AT11462

Shaking All Over – Using simulation to understand vibrating products

Allen Fowkes/Autodesk
Mfg Simulation

Learning Objectives

- Simplifying CAD for vibration analysis
- Understanding loads and constraints for vibration analyses
- Differences between simulation products for vibration
- Results interpretation of vibration simulation

Description

Vibration is a part of every design. It's up to us to decide how much to consider its effects. Autodesk Simulation tools include analyses to take into consideration a wide range of environments and design objectives. This class will review in detail the various workflows, input requirements, and output results for Autodesk Nastran In-Cad and Simulation Mechanical. By knowing the vibration analysis workflows, users can gain a great understanding of their products behavior in various environments.

Each analysis type has a unique type of output that will be covered. Of special attention will be for options that need to be setup prior to the analysis that has an effect on the output. Items like the "cluster factor" and response spectrum combination methods can be explored in-depth.

Customization is a key feature in Autodesk Simulation. Configuring the interface, custom geometry parameters, and optimized results (including modal frequencies) can be done. Flexibility is also imbedded to schedule analyses locally or on the cloud. By customizing the simulation software, a user can streamline different product workflows and enable creation of new ones that can improve the company works. The workflows might range from proving existing designs or modifying prototypes to reduce costs.

Your AU Experts

After completing his physics and master's degree in mechanical engineering, Allen worked in the dynamics area in the aerospace industry. In addition, Allen has over 30 years of finite element analysis experience. He is a registered professional engineer in Pennsylvania.

Simplifying CAD for vibration analysis

What's Important?

Lumped masses, beam elements, remote load applications, among others, can be used to simplify geometry to achieve the most efficient analyses, and usually the most appropriate. Simulating the mass effects of a battery in a system can be automatically included.

Geometric simplification

This includes lumping masses to a certain location, as well as overall looking at the stiffness of the model and how to get accurate mode shapes. Generally, simplifying models is the same as simplifying them for linear static analysis, but in dynamics, we are mainly concerned with the overall stiffness of the assembly or part.

FIGURE 1: THE FINISHED PRODUCT AND THE SIMPLIFIED MODEL

Other element types

Beam and plate elements can greatly reduce the time and effort of the project, while being very accurate. We will explore the various element types to be sure there are no questions unanswered.

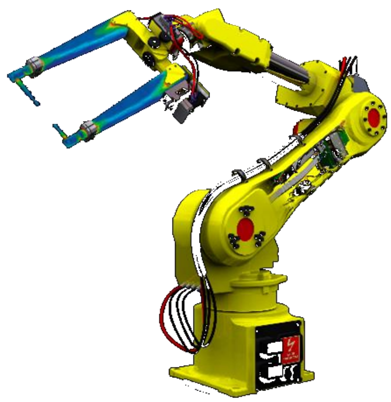

FIGURE 2: SPREADER ARM RANDOM VIBRATION ANALYSIS

Understanding loads and constraints for vibration analysis

The various input loads

The input loads are very different from the typical loads applied in a linear static stress analysis. They can be time dependent loads or loads that describe the type of environment the part is subjected to, such as a random vibration.

What constraints you can and can't include

Constraints are typically the same as in linear stress, except that contact is not included in linear dynamic analyses. Contact is not included in any linear dynamic analysis.

Differences between simulation products for vibration

Nastran In-Cad

Nastran In-Cad extends the capabilities beyond Inventor Professional by adding dynamic analyses. These are very comparable to simulation mechanical analyses, so we mainly recommend a more in-depth dig into these for specific options, or contact us.

Simulation Mechanical

Simulation Mechanical has a wide range of vibration capability which we will explore. We will dig into the specific options that are common in these types of analyses such as combination types in response spectrum analysis, but we won't go as far as comparing these to Nastran In-Cad. We will point you to the resources you need to look into any analysis options as necessary.

Results interpretation of vibration simulation

Analysis Input/Output options

Each analysis type has a unique type of input and therefore different output options that should be covered. Of special attention will be for options that need to setup prior to the analysis that has an effect on the output. Items like the "cluster factor" and response spectrum combination methods can be explored in-depth. How each analysis outputs results such as time dependent, individual modal contributions, and SRSS will be reviewed.

Additional results tips and tricks

We'll cover the various options when reviewing results. Tips and tricks for animation of results, how to view the specific outputs from the different analyses (SRSS response, individual modal results, plotting results, etc.).

Modal Analysis

Why run it?

This gives you a better idea of the stiffness of the part or assembly. Why not? Structures are exposed to vibration no matter what. With modal analysis you obtain a clear view of what obstacles are there. Lumped masses, beam elements, remote load applications, among others, can be used to greatly simplify geometry to achieve the most efficient analyses, and usually the most appropriate. Simulating the mass effects of a battery in a system can be automatically included with features such as "remote loads".

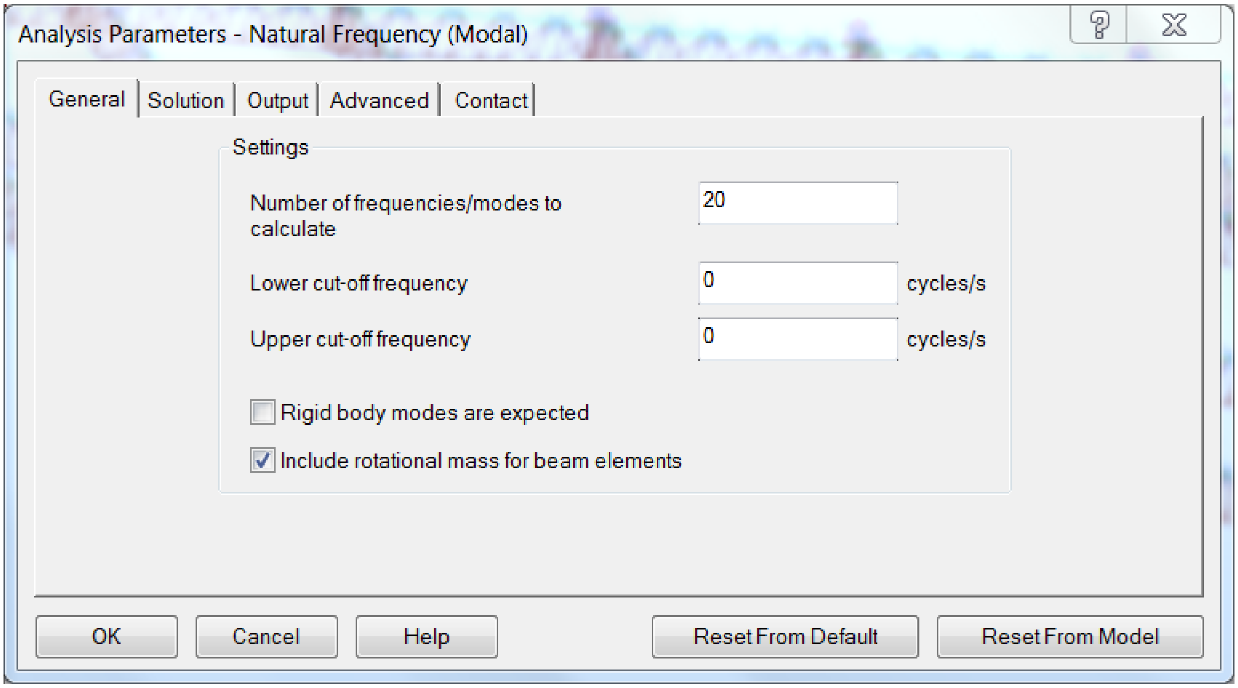

FIGURE 3: MODAL ANALYSIS CONTROLS

Modal analysis output

Mode shapes and natural frequencies are the main output, but this really sets you up for a successful dynamic restart analysis. This is where you do most of the work in determining how stiff or flexible you want your structure.

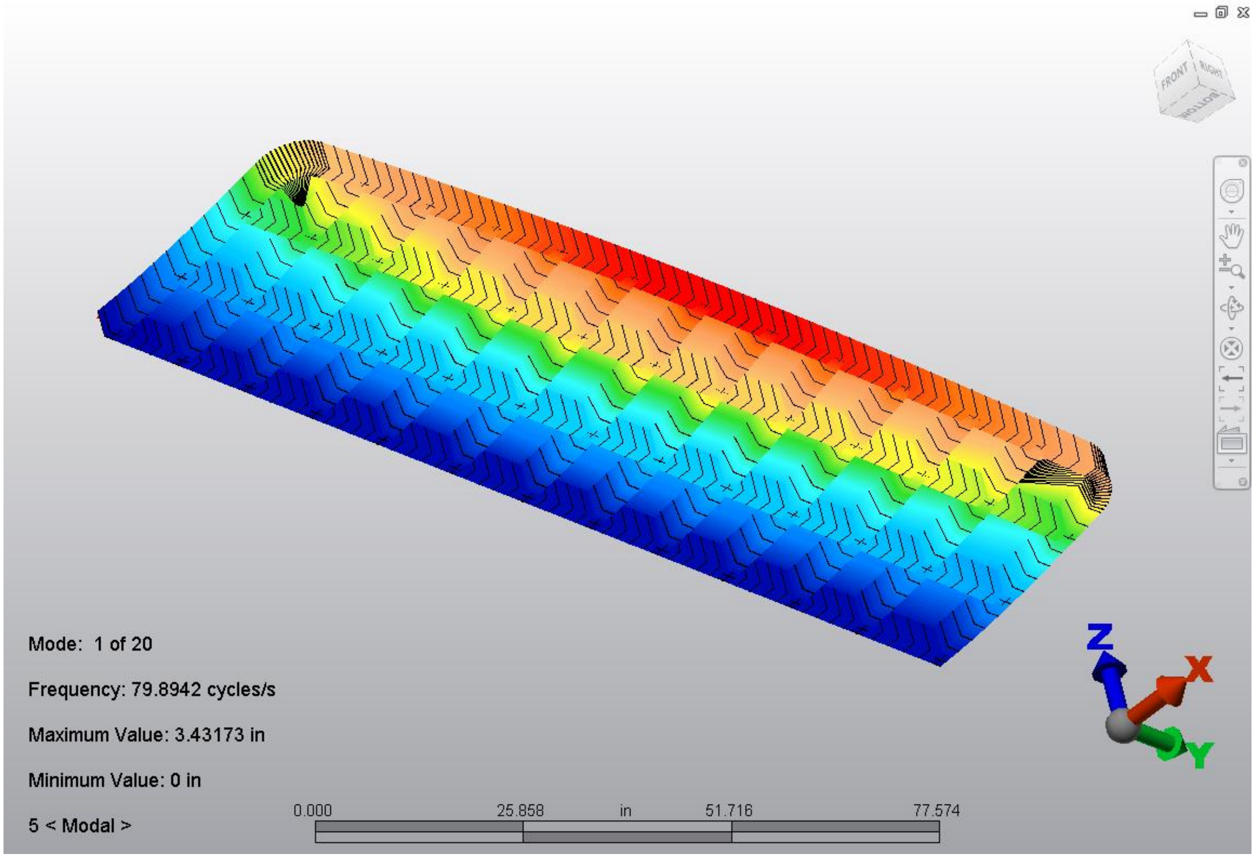

FIGURE 4: MODAL ANALYSIS MODE 1

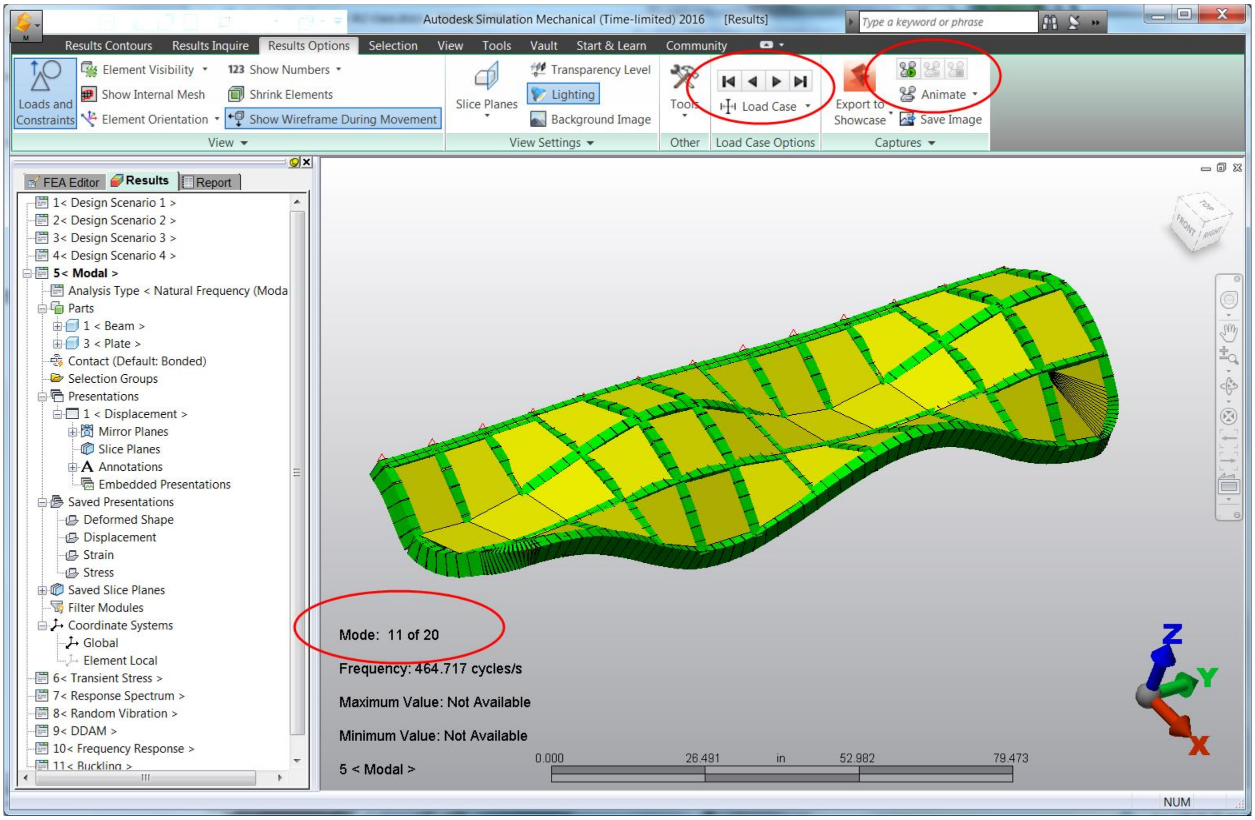

FIGURE 5: MODAL ANALYSIS RESULT OIPTIONS

Time History Analysis

When you want to simulate short-term events

Impact loads, jumping or walking on a structure or any other time dependent loads, can be input to see the behavior of the structure. Linear and nonlinear dynamic simulations can possible which we will describe by the end of this class. Damping can be applied to simulate real world situations.

Input

The loading is in the form of force versus time generally, but ground motion can also be applied. This is typical in a situation where acceleration is applied to the entire structure. Translation or rotational acceleration can be applied. An example would be accelerating a structure down a railroad or a rotational acceleration applied to a robotic arm (or pitcher). Control of the amount of output is easily setup.

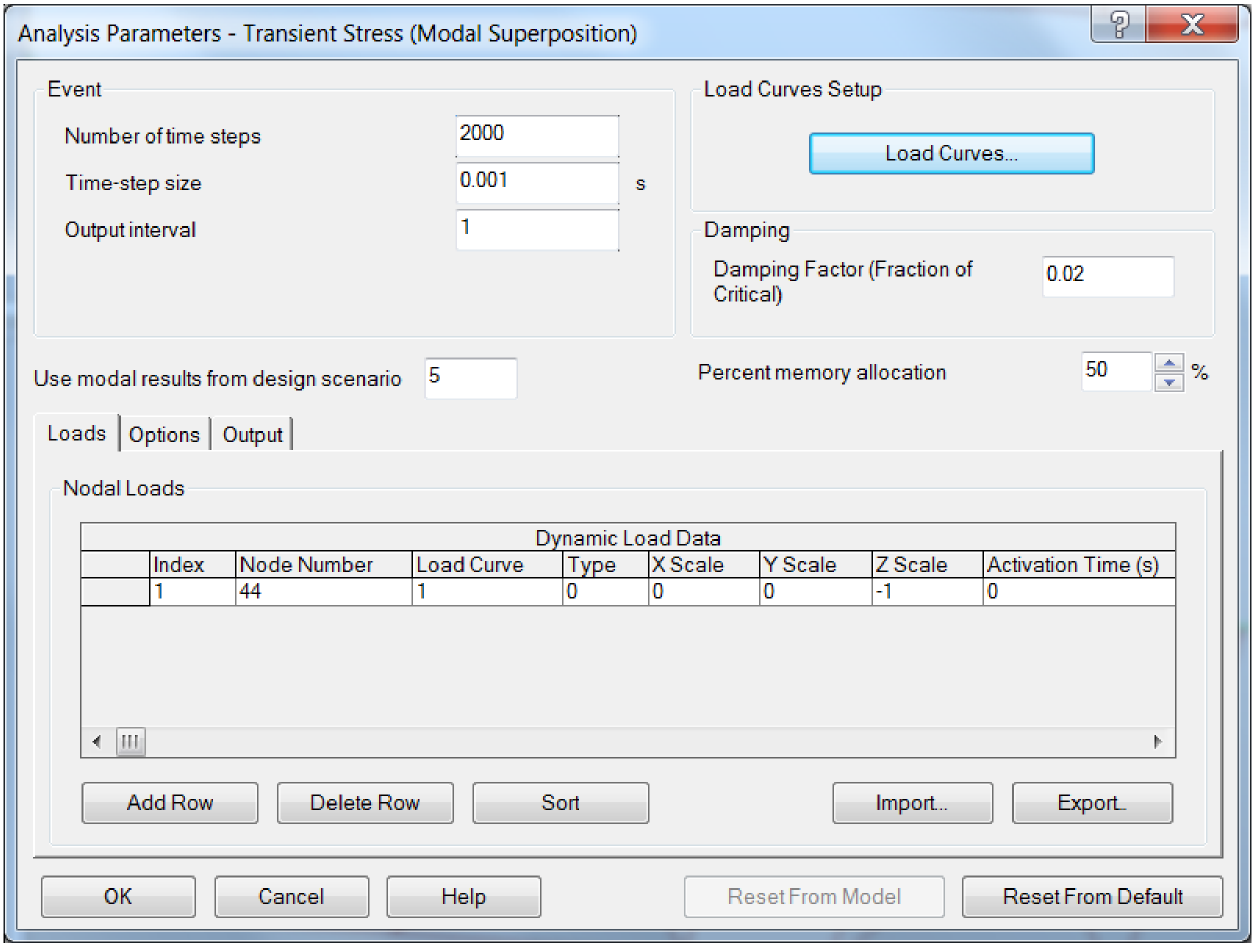

FIGURE 6: TIME HISTORY MAIN CONTROLS

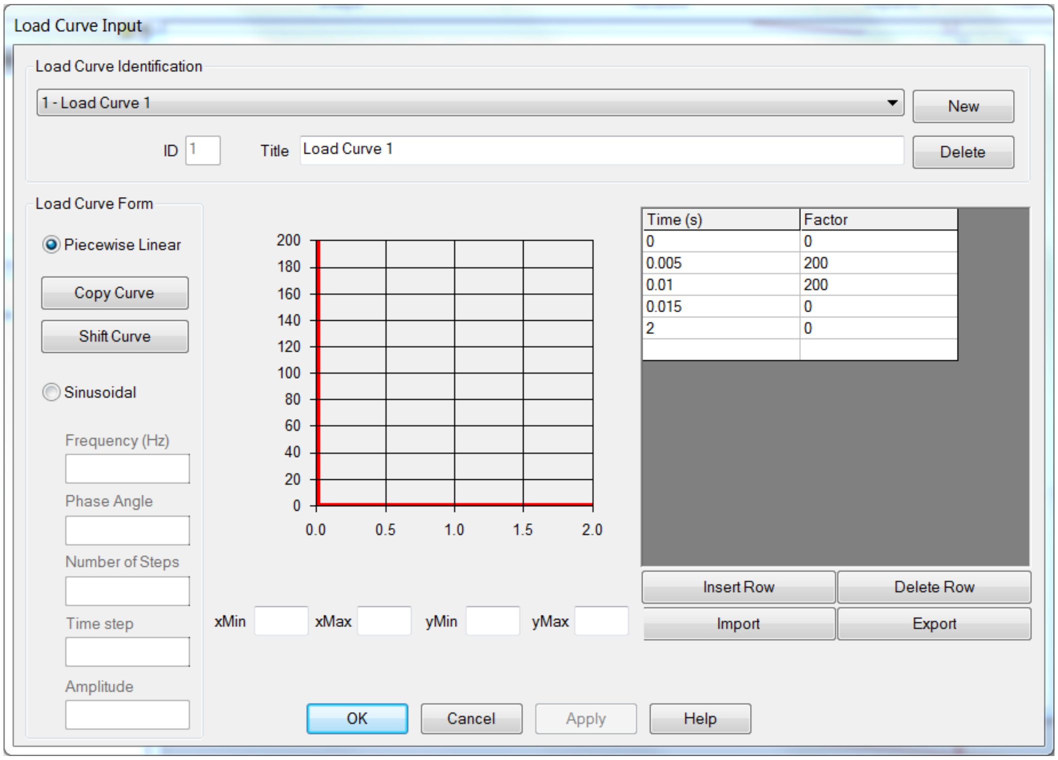

FIGURE 7: TIME HISTORY IMPACT CURVE

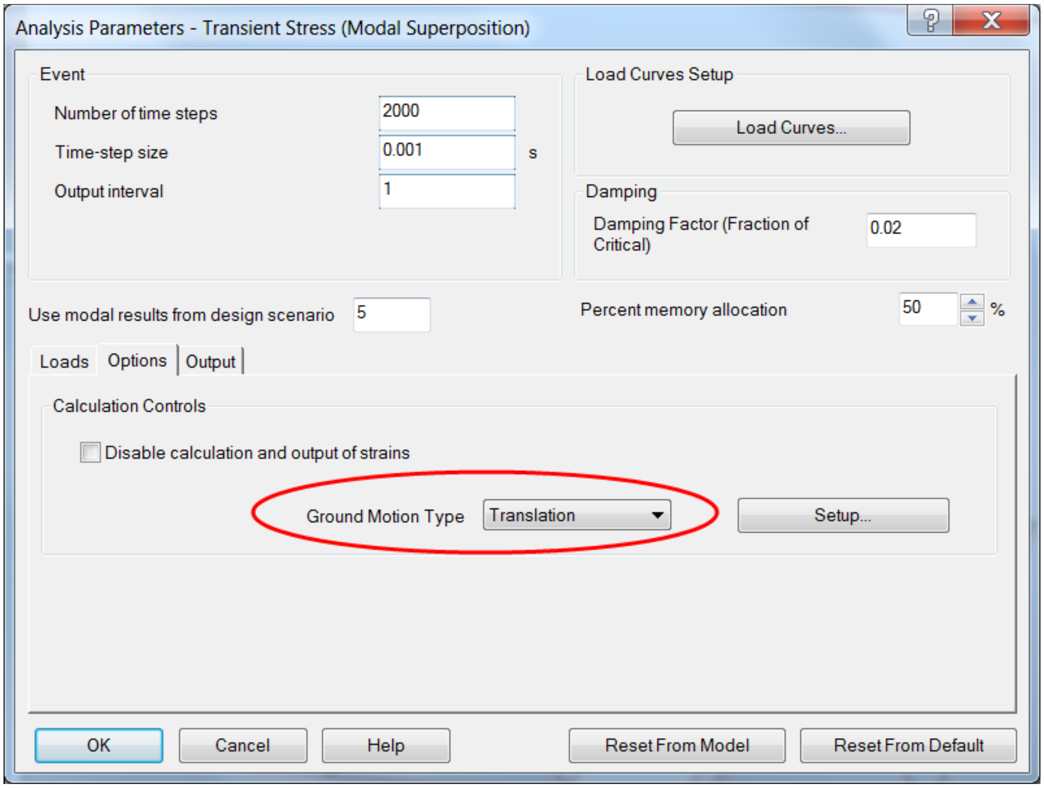

FIGURE 8: TIME HISTORY GROUND MOTION

Output

The output is displacement and stresses over time. Graphs can be generated easily to see the response of any area over time. Animations of the response over time can be easily viewed or output. At each time step, results can be viewed and output to a report.

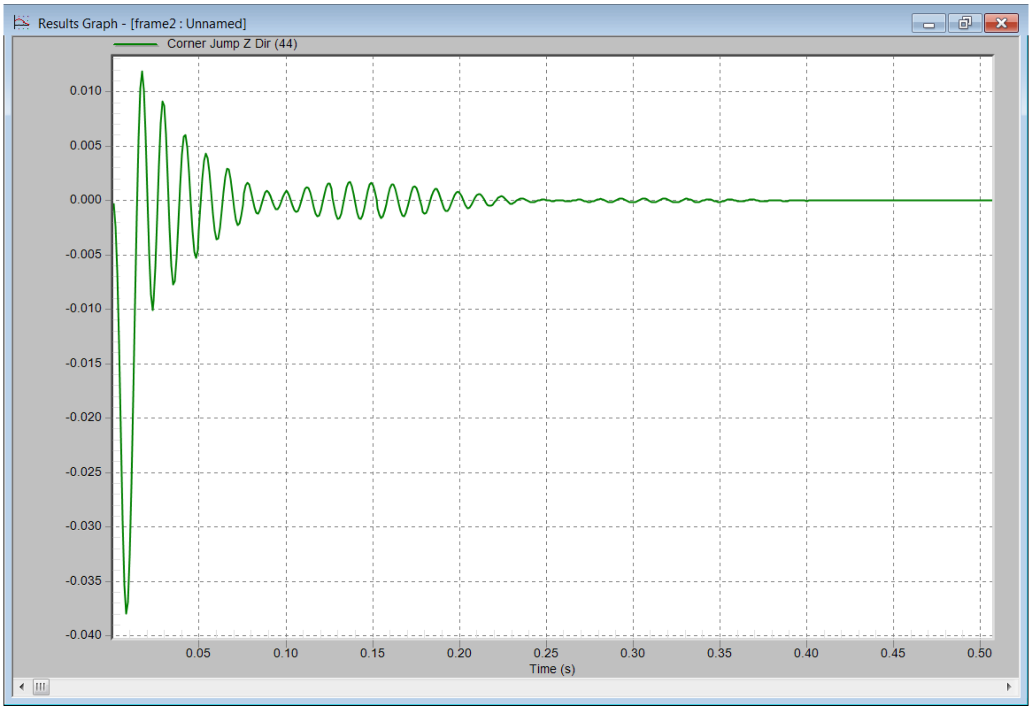

FIGURE 9: TIME HISTORY DISPLACEMENT OUTPUT

Response Spectrum Analysis

Shock and Impact applications
Earthquake, seismic activity, explosions, unknown events are some of the likely applications for a response spectrum analysis. This allows you to study the effect of the shock to the overall structure but also the individual modal contributions. Lumped masses, beam elements, remote load applications, among others, can be used to simplify geometry to achieve the most efficient analyses, and usually the most appropriate. Simulating the mass effects of a battery in a system can be automatically included.

Input
The loading is in the form of acceleration or displacement vs frequency or period. Otherwise known as shock analysis, this input represents the amplitude given to any particular mode shape when excited. A single degree of freedom input is automatically generated.

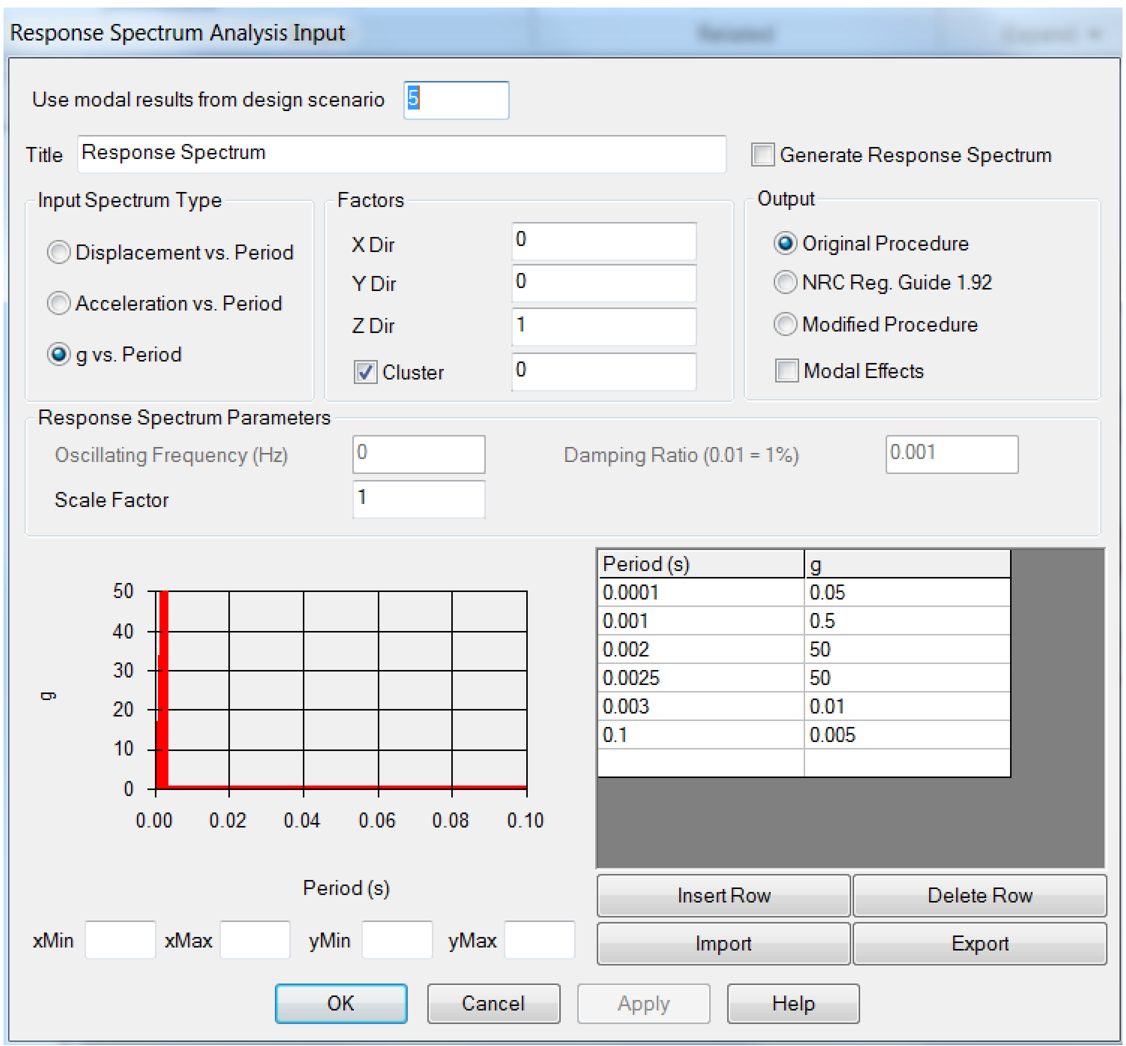

FIGURE 10: RESPONSE SPECTRUM INPUT AND COMBINATION OPTIONS

Output

An SRSS of the individual modal contributions is output to show the maximum response. Individual modal responses can be viewed. Output for modal participation factors is output to log files for report generation. Different combination methods are available for the SRSS result. Note that the below image is all positive displacement (after squaring all modes).

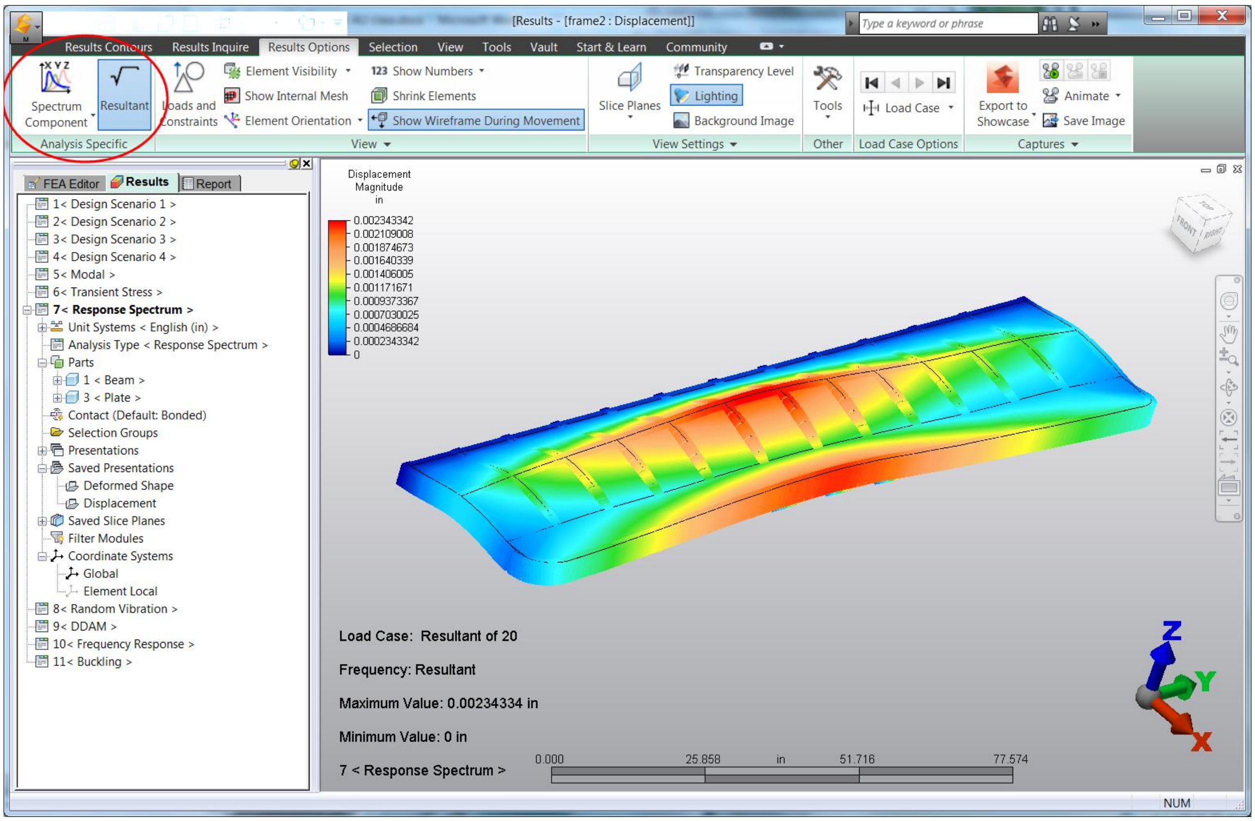

FIGURE 11: RESPONSE SPECTRUM RESULTANT MAXIMUM DISPLACEMENT

Random Vibration Analysis

White Noise to Specific Inputs

Random vibration lets you study everything from a white noise to a specific spectrum that mimics the environment. Random vibration from operating the boat as sea, or pulling a trailor on the road can be analyzed. This is even more reason to simplify the model to its main aspects. Again, which modes contribute to the overall response can be determined. Many qualification tests are performed using random vibration analyses and tests.

Input

The loading is in the form of q^2/hz (or acceleration^2) vs frequency excitation. This input is into the entire structure typically to simulate whatever vibration environment exists. A typical analysis may simulate a random vibration test where the input is fully known or one where a typical environment is an extreme.

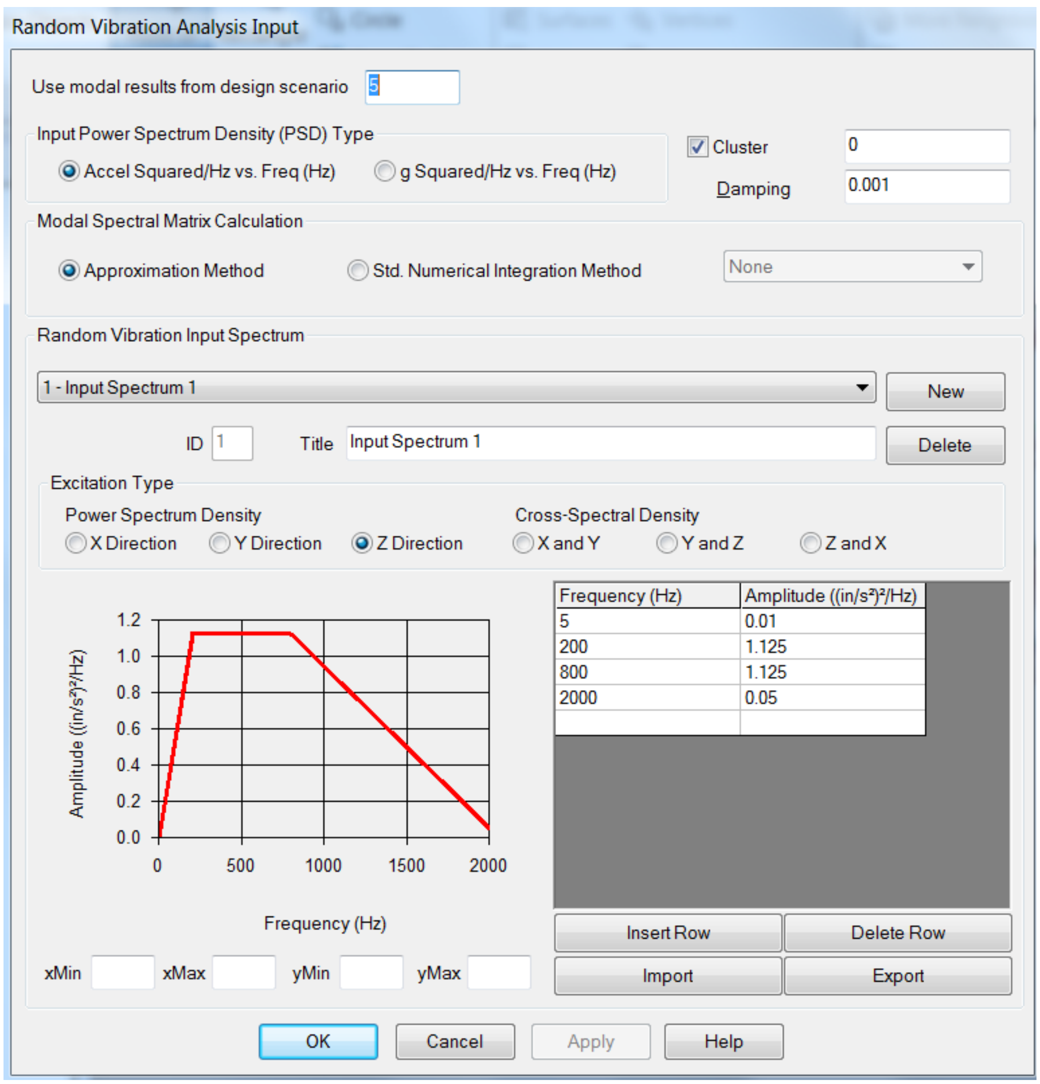

FIGURE 12: RANDOM VIBRATION INPUT INFORMATION

Output

The output is an SRSS of the individual modes Displacements and stresses are available. Being an analytical tool, random vibration lets you look at what the maximum vibration amplitude will exist during exposure to the loading.

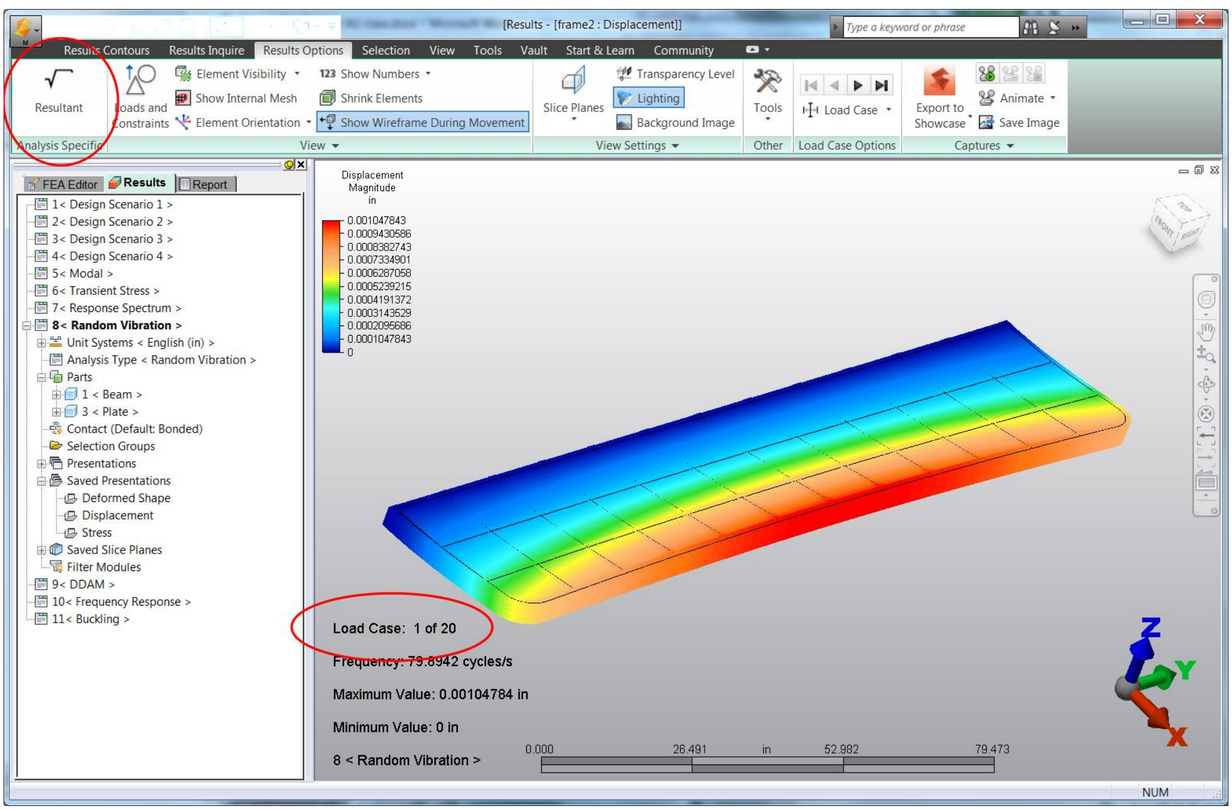

FIGURE 13: RANDOM VIBRATION INDIVIDUAL MODAL RESPONSE

DDAM Analysis

Dynamic Design Analysis Method Inputs
Input accelerations are calculated internally according to this method and input to response spectrum analysis. Alternatively, an input spectrum can be read in for use.

Input
The loading is in the same form as the previously described response spectrum analysis. The input spectrum type can be selected from a few choices, allowing for the conservative analysis result.

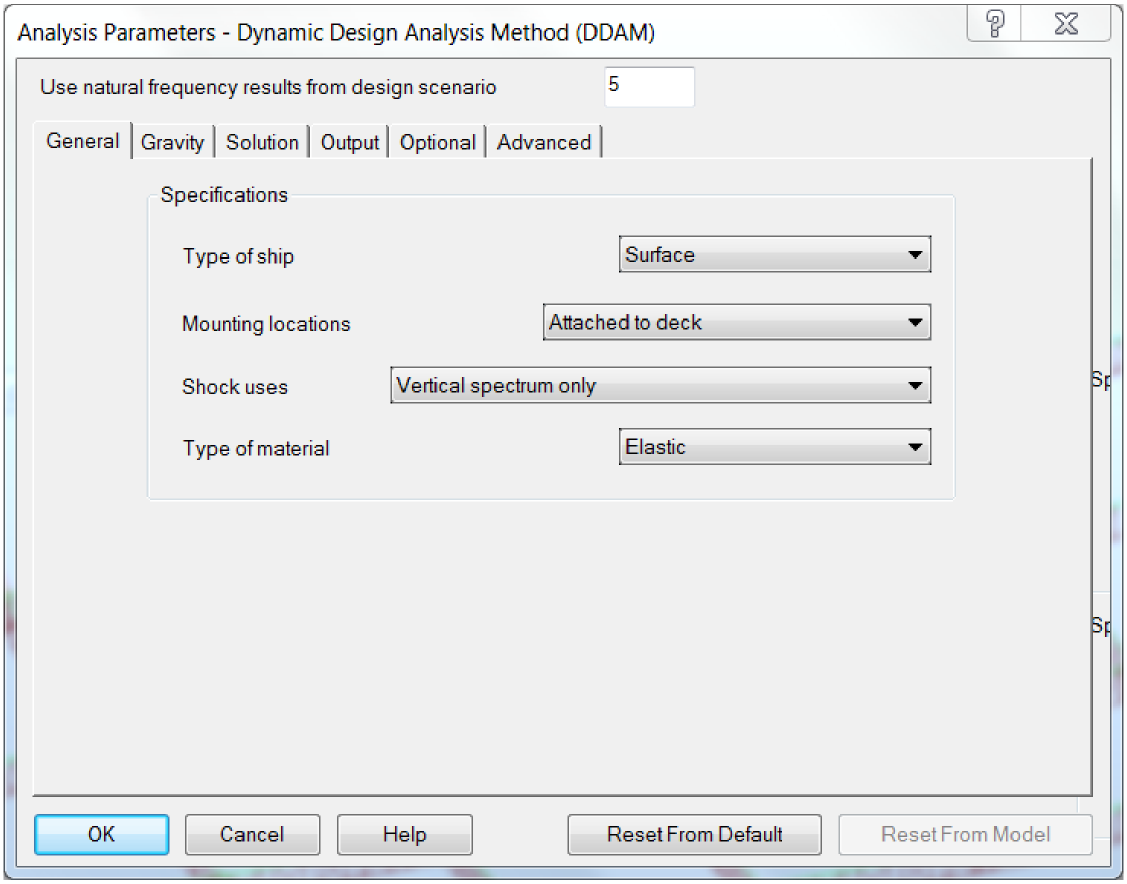

FIGURE 14: DDAM INPUT SETUP SCREEN

Output
The in-phase and out-phase are output as well as the SRSS of the 2 are output for each exciting input frequency. An In-phase and/or Out-phase plot can be output for a range of input frequencies. Additional output includes mass participation factors to show whether enough modes have be included in the solution.

Frequency Response Analysis

Typical Applications
Inputting just the engine frequency into the structure to simulate idle conditions is perfect. We want to be sure that the deck is not exciting any modes, but also want to be sure the it's not exciting any harmonics along the way. Trying out the operating frequency is not a bad idea either.

Input

The loading is in the form of a frequency and amplitude that I usually refer to as a*sin(w*t). You can input an exciting frequency at certain locations, or input a "sine sweep" excitation which is a set of frequencies. Amplitudes and damping can be input for a range of frequencies.

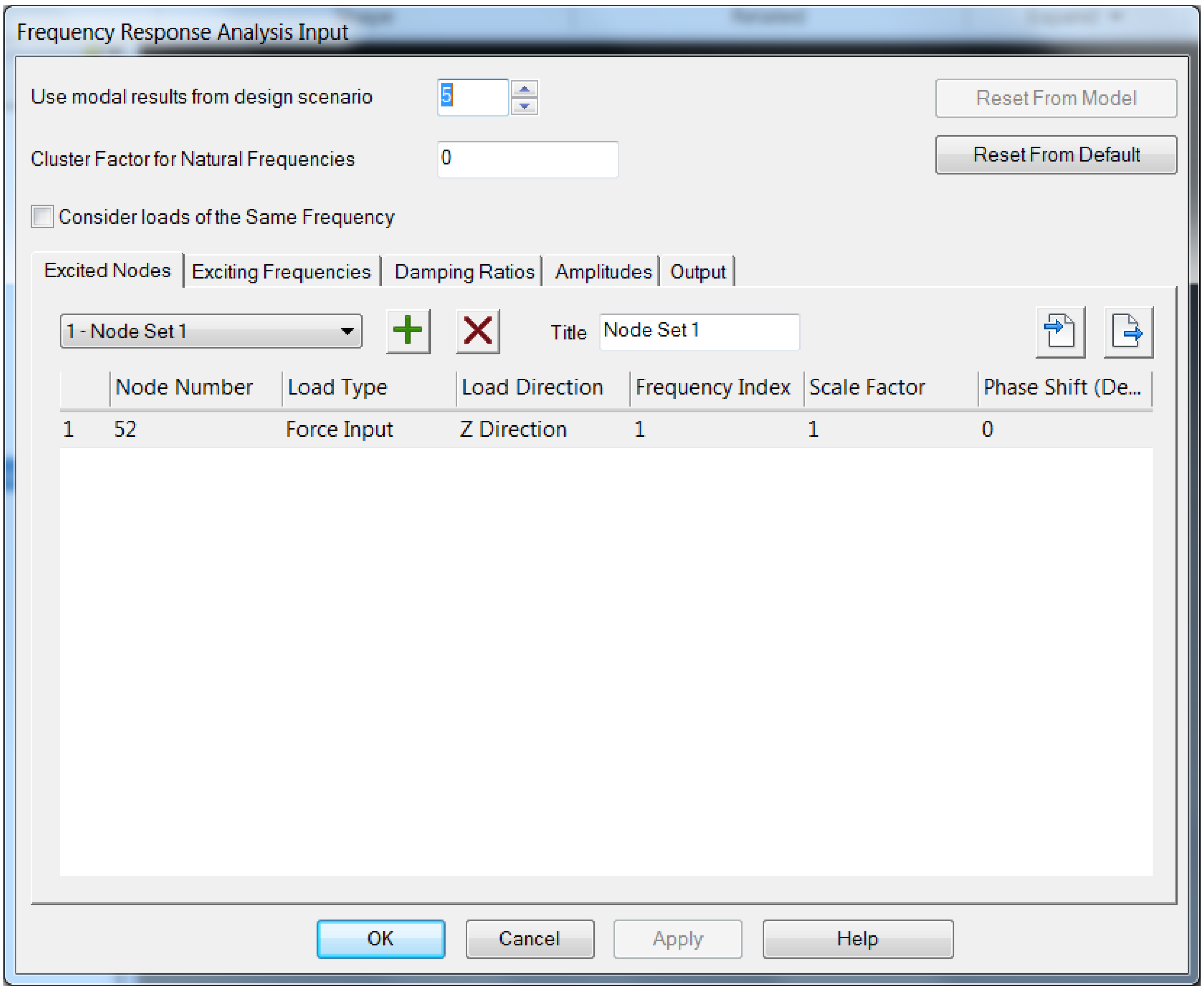

FIGURE 15: SETTING UP THE INPUT LOCATION

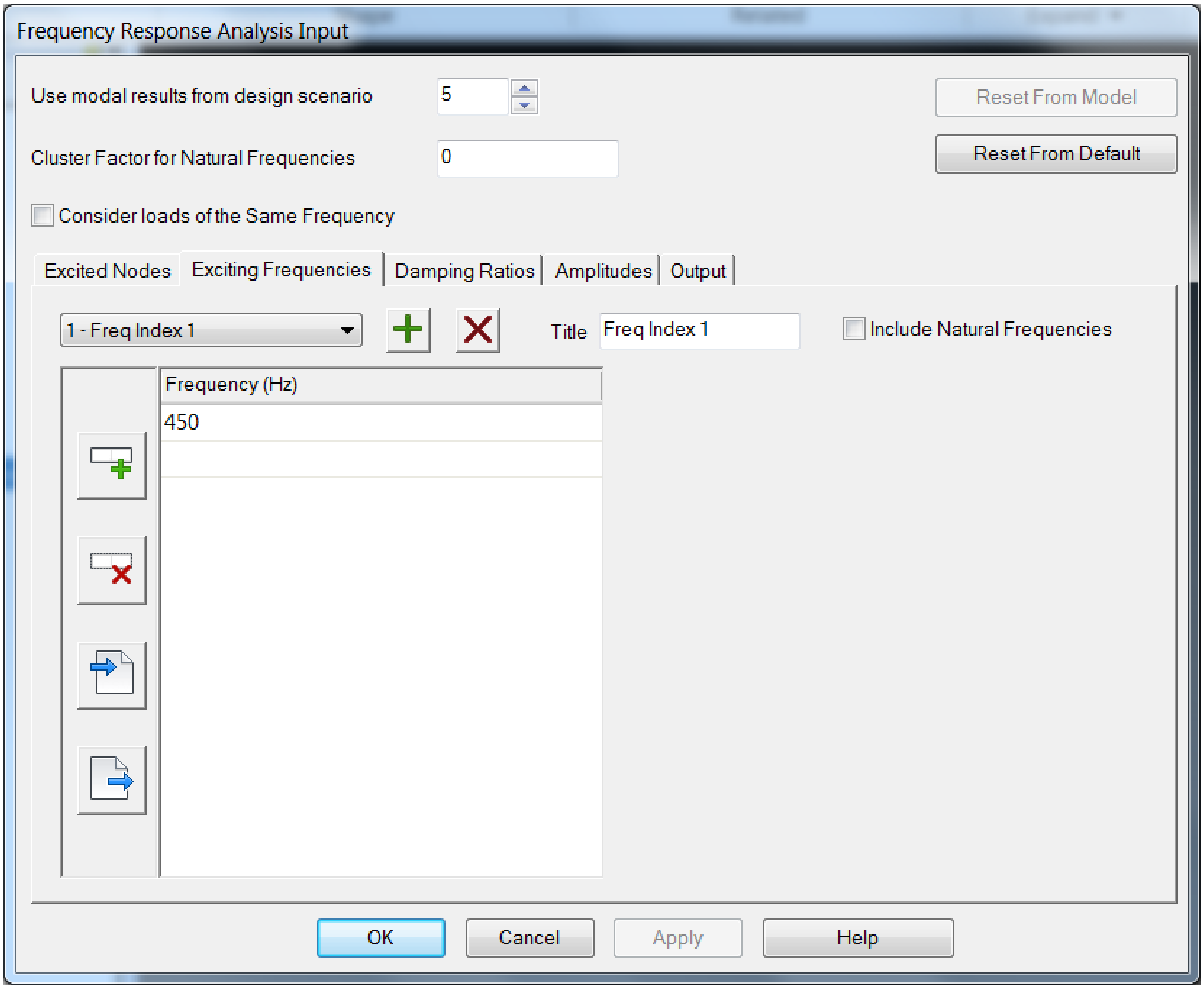

FIGURE 16: FREQUENCY RESPONSE EXCITING FREQUENCIES

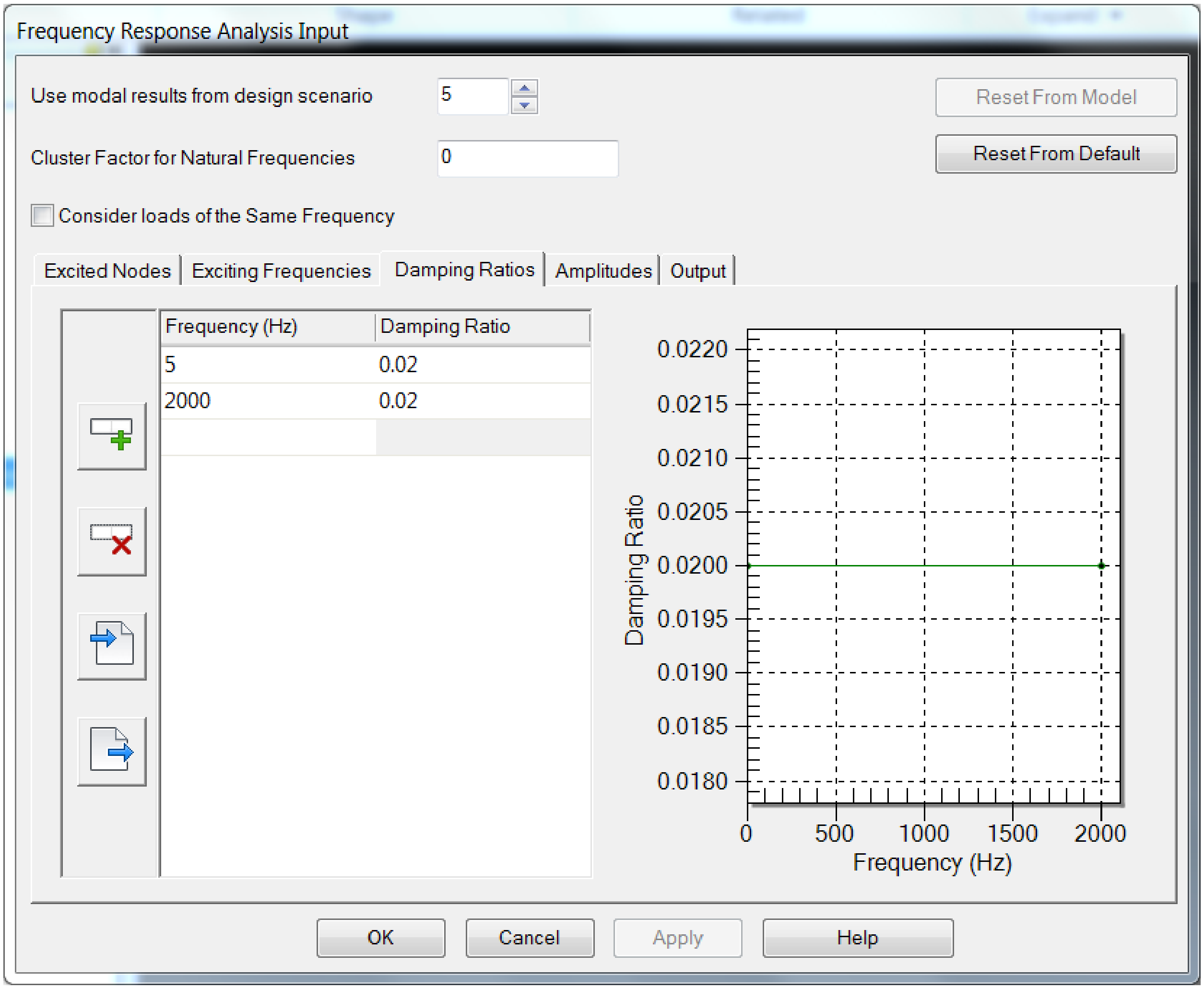

FIGURE 17: DAMPING RATIO INPUT

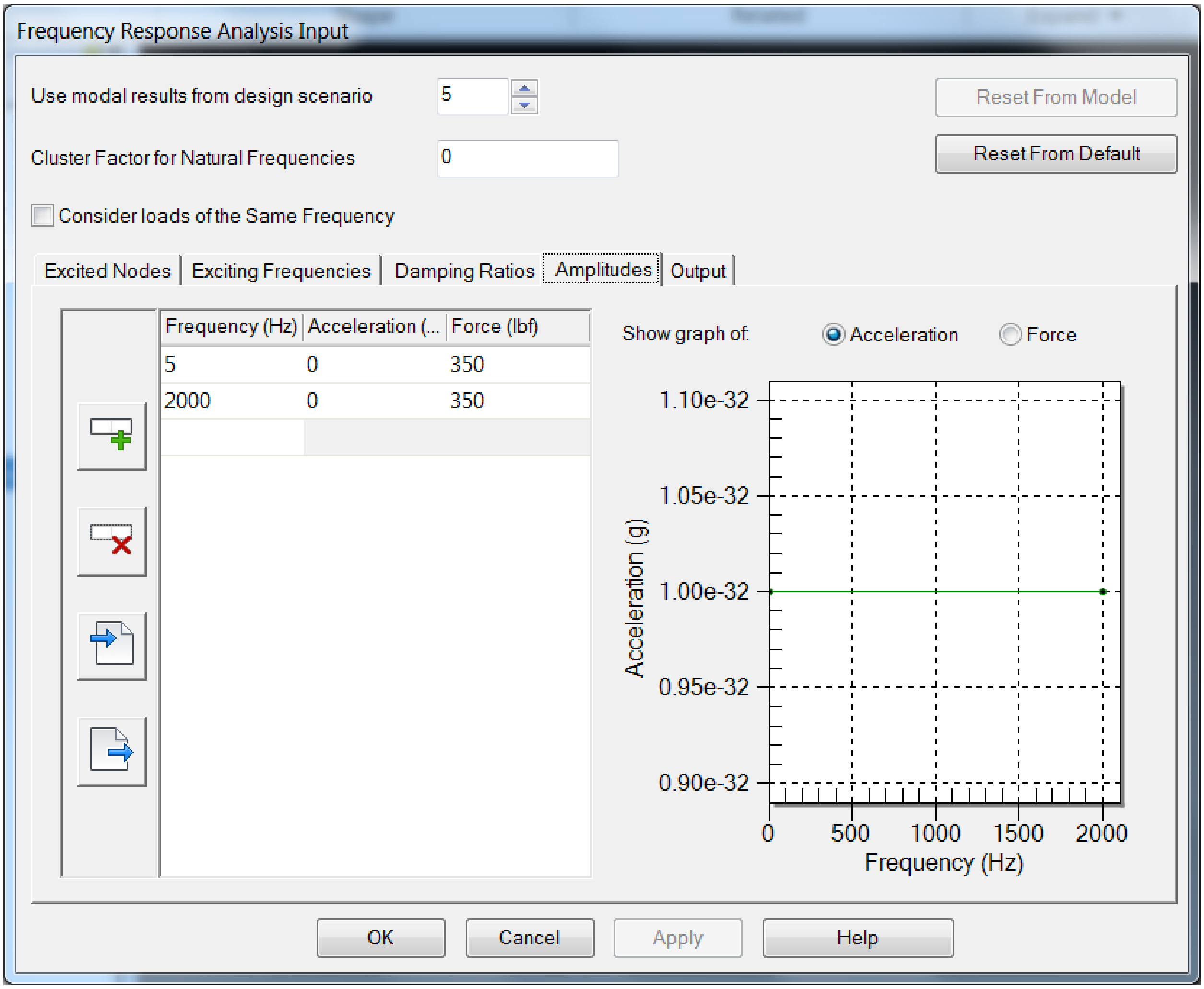

FIGURE 18: FREQUENCY RESPONSE INPUT LOADING MENU

Output

The in-phase and out-phase are output as well as the SRSS of the 2 are output for each exciting input frequency. An In-phase and/or Out-phase plot can be output for a range of input frequencies. The output shows which predominant mode or modes are being excited and therefore which to redesign.

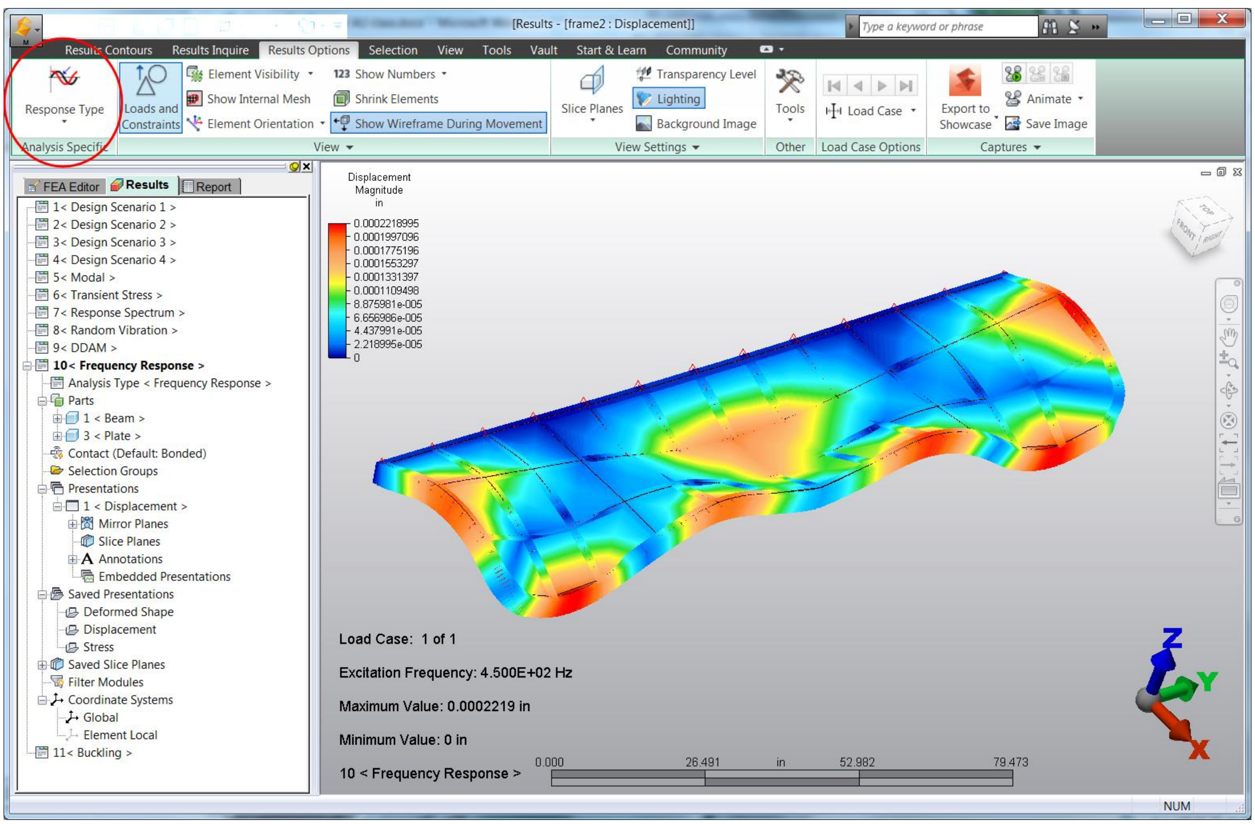

FIGURE 19: FREQUENCY RESPONSE RESULTANT OUTPUT

Nonlinear Dynamics

Contact can now be considered
Nonlinear dynamics can allow for contact including surface and impact plane contact. Larger motion can now be included. Snap-fit, drop tests, body to body impacts are applications for the Nonlinear MES analysis type. Additional simplifying of the model can be accomplished with kinematic elements which act as rigid bodies.

Input
The loading is similar to the linear transient stress analysis where forces or displacements can be input over time, and also includes Initial velocities. Geometric and material nonlinearities can now be included, which may exist in thin walled parts or local stress concentrations. Nonlinear materials such as stress strain curves for metals or rubber type can be defined.. Contact can be defined between parts including large sliding.

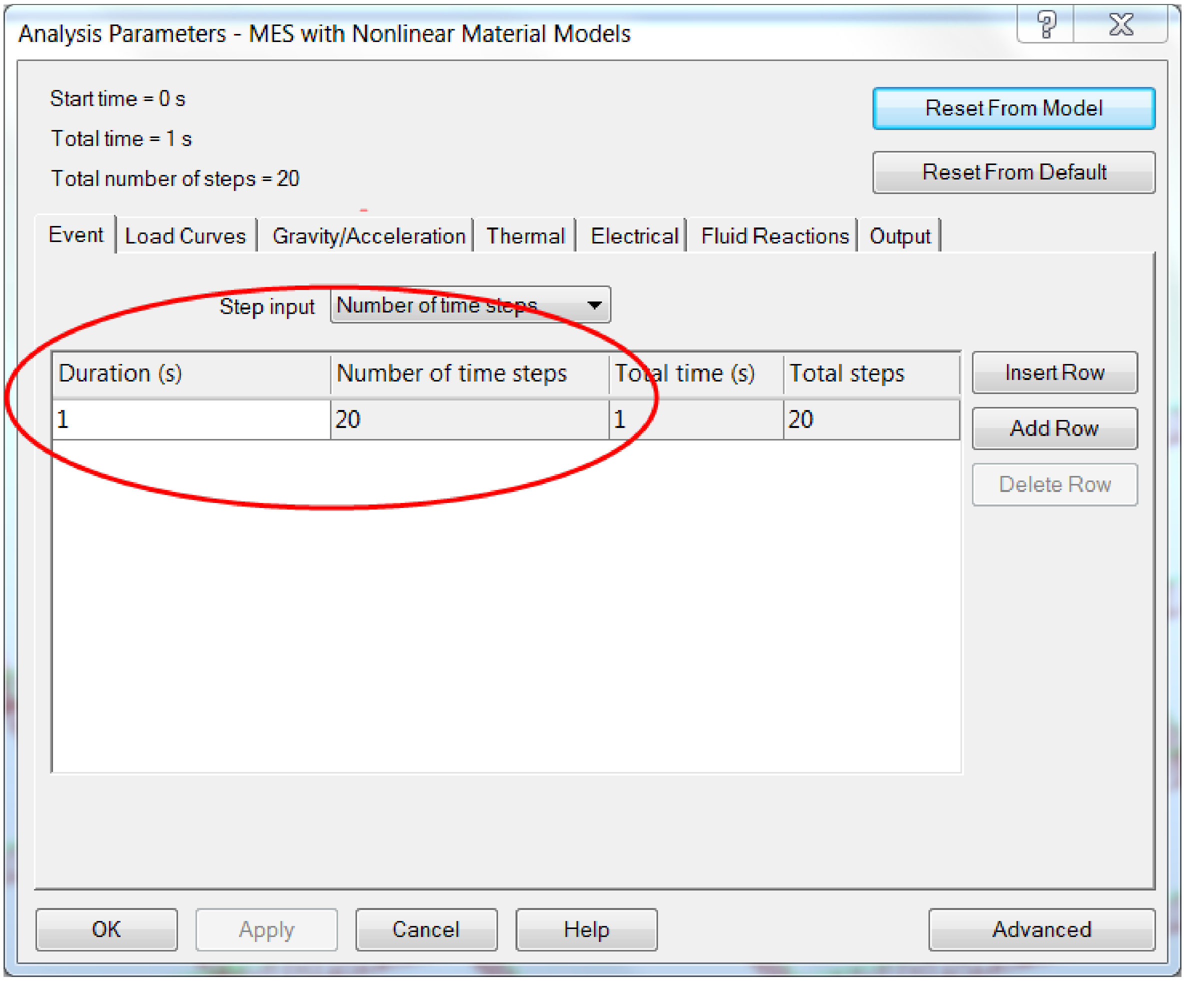

FIGURE 20: CONTROLLING THE EVENT

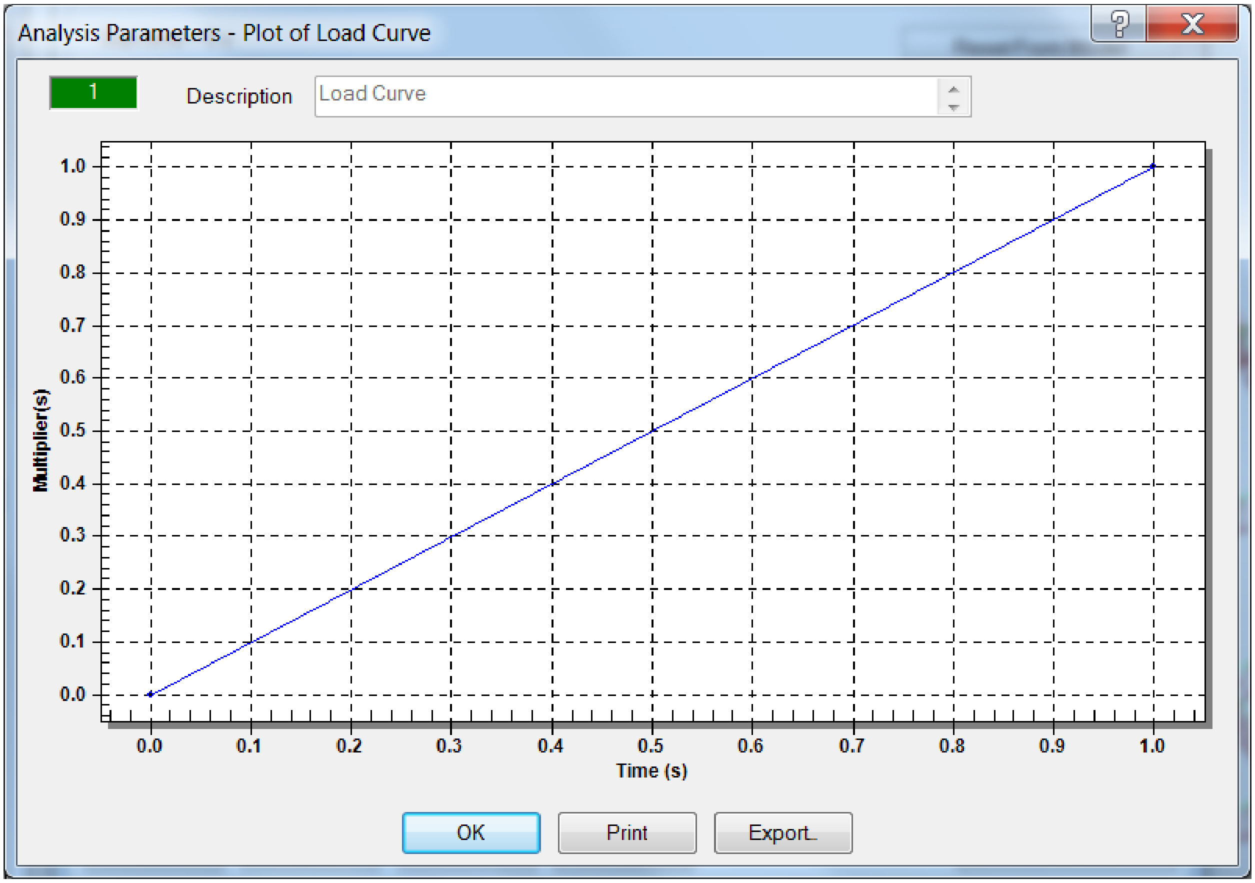

FIGURE 21: THE INPUT CURVE FOR LOADS

Output

Output is time dependent stress and displacement over time. Displacements are now by default true displacement in order to see large motion over time. Output during impact or other short time duration events can be output in great detail. Automatic convergence that is used in the solver allows for all time steps to be seen.

2