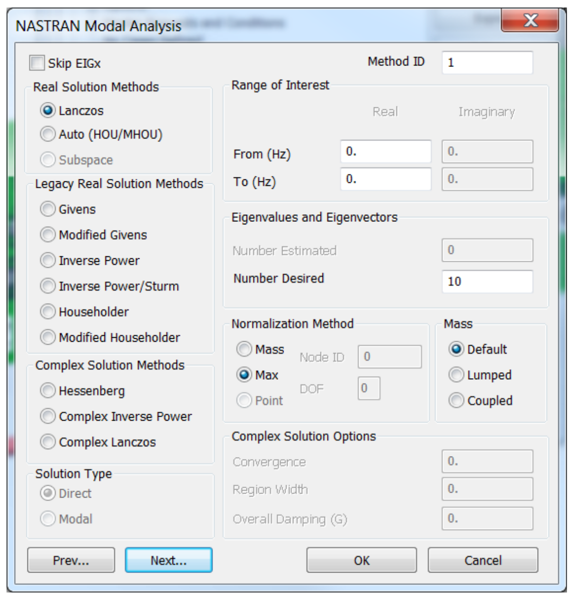


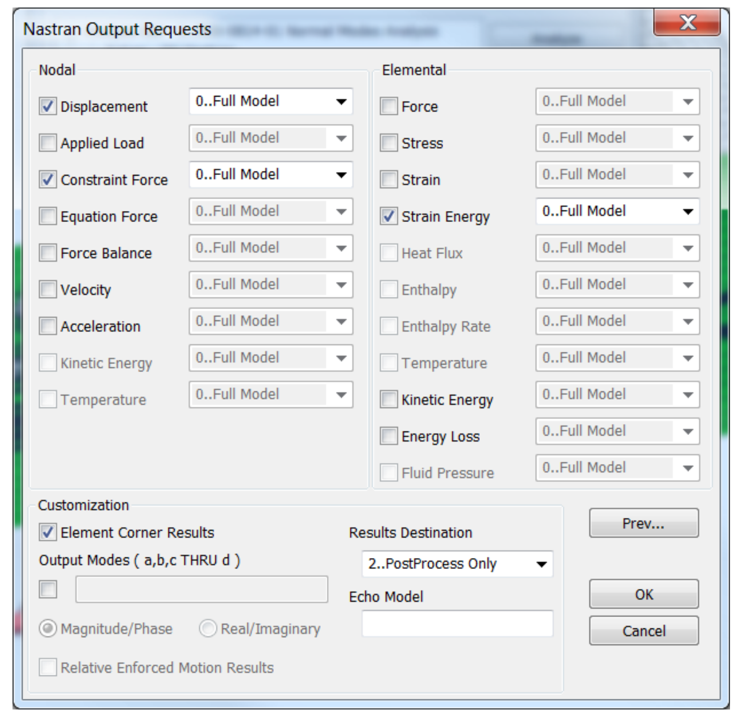
3.2 NATURAL FREQUENCY RESULTS AND INTERPRETATION



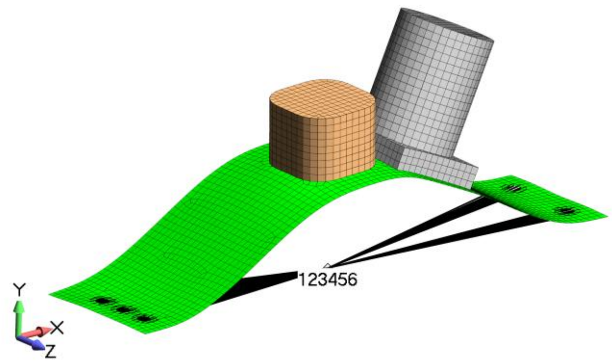
A normal modes / Eigenvalue analysis is the starting point for all linear dynamics work. It is simple to setup but difficult to interpret the results.



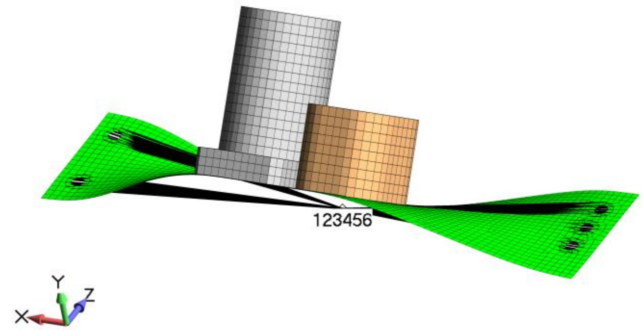




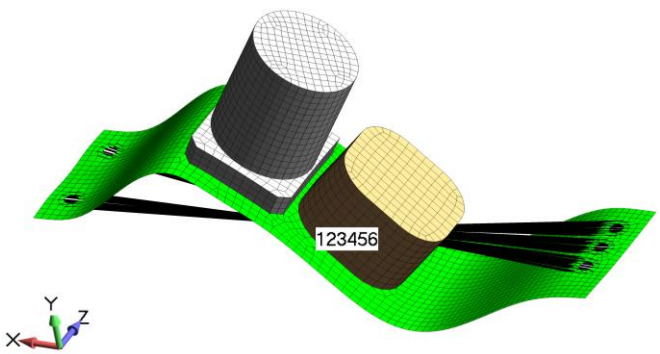
The mode shapes indicate the shape of that particular natural frequency. Since we are solving the EOM that has no {Force} or {Load}, the mode shapes have an arbitrary magnitude but they do tell us something very important. For example, the first mode flexes in the Y-direction and if excited in that direction,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



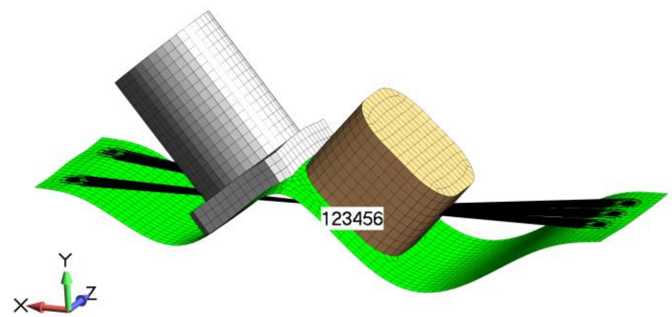
Output Set: Mode 1, 200.4665 Hz
Deformed(1.112): Total Translation



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



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

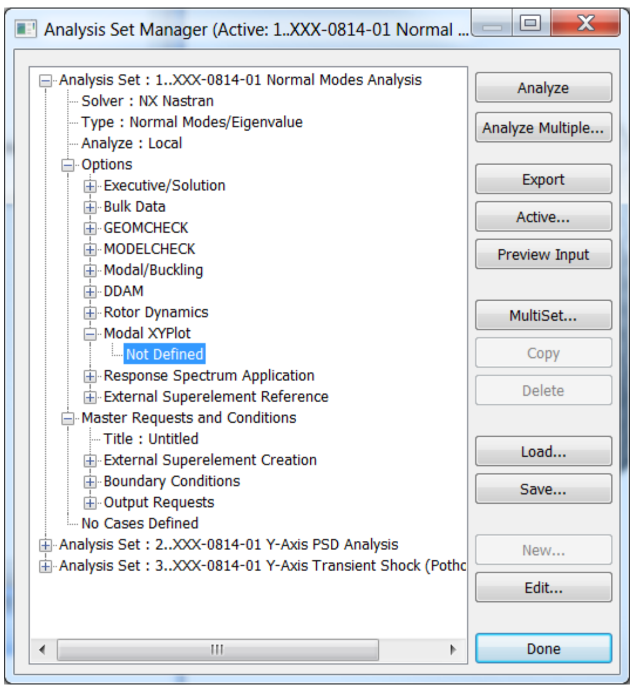


Output Set: Mode 4, 1015.848 Hz
Deformed(1.08): Total Translation

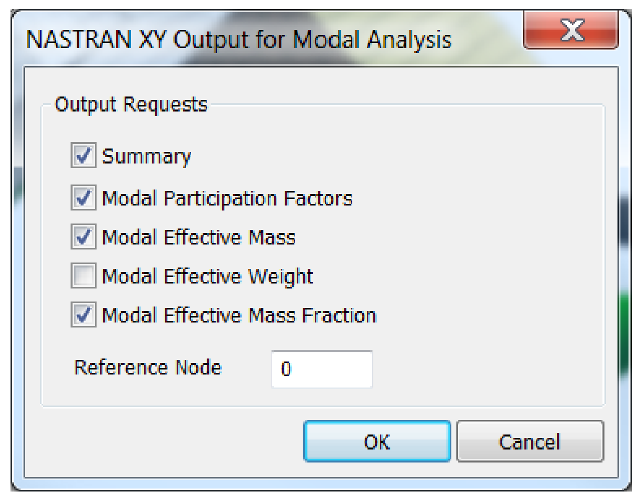
3.2.1 MASS PARTICIPATION

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).

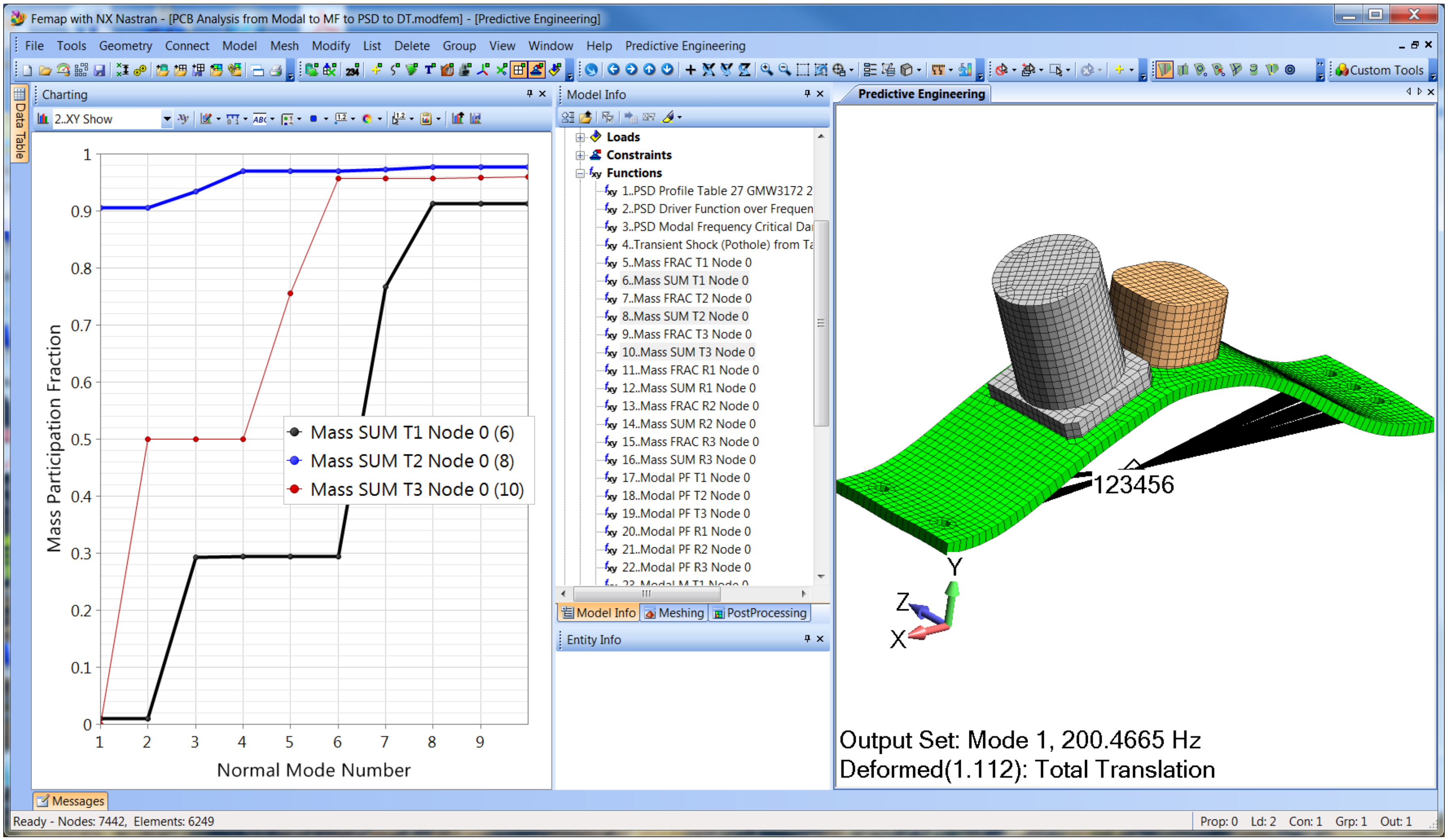
Analysis Set Manager / Normal Modes



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 particpation 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:

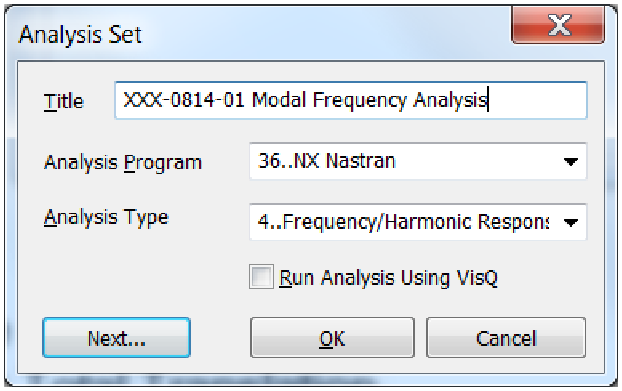
$$F_o \sin(\omega t - \theta) = m\frac{\partial^2 u}{\partial t^2} + c\frac{\partial u}{\partial t} + ku$$

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:

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

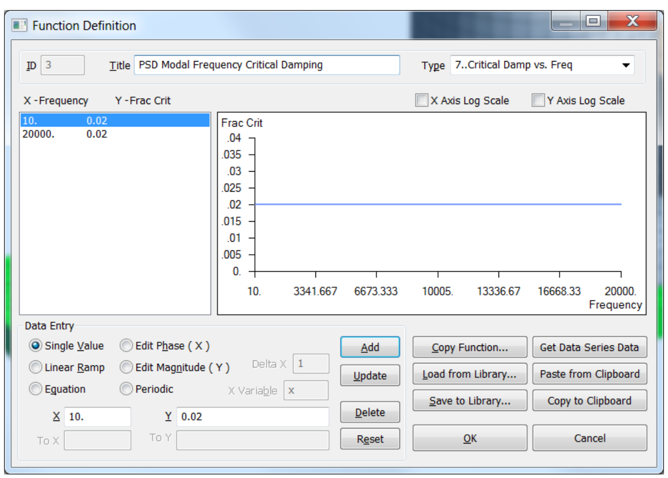
Thus, a modal frequency analysis assumes that the forcing function is sinusoidal and solves the EOM in the frequency domain with results kicked-out in the form of absolute magnitudes and phase analysis. This makes interpretation of the results somewhat challenging and requires a bit of understanding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

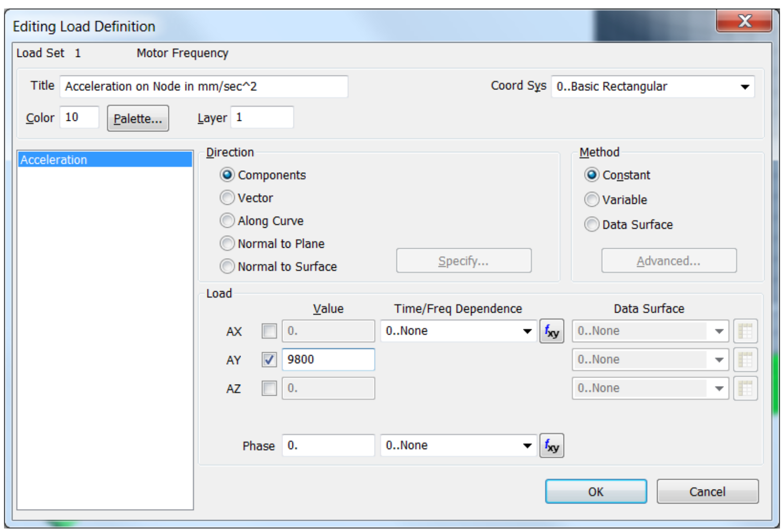


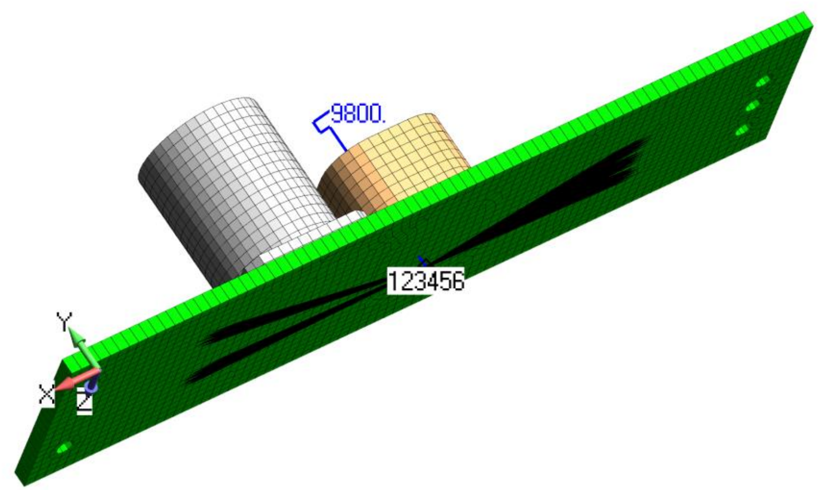
Damping is given as a function and is constant over the complete range of interest and since it doesn't matter, I just set it at 0.02 from 10 to 20,000 Hz.

If one wants to know more, take a look at this document:

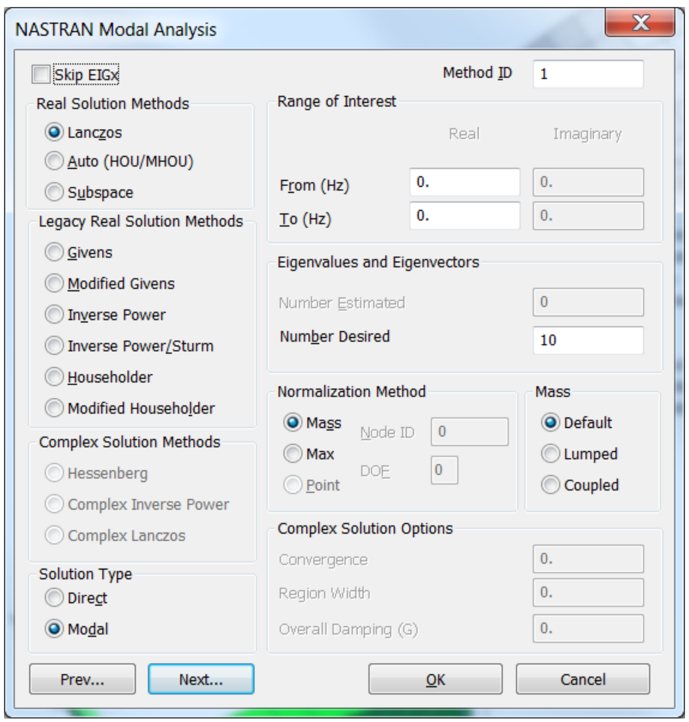
NX Nastran Dynamic Analysis.pdf

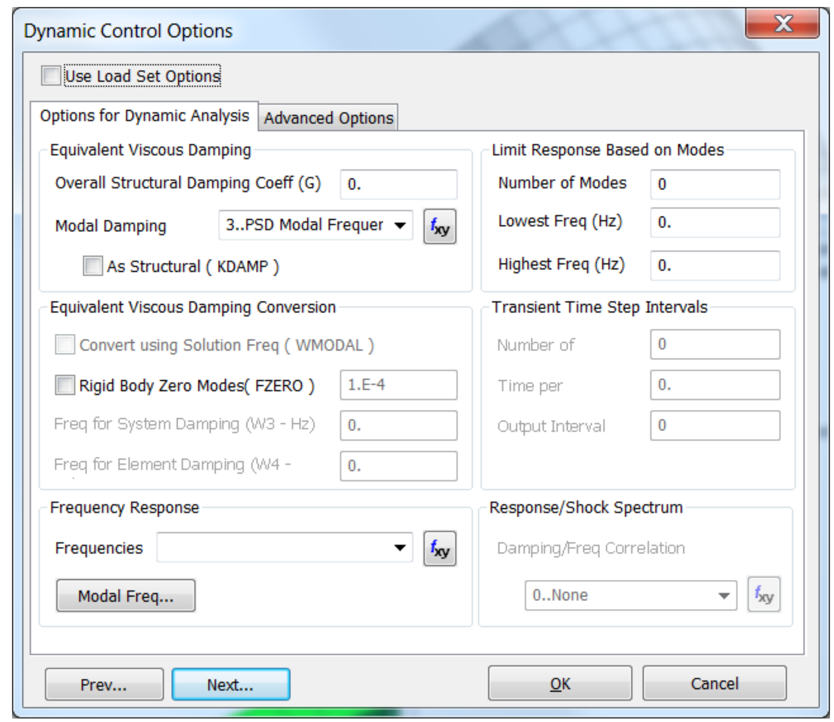
Units for dynamic analysis can be challenging. In this example model, the units are a modified SI system of N, tonne, mm and second. For the acceleration load of 1 g we have 9,800 mm/s^2. The load is applied at the independent node of the RBE2 element. It may seem funny that one can apply an acceleration load to a node that has all six DOF fixed but the modal frequency analysis understands the request and ignores the T2 SPC tag.



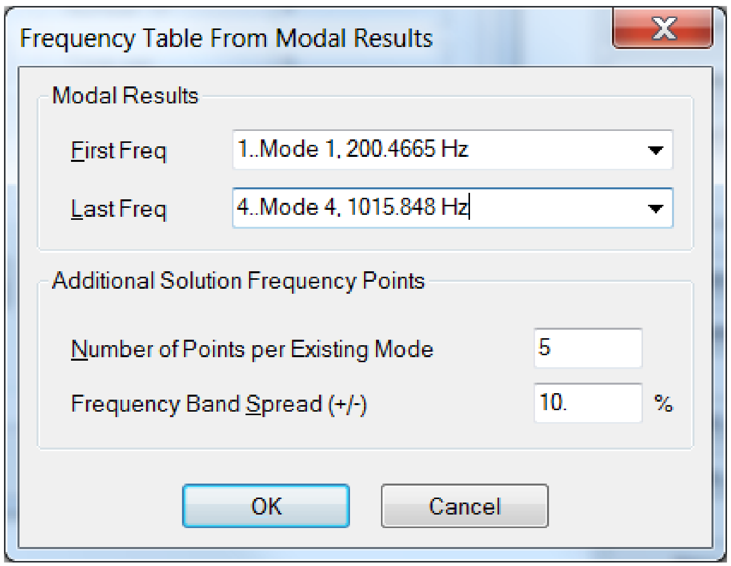


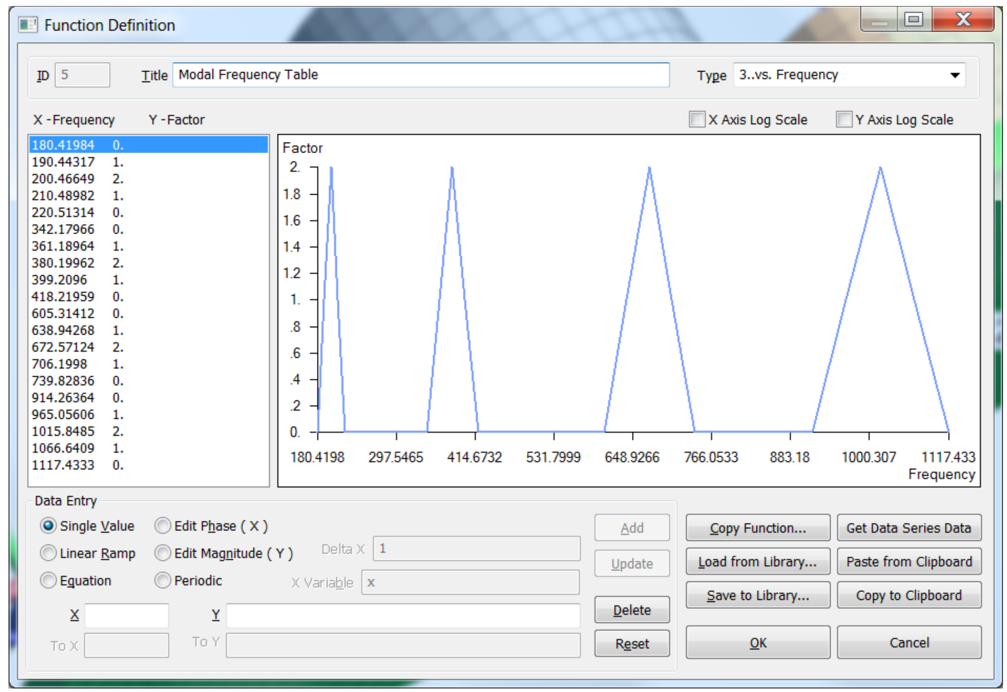
This is the heart and soul of the Modal Frequency Analysis setup. As one walks through the screens, we chose the Modal solution type, and request that 10 Eigenvalues and Eigenvectors be used to form the solution set. The next screen, we set damping to use our 0.02 critical damping curve and we request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



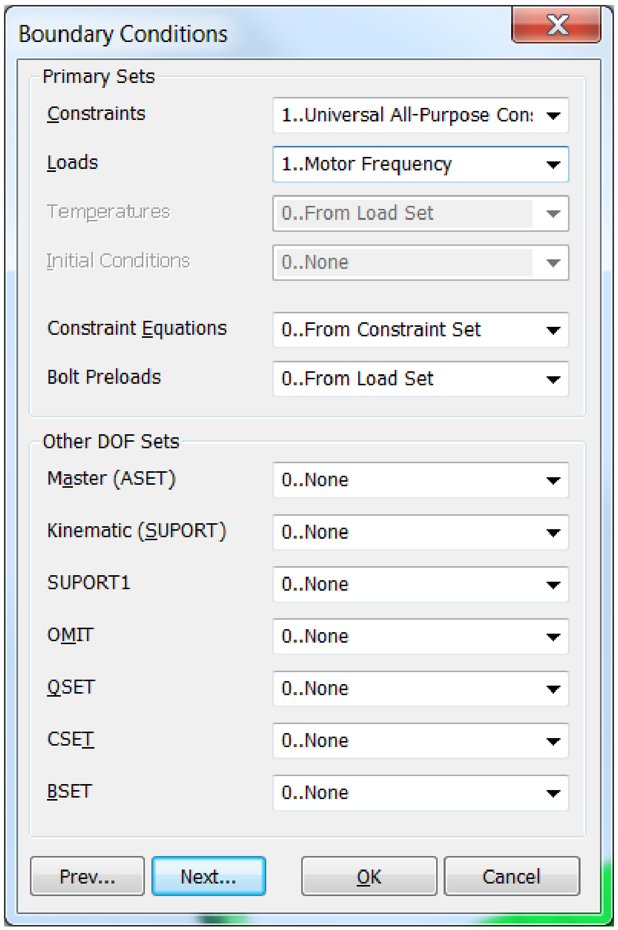


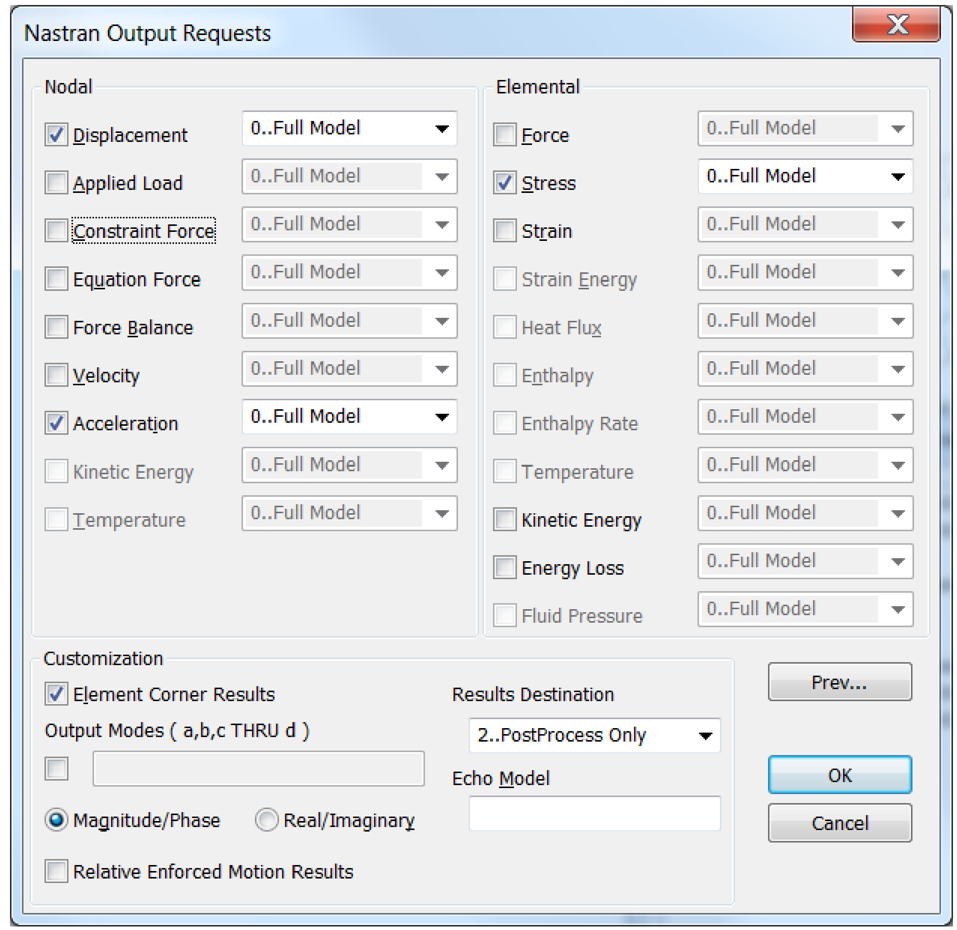
Since the linear dynamic response of a structure is determined or composed of its natural frequencies it often makes the most sense to request solutions at and around (Frequency Band Spread) these natural frequencies. Once this is done, the program creates a function showing how these solutions are spaced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



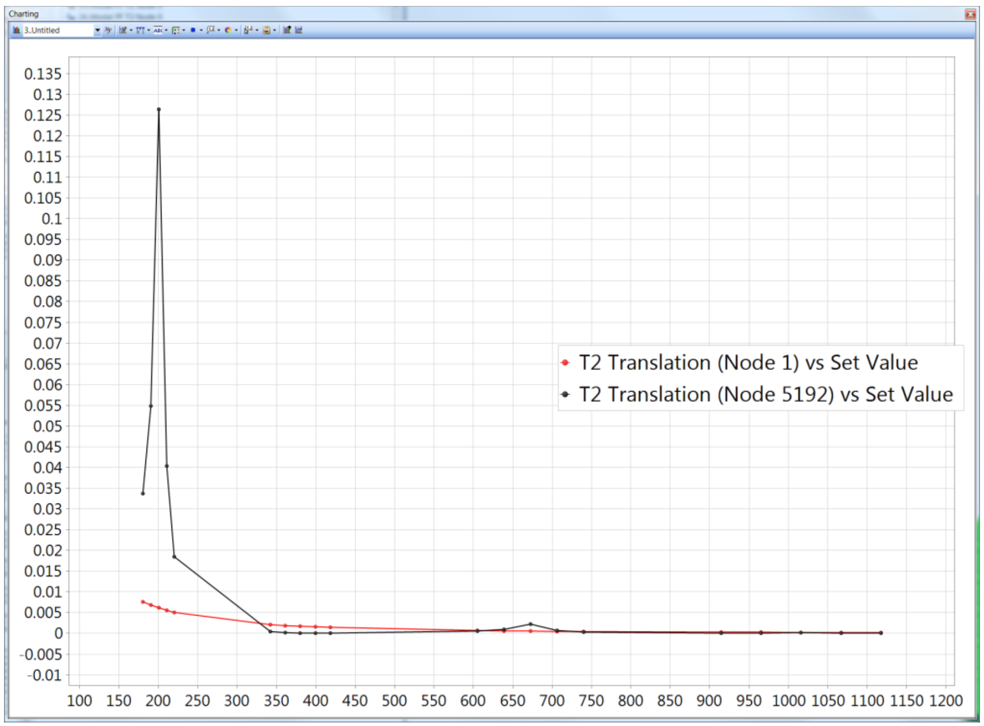


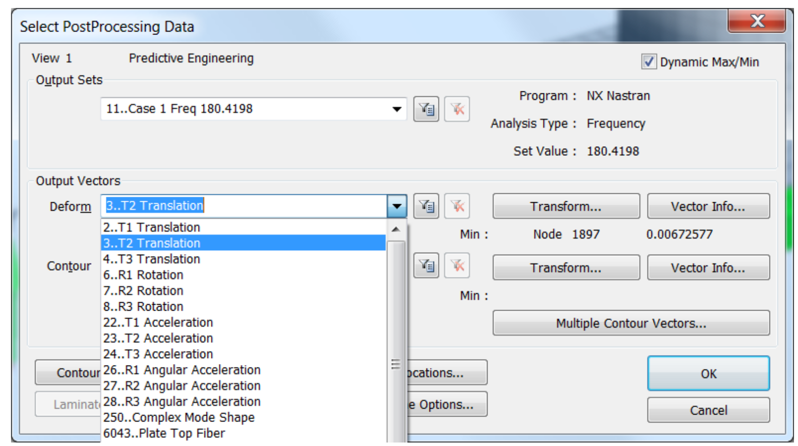
Then, one applies the boundary conditions and then lastly, one sets the output requests.



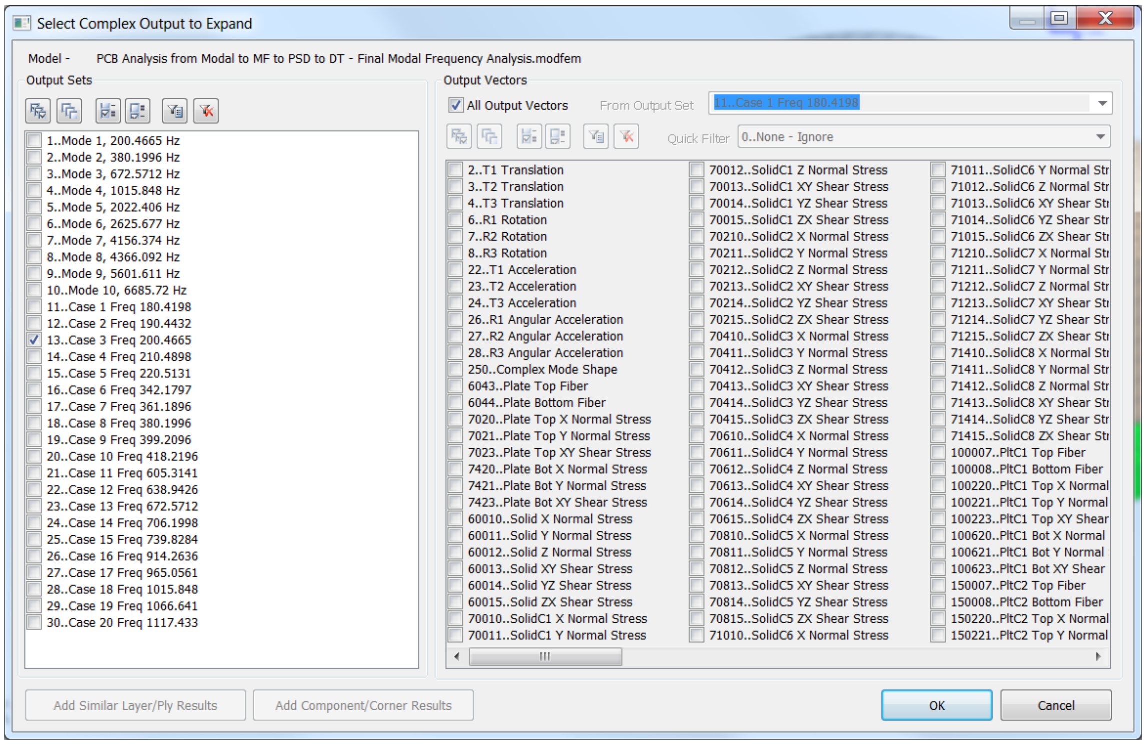


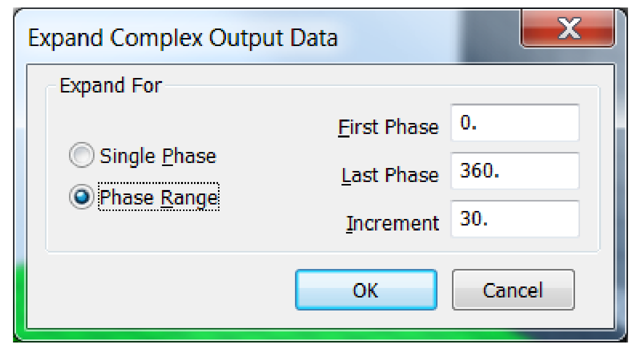
And the results show no surprises with the response peaking at the first normal mode at 200 Hz. If it is your first time with Modal Frequency, then the output results will seem a bit odd since you don't have Total Translation or a von Mises stress. All that you have are individual displacement and stress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



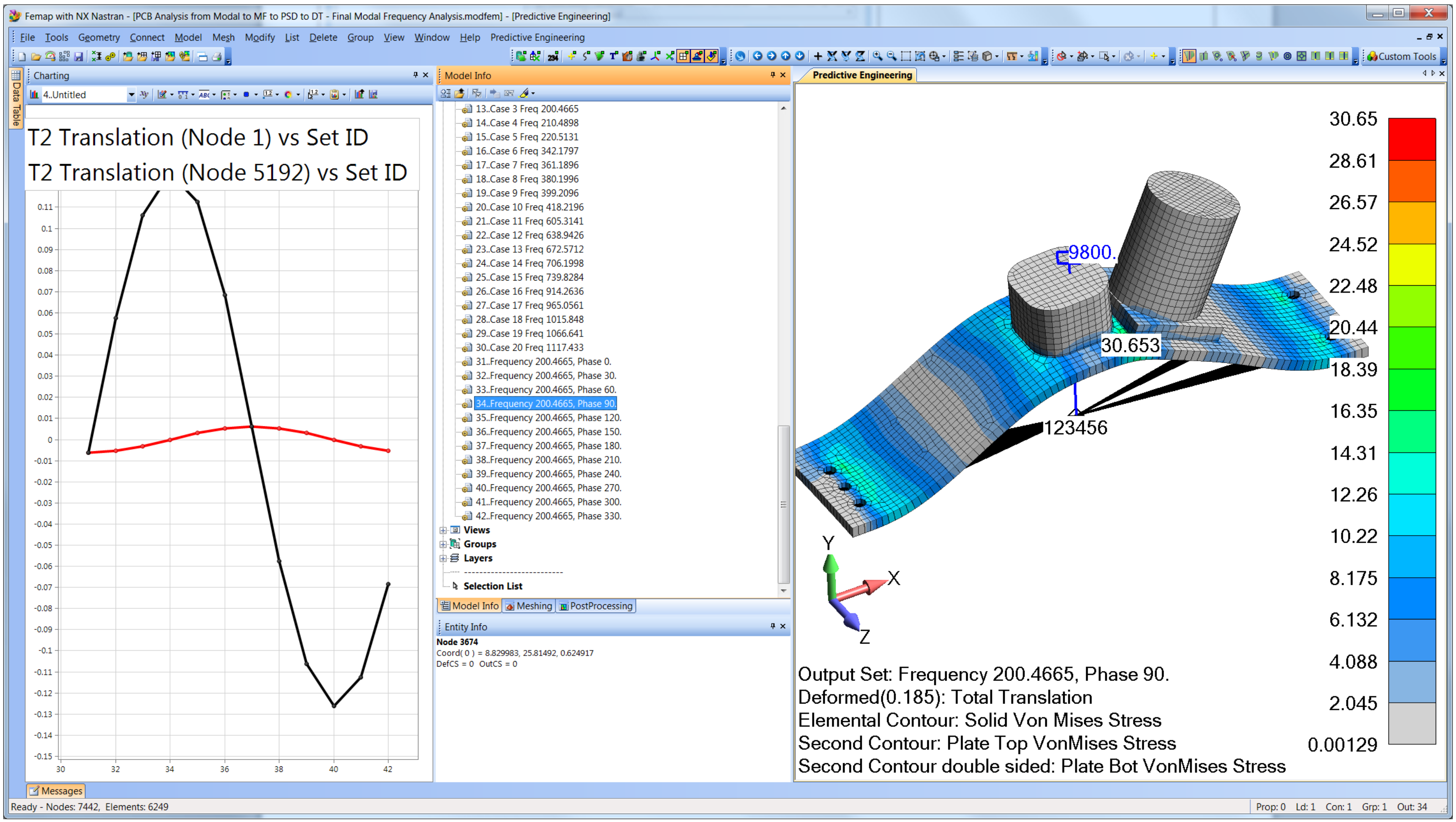


To obtain the time varying response from a Modal Frequency, one goes to Model / Output / Expand Complex and pick your solution of interest. For this structure it is the maximum response at 200 Hz and then we'll request that it is expanded into 12 solutions.





After expanding the solution, we have the full-field solution with Total Translation and von Mises stresses. Keep in mind that this maximum response requires that the excitation is in the direction of the mode shape (Y-direction) and that this particular mode has mass (mass participation 90%).



5. PSD ANALYSIS (MODAL FREQUENCY WITH STATISTICS)

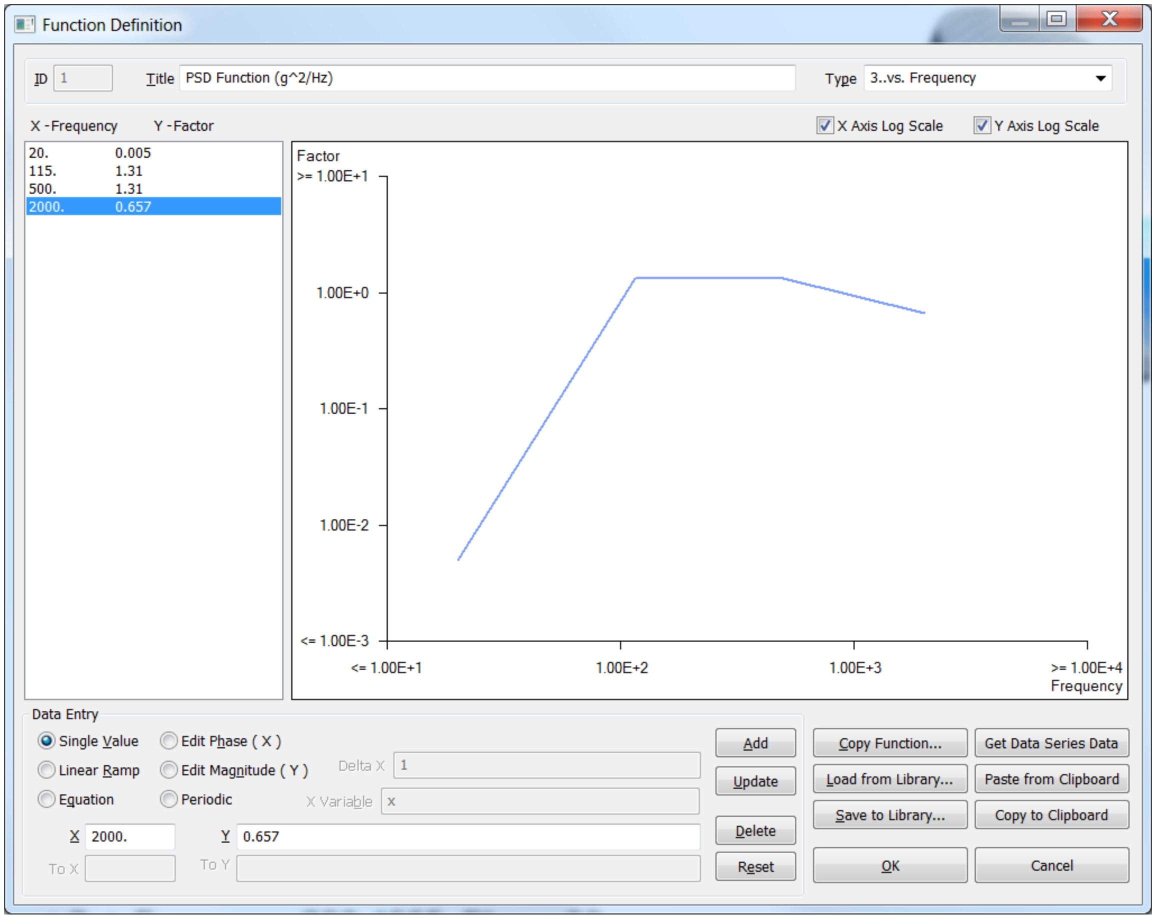
A PSD analysis is just a sophisticated and extremely useful form of the modal frequency analysis. Instead of having to interrogate multiple results sets, the PSD approach excites the structure using a broad-spectrum acceleration load and then nicely sums up the solutions into one single-valued result. One can also think of it as a white noise or broad-band excitation where the structure is excited at all frequencies simultaneously. The approach is statistical and the displacement and stress results (and anything else) are termed 1-σ responses. The sigma (σ) refers to a Gaussian distribution where 1-σ to 3-σ refers to 68, 95 and 99.7% ranges. At 1-σ you have a 68% chance that the stresses are within this value. For many applications, one is required to use a 3-σ interpolation and thus your results are multiplied by 3x.

The actual theory behind the PSD approach is beyond this simple note but if one just considers that it is based on the response of the natural frequencies of the structure in the form of a modal frequency analysis and that the results are statistical quantities, you'll be in good shape for doing most basic PSD work.

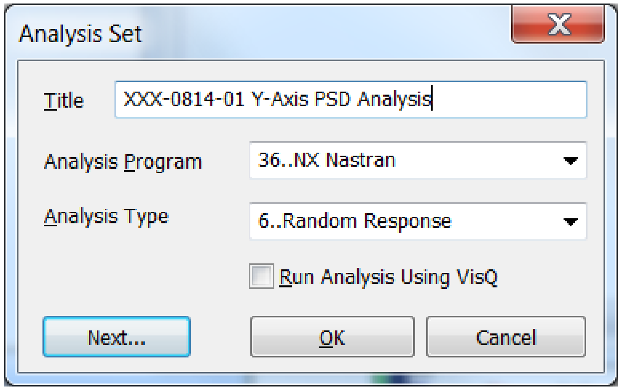
If you are interested in a tutorial just covering PSD analysis, then please take a read on our white paper:

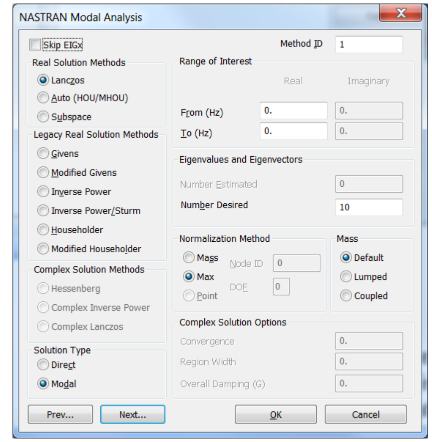
PSD Random Vibration Tutorial for Femap and NX Nastran.pdf

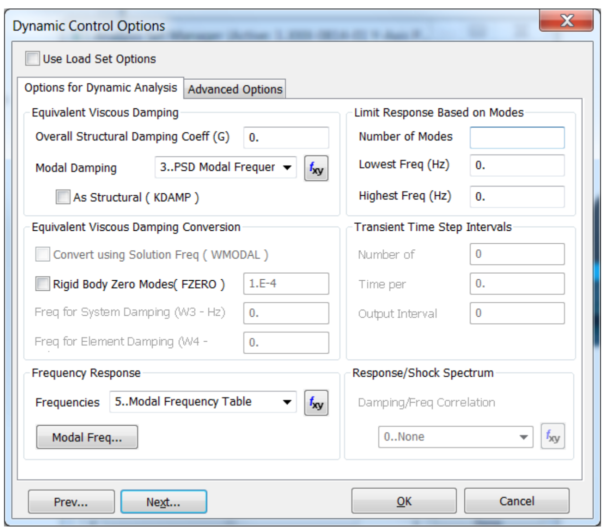
For completeness, let's do a simple PSD analysis on our circuit board. Again, units are very important. The PSD spectrum (load) is given as g^2/Hz. In the center of the spectrum from 115 to 500 Hz, the PSD input is 1.31 g^2/Hz and then tapers.

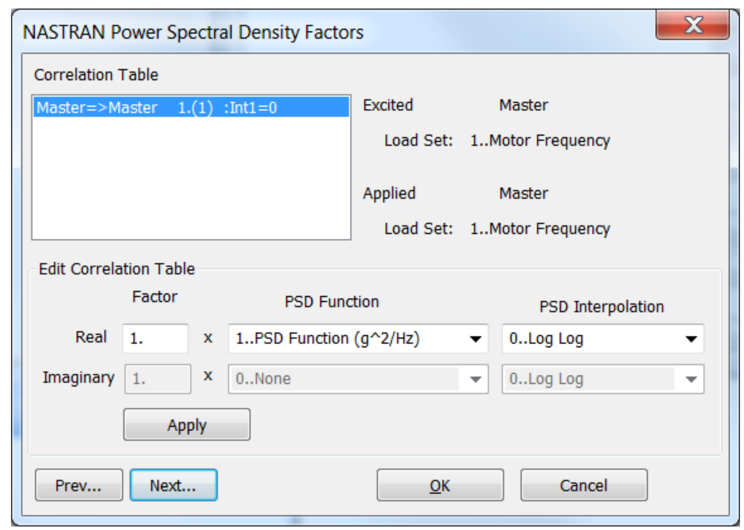


The PSD procedure is almost identical to the modal frequency analysis. There are some new screens but the only critical one is where you apply your PSD spectrum (lower-right-hand-corner). Otherwise, it is identical to that of the prior modal frequency analysis.

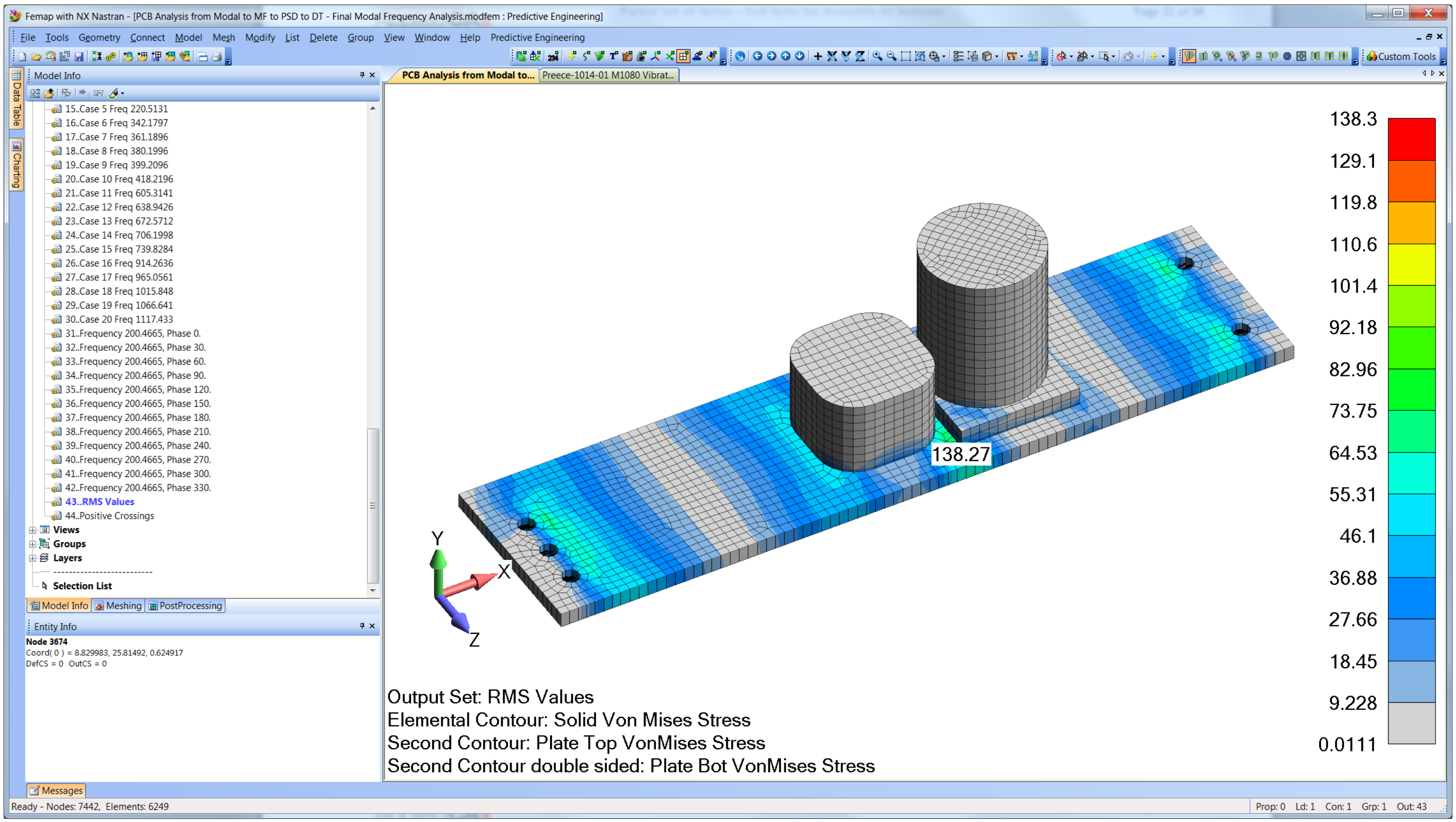








Given that a PSD analysis can be a numerically intensive calculation, Femap provides the ability to restrict your analysis output to just a few items or the complete model. For this analysis, all output requests are left blank except the very last screen where just displacements and stresses are requested. This is identical to that which was done for the modal frequency analysis. At the end, we have the RMS von Mises stresses contoured over the system and they are significantly greater than just the modal frequency result.

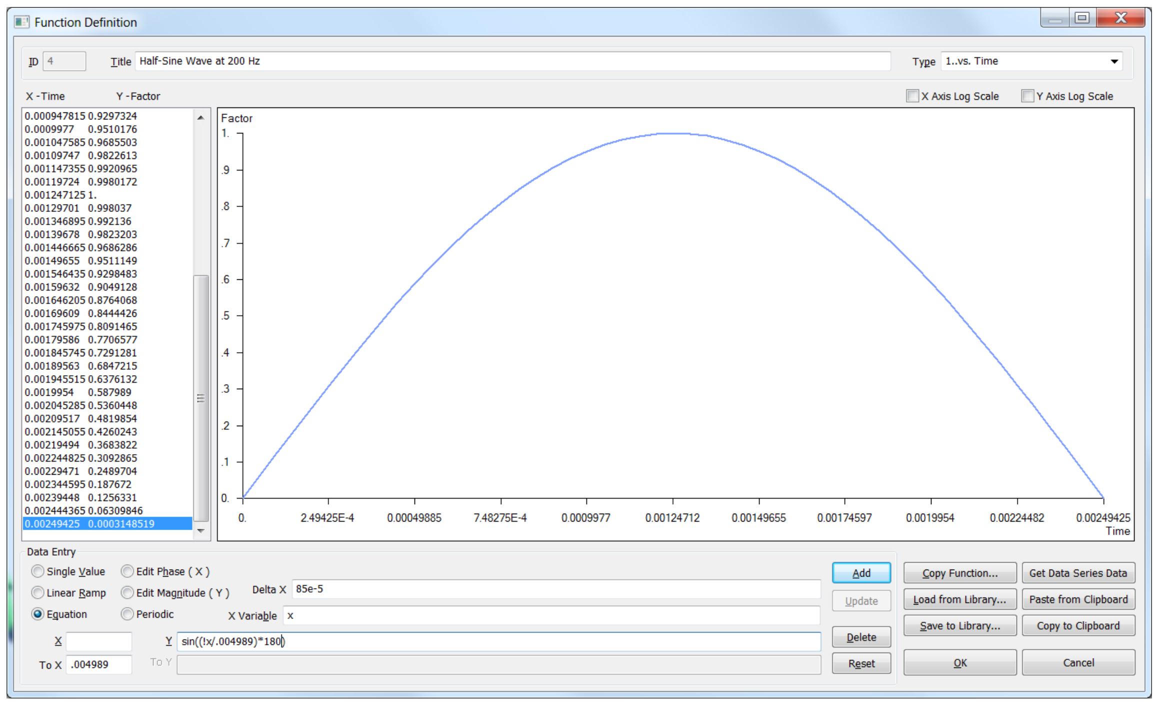


6. DIRECT TRANSIENT ANALYSIS

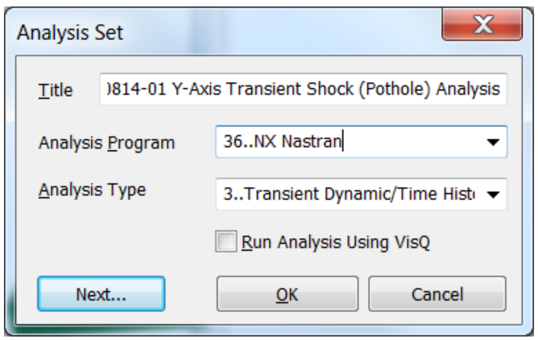
Sometimes you just want to whack the structure and not mess around. In this scenario, we are going to hit the circuit board with a 100 g pulse at a frequency of 200 Hz in the Y-direction (one can detect a theme to this seminar?). The procedure just requires a function for the hit and then a few setup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

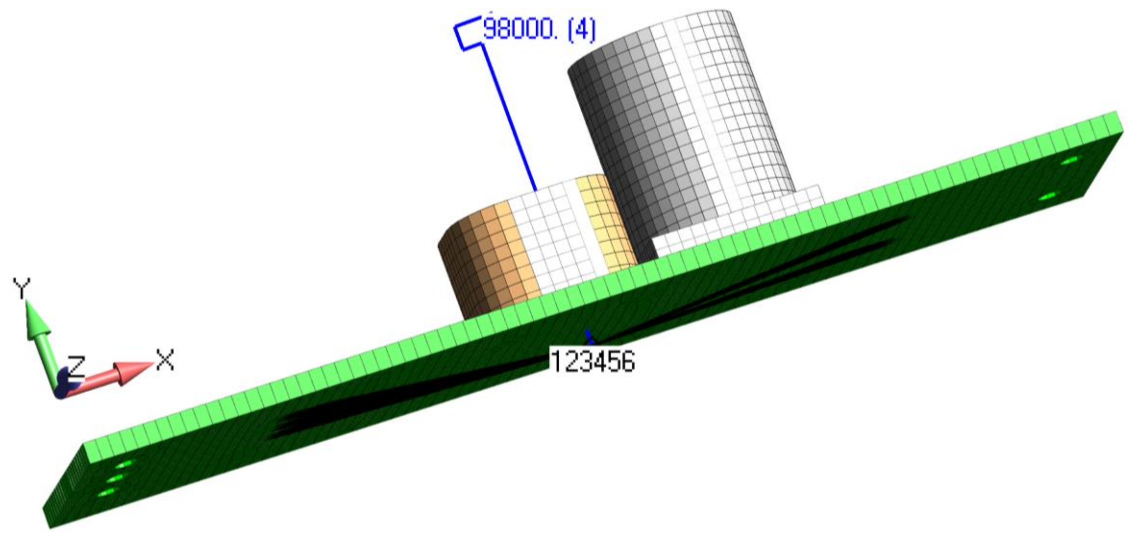
$$F_o(t) = m\frac{\partial^2 u}{\partial t^2} + c\frac{\partial u}{\partial t} + ku$$

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

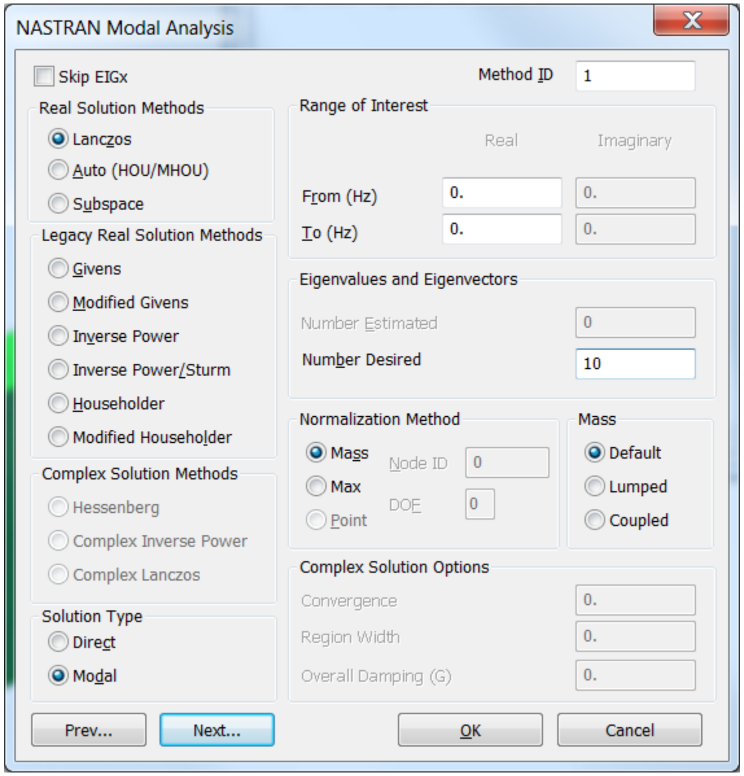


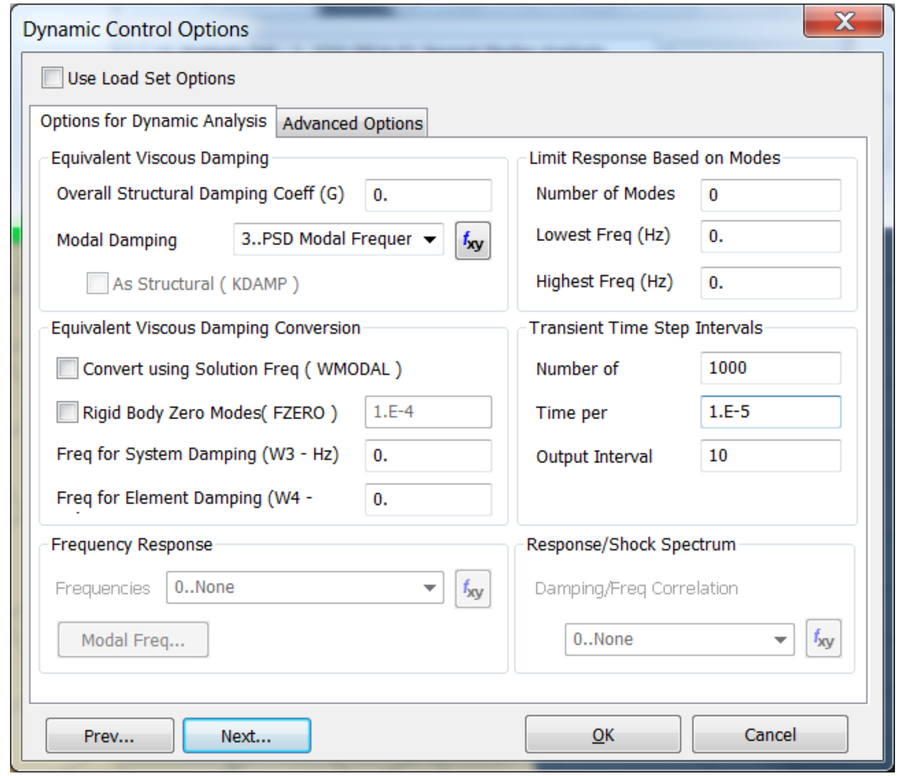
For our work, we are just going to use half the sin wave to give the system a shock pulse. The load for this analysis is 100 g (98,000 mm/s^2) with our half-sine function at 200.4 Hz.



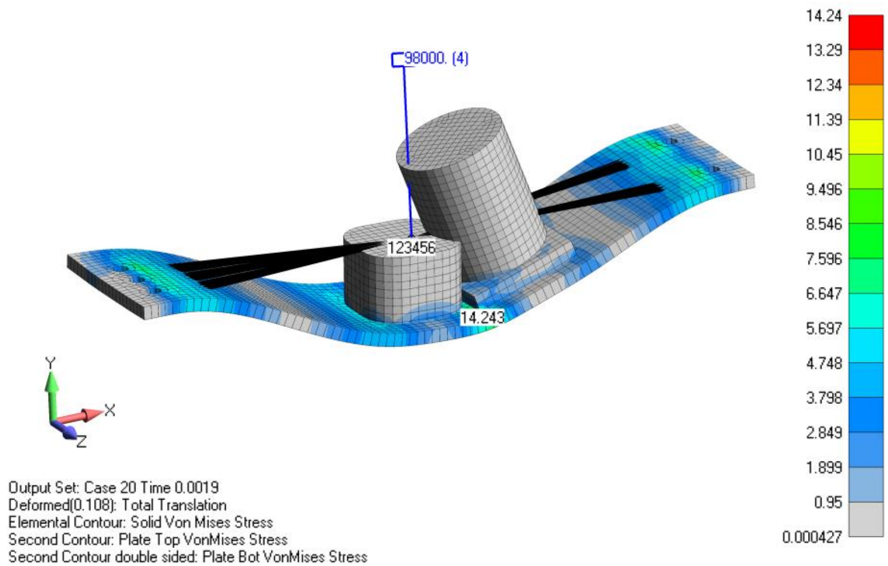


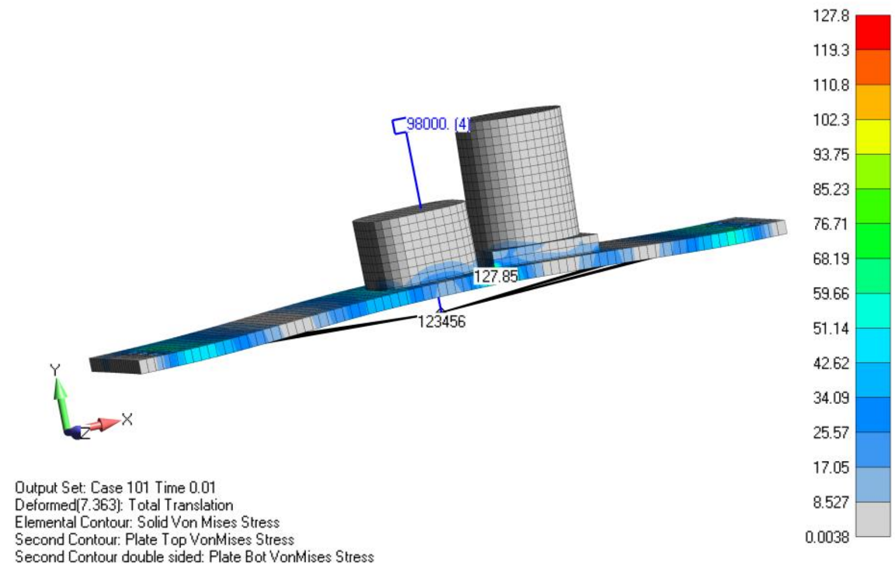
Our transient analysis is based on the first ten Eigenvalues and Eigenvectors.



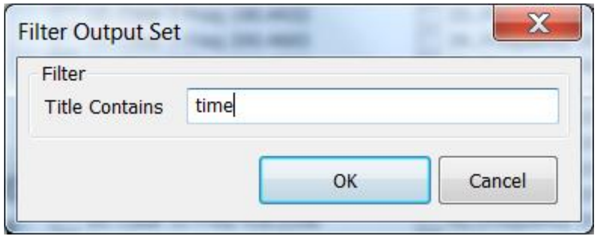


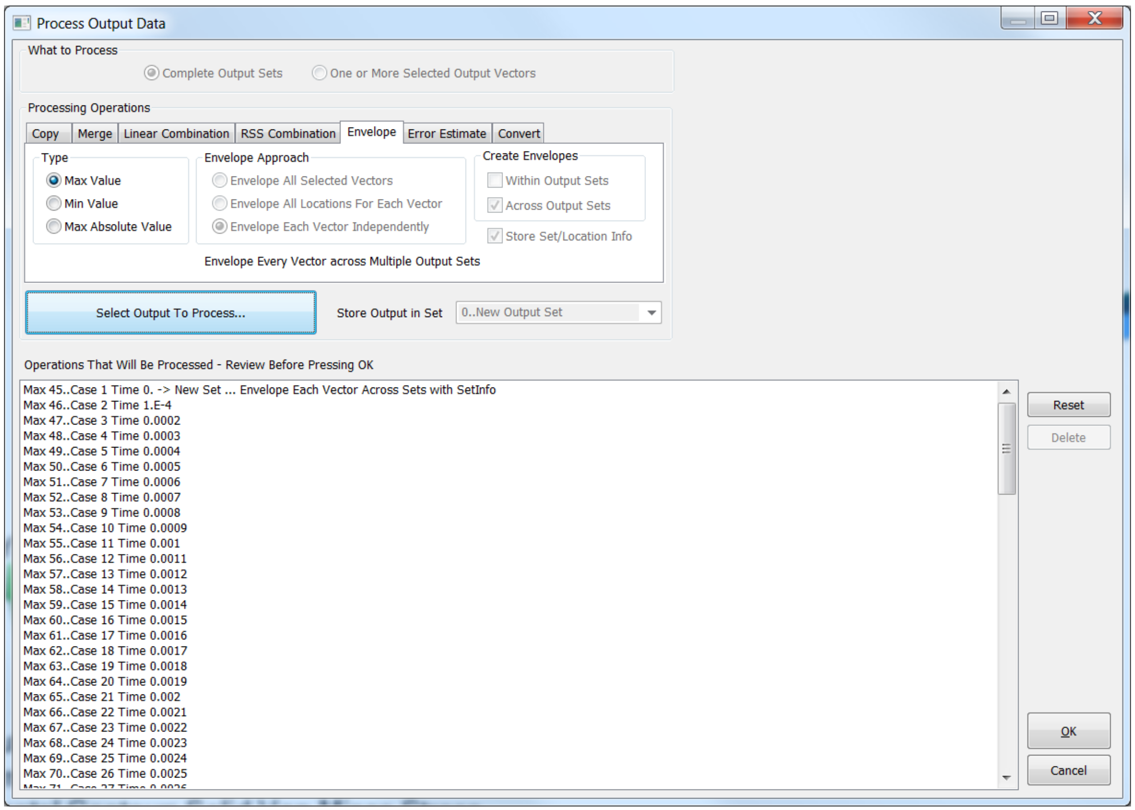
At the end of the simulation, one has a hundred result sets to claw through.



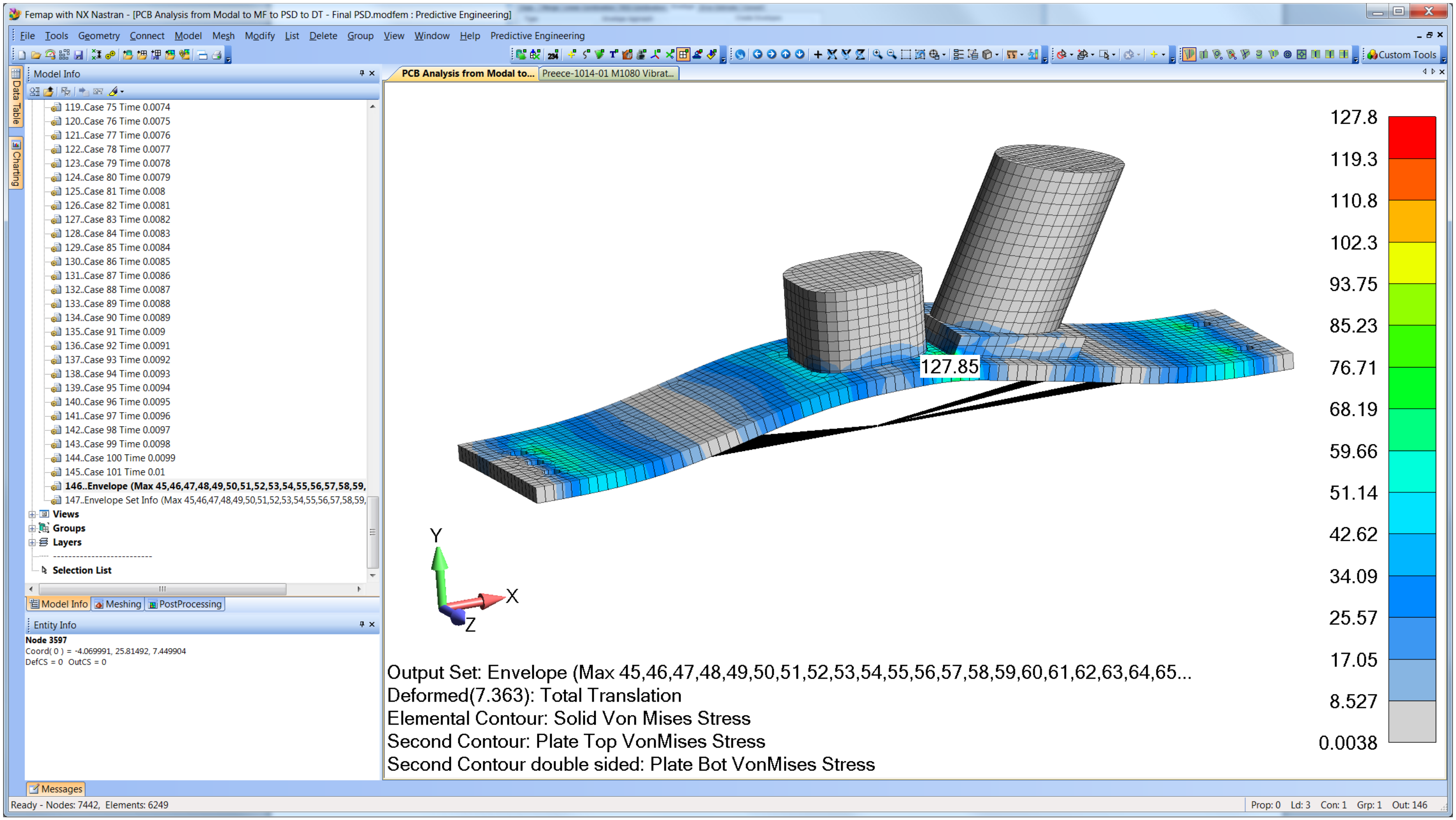


A much simpler way to process transient results is to use the Femap envelope function and then select all the output sets with “time” in the title:





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



7. QUESTIONS AND ANSWERS ABOUT FREQUENCY ANALYSIS

Question: What happens when a structure is loaded by harmonic load that is below the structures lowest natural frequency?

Answer: Let's say that we have a transmission where the motor has an operating speed of 1,800 RPM (30 Hz). The transmission's first natural frequency is 36 Hz (20% margin since we don't really trust our FEA results). The transmission is stable and the applied load has a magnitude effect equal to that of a static load.

Question: I have a very small natural frequency number (i.e., <<0.1), what happened?

Answer: Well, most likely you have something not constrained and NX Nastran is telling you that you have a rigid body motion. If one animates this frequency, one will see the complete model moving. Note: A structure that has no constraints or a constraint set attached to the solution, will have six low-number natural frequencies and likewise, if you have a part within your model that is not attached, it will exhibit a low frequency mode (rigid body motion). This is a super effective trick to find lose parts in your model that would cause a static stress analysis run to fail.

Question: (We'll add additional material here from questions asked during the Seminar)

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

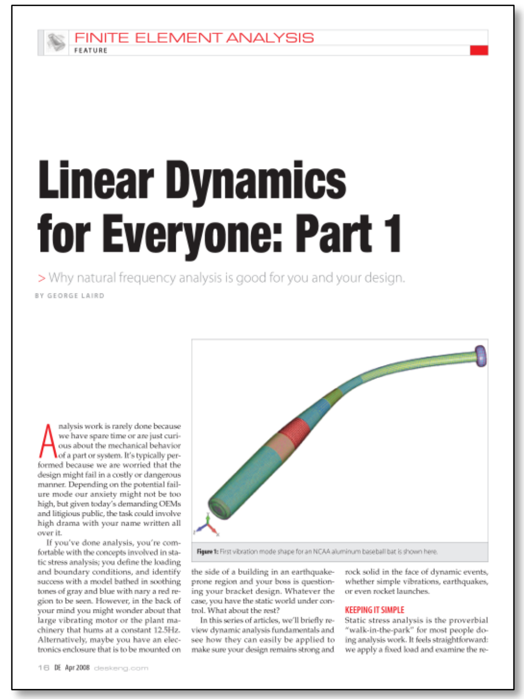
8.1 CHECK F06 FOR MASS SUMMATION AND KNOW WHAT YOU KNOW

Although this is just another check, we wanted to let you guys know

FO6 Check-Out Basics

- Do the element types and numbers make sense?
- Does the model mass exactly match that reported in the "OUPUT FROM GRID POINT WEIGHT GENERATOR"?
- 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

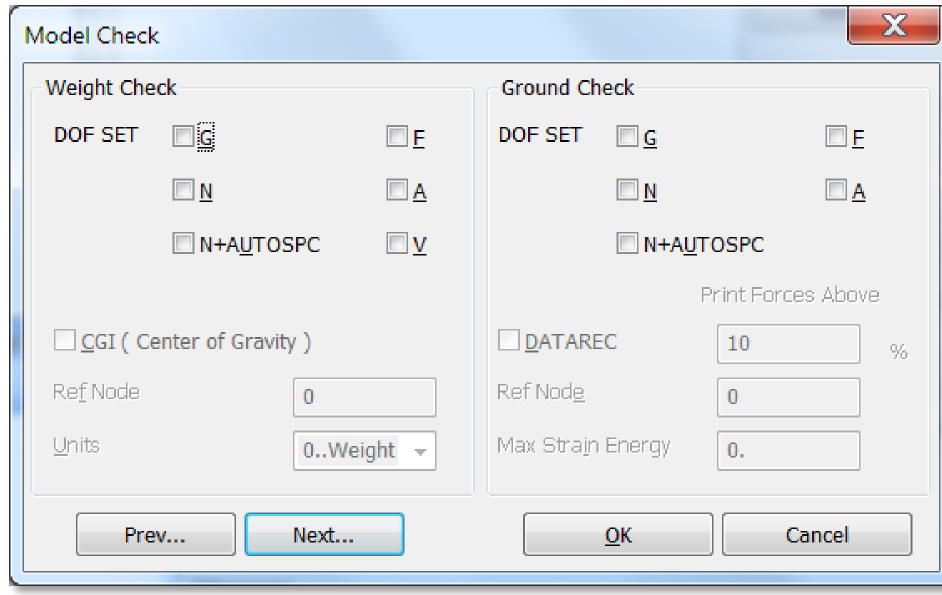
Static stress analysis is the proverbial "walk-in-the-park" for most people doing analysis work. It feels straightforward: we apply a fixed load and examine the re-

18 DE Apr 2008 deskeng.com

For more information see our Seminar: Normal Modes Analysis

8.2 GROUND CHECK IF YOU ARE DOING AEROSPACE QUALITY WORK

This check-out technique provides a numerical proof that your stiffness matrix is up-to-snuff. It is a rather dry subject and we'll leave it up to the seminar to flesh-out exactly how to do Ground Check, but if you have ever wondered what this screen does – this is your opportunity.



For more information see our Seminar: What is Groundcheck?

9. ADDITIONAL READING

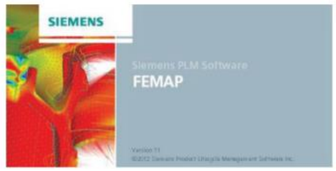
- Linear Dynamics for Everyone.pdf
- Femap and NX Nastran Technical Seminar on Vibration Analysis for Engineers.pdf
- PSD Random Vibration Tutorial for Femap and NX Nastran.pdf
- NX Nastran Dynamic Analysis.pdf
- Vibration Analysis for Electronic Equipment, Dave S. Steinberg

10. TRAINING OPPORTUNITIES

Femap & NX Nastran Training

Foundation | Advanced | Customization

AppliedCAx We do this every day



When: October 20-24, 2014 (Monday-Friday)

Where: Portland, Oregon

Cost: Foundation and Advanced training (Mon-Thurs) is $2,100 per student with the optional Customization/API training on Friday for an additional $525.

What's Included: Course manual and workshop videos saved to a USB stick. A lunch and social event are provided to encourage class interaction with fellow users.

Registration: Early registration is encouraged since space is limited to 18 students and it is expected that the class will fill. *To register please send email to:*

Training@PredictiveEngineering.com
Attn: George Laird, PhD, PE

About Predictive Engineering
Based in Portland, Oregon, Predictive has more than 18 years of experience with Femap, Nastran and LS-DYNA and is well known as the "go-to-company" for Femap training. References can be found on our website: **www.PredictiveEngineering.com**

Welcome Femap and NX Nastran Colleague,

This week-long course taught by **Predictive Engineering** will take the new user from ground floor through FEA best practices to advanced subjects dealing with manifold and non-manifold surface modeling, detailed plate meshing and tet versus hex meshing. The final day will finish with a focus on customization and automation using Excel and Femap's own API interface. The course will be fast paced and follow a workshop format with theory, practice and Q&A sessions.

Course Outline

Foundation of FEA Modeling with Femap + NX Nastran (Two Days)

I. FEA theoretical background w.r.t Beam, Isoparametric and special elements
II. Tour of Femap interface: Preferences, Panes, Toolboxes, Help and Tips & Tricks
III. Femap modeling workflow for Beam, Plate and Solid (BPS) elements
IV. Static stress analysis and results interpretation of BPS elements
V. Introduction to Plate and Solid modeling with surface and solid geometry and Mesh Toolbox
VI. Introduction to Assembly Modeling: Glued, Contact and Rigid element Usage

Advanced Femap + NX Nastran (Two Days)

I. Surface modeling using Manifold and Non-Manifold geometries
II. Advanced surface preparation for high-accuracy Plate modeling
III. Meshing toolbox tips and tricks with Jacobian optimization
IV. Building efficient assemblies via efficient Solid modeling (tet & hex elements) and Linear Contact
V. Introduction to linear dynamics (modal analysis tips & tricks)
VI. Non-linear analysis: geometric versus material non-linearity and best practices

Customization & Automation of Femap (One Day)

I. Automation of results processing via Excel
II. Introduction to Femap's macro capability
III. Introduction to Femap's API via Custom Tools
IV. Programming Femap's API

503-206-5571 | Training@PredictiveEngineering.com | www.PredictiveEngineering.com