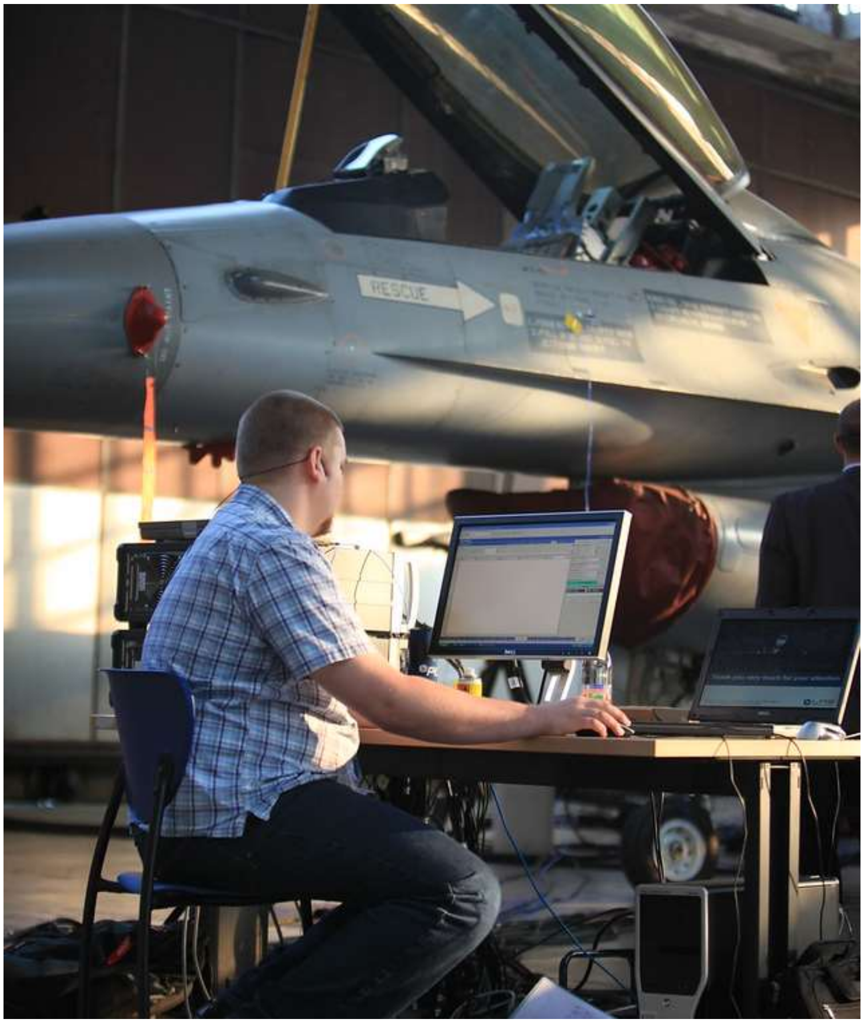
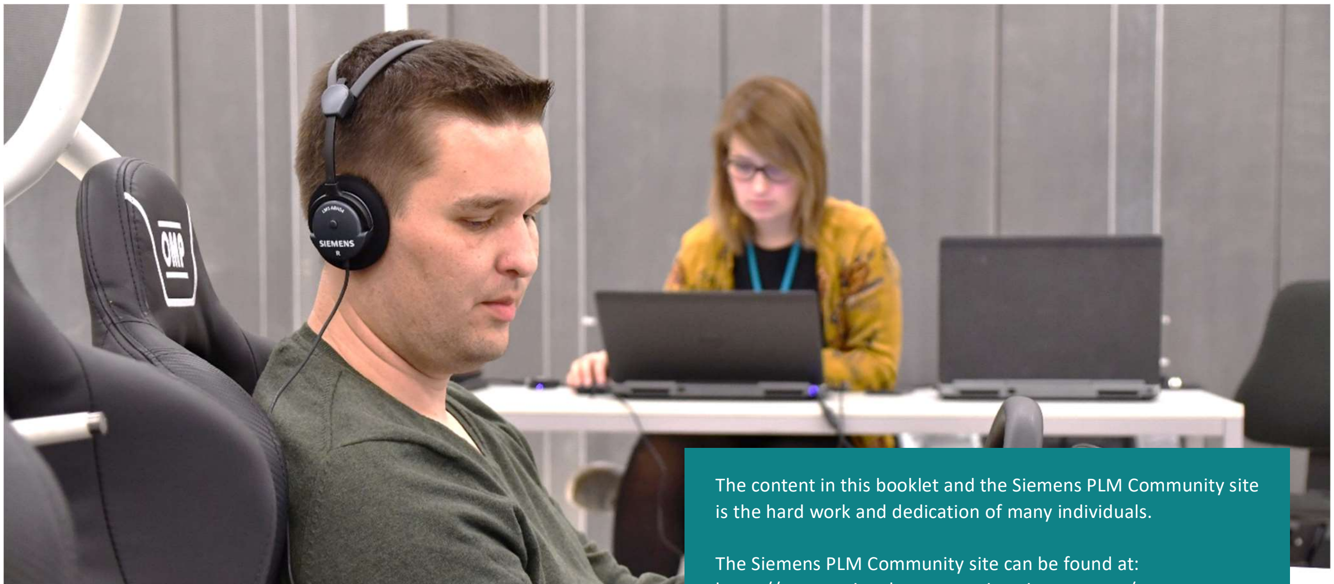


Modal Analysis Knowledge booklet

Testing Knowledge Base compilation

https://community.plm.automation.siemens.com/

Unrestricted



The content in this booklet and the Siemens PLM Community site is the hard work and dedication of many individuals.

The Siemens PLM Community site can be found at: https://community.plm.automation.siemens.com/

Table of Contents

Natural Frequency and Resonance

The definition of these items are:

- Natural Frequency: All physical structures have natural frequencies. These are the frequencies at which the structure will tend to vibrate when subjected to certain external forces. These frequencies are dependent on the way mass and stiffness are distributed within the structure.

- Resonance: Resonance is a phenomenon in which a dynamic force drives a structure to vibrate at its natural frequency. When a structure is in resonance, a small force can produce a large vibration response.

What does this mean in practice? When a dynamic force is applied to a physical object, it will vibrate. When a force is applied at the object's *natural frequency*, it goes into *resonance*, and a higher amplitude vibration response is created.

An analogy with a guitar may help. Pluck a string on a guitar and it will make the same sound each time. That is the guitar string vibrating at its natural frequency! The natural frequency is a property of the object itself: it will always vibrate at the same frequency independent of how hard or where it is plucked. A force had to be applied to cause the string to resonate and be heard.

All physical objects have multiple natural frequencies and can resonate under the right conditions. Sometimes the natural frequencies are excited by external forces acting on the object, which creates vibration. These vibrations may be so small that they cannot be seen by the human eye. Sometimes, they are quite large and easily observable as seen in *Figure 1*.

Figure 1: Everyday objects, like structures holding street lights, can resonate.

Resonance can cause discomfort (vibration in steering column caused by resonance) or be catastrophic (resonance in airplane wing leads to failure).

Single Degree of Freedom Example

A mass-spring-damper system is a simplified representation that is useful for understanding natural frequencies and resonant behavior in real world objects.

This is referred to as a Single Degree of Freedom (SDOF) system, because it has only one natural frequency/mode of vibration. A real-world object has many natural frequencies.

A diagram of a mass-spring-damper system is shown in *Figure 2*.

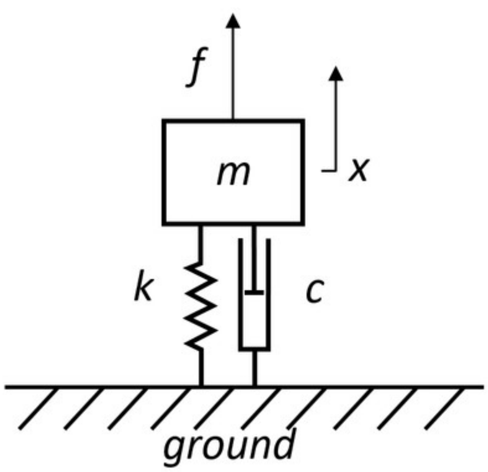


Figure 2: Mass-spring-damper system.

The system consists of:

- Mass (m)
- Stiffness (k)
- Damping (c)

The natural frequency (w_n) is defined by *Equation 1*.

$$\omega_n = \sqrt{k/m}$$

Equation 1: Natural frequency of mass-spring system

The natural frequency is an inherent property of the object. There are only two ways in which the natural frequency can be changed: either change the mass, or change the stiffness.

Amplitude Response

A force (f) can be applied to the object and the frequency response in displacement (x) or acceleration (a), can be plotted as shown in *Figure 3*.

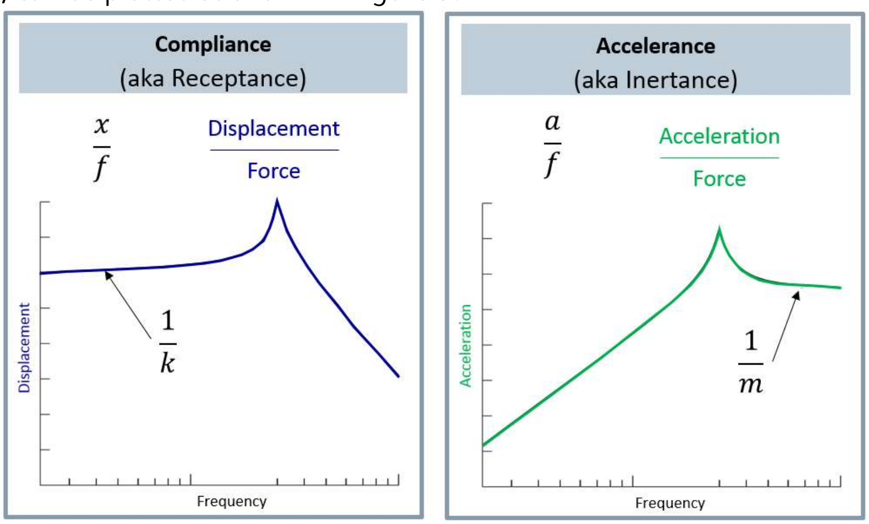


Figure 3: Left - Displacement response (compliance) graph of mass-spring-damper system due to force as function of frequency. Right – Acceleration response (accelerance) graph of same.

The largest displacement/acceleration of the mass occurs at the system's natural frequency. Other amplitude response observations include:

- Compliance plot - Below the resonant peak, the amplitude of the response is nearly constant, approximately 1/k. This comes from Hooke's law where force equals the product of stiffness and displacement (f=kx). Below the resonant frequency, the response of the system can be said to be stiffness dominated.

- Accelerance plot - Above the resonant peak, the amplitude is nearly a constant value of 1/m (really -1/m if phase is accounted for) as shown in *Figure 3*. This behavior is due to Newton's second law where force is the product of mass and acceleration (f=ma). Above the resonant frequency, the response of the system can be said to be dominated by the mass.

Knowing about these stiffness or mass regions can be useful in reducing vibration levels away from the resonance.

Phase Response

Applying the force through a moving base, and observing the mass response, yields some interesting phase relationships as shown in *Figure 4*.

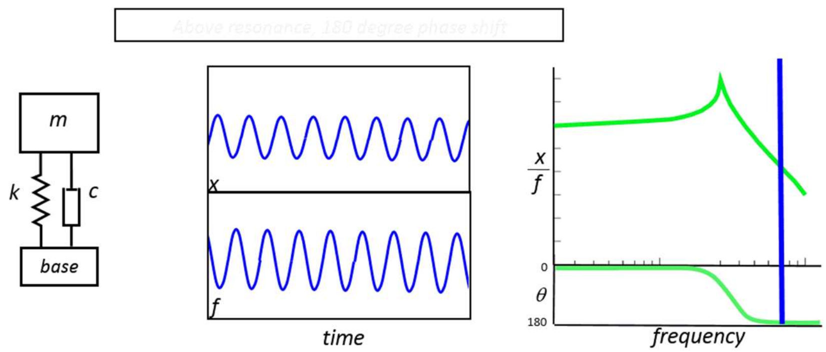


Figure 4: SDOF system response below, at, and above natural frequency of system.

The following can be observed:

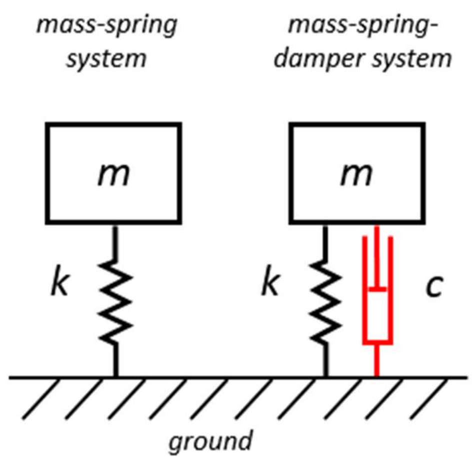
- Below the natural frequency, the base and mass move together in phase.
- At the natural frequency, the base and mass move 90 degrees apart, which creates a kind of "bucking" motion causing the high levels of vibration.
- Above the resonant frequency, the base and mass move out of phase.

Real world objects, from cars to airplanes to washing machines, can be thought of a collection of mass, stiffness, and damping elements. They have many natural frequencies. Finite element models, used in calculating natural frequencies virtually, use this approach. The models consist of a collection of elements composed of mass (mass density) and stiffness (Young's modulus).

Damping

Damping is the way a system naturally dissipates energy. Think back to the guitar example: does the guitar string oscillate forever after it is plucked? No! Energy is dissipated in the form of friction and sound which causes the string to return to rest after it has been plucked.

In the single degree of freedom example covered in the previous section, the mass-spring system (m and k) would stay in motion forever if there was no damper (c) present as shown in *Figure 5*.



The higher the damping, c, the sooner the response of the system decays to zero. The system response amplitude at the resonant frequency is reduced by increased damping. At the resonant frequency, the response of the system can be said to be damping dominated.

More information about damping, and how to calculate it, can be found in the Knowledge base article: How to determine damping from a FRF.

Mode Shapes

The SDOF example system had one natural frequency. Structures in the real world are more complex and have multiple degrees of freedom (MDOF). As a result, real world structures have many natural frequencies. The structure vibrates differently at each of these natural frequencies. How it moves at a particular frequency is called a mode shape.

Two modes of an aircraft (selected from many modes) are shown in *Figure 6*. Each mode shape is unique, with different parts of the aircraft participating in the mode.

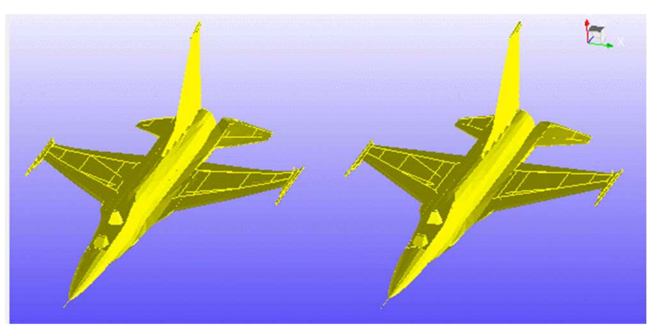


Figure 6: Left – Lower frequency global mode of aircraft, Right – Higher frequency local mode of aircraft tail.

Why is it important to understand modes and mode shapes? Mode shapes give valuable insight into how a structure behaves when operating at its natural frequencies. The shape can show the engineer where to constrain/modify a structure to reduce the vibration response, or how to shift the natural frequency so it does not coincide with the frequency of an excitation.

In *Figure 6*, for example, the tail wing mode (*right side*) would need to be modified to change the natural frequency. Changing the nose of the aircraft would have no effect on the natural frequency. The shape gives insight into how to tackle a dynamic issue.

At higher frequencies, generally speaking, modes become local in natural, rather than global. In a global mode, the entire structure participates (*mode shape on left in Figure 6*), while in a local mode, only part of the structure participates (*mode shape on right in Figure 6*).

It is also typical that mode shapes become more complex at higher natural frequencies as seen in *Figure 7*.

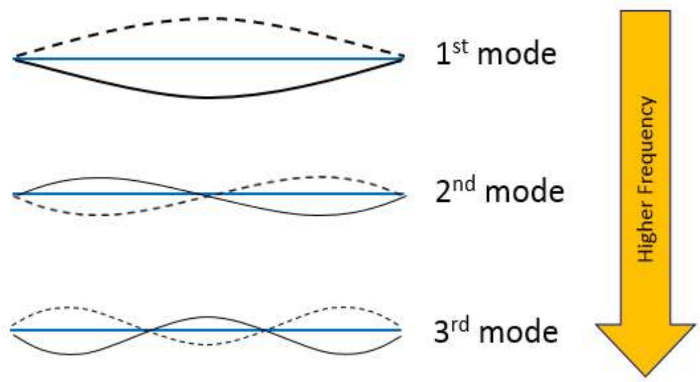


Figure 7: Mode shape of a simply-supported beam becomes more complex at higher frequencies.

The fact that modes become more complex and localized at higher frequencies has implications for structural dynamic simulations and tests. Simulations require a finer mesh and more elements, increasing solution times. Tests will require more locations to be measured on the structure.

Mitigating the Effects of Resonance

Knowing how destructive mechanical resonance can be, what can be done to avoid it? Options include:

- Mass/Stiffness Modifications
- Damping Changes
- Tuned Absorbers

Mass and Stiffness Fixes

To avoid resonance, the forcing frequency applied to the structure should not be at or near a natural frequency. If the forcing frequency cannot be changed, then the natural frequency of the structure needs to be modified. This can only be done by altering the mass or stiffness (see *Equation 1*).

The guitar is a good example of how changing the mass or stiffness of a system effects the natural frequency. The strings on a guitar have different thicknesses. The strings which produce lower notes are thicker (more mass) than those which produce higher notes (less mass). As the mass of a guitar string increases, the natural frequency decreases (*Figure 8*).

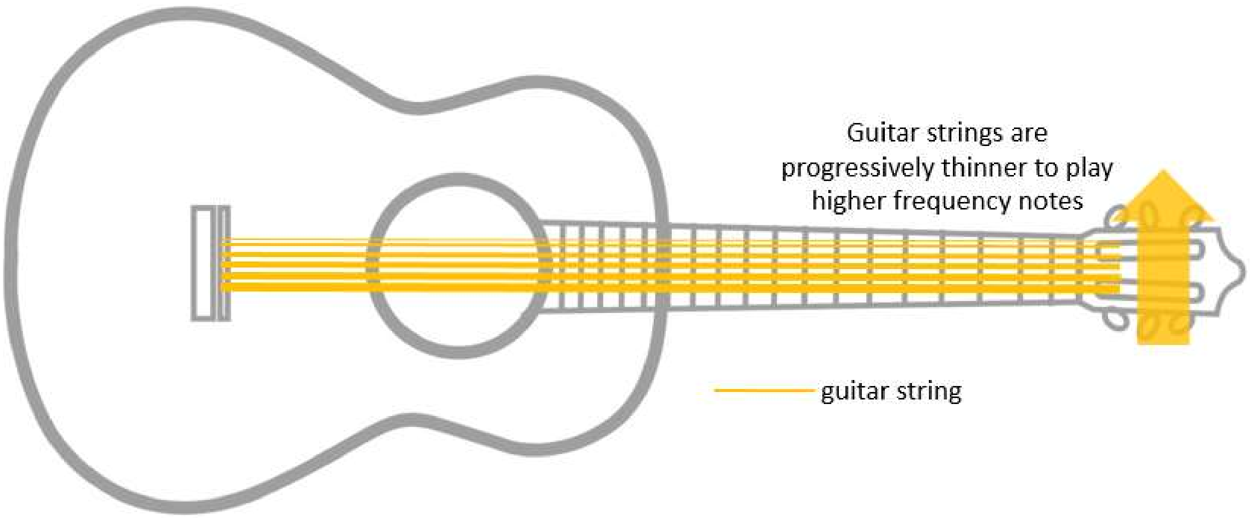


Figure 8: Progressively thinner strings create higher frequency notes on a guitar.

When tuning a guitar, pegs/knobs on the guitar are turned to tighten or loosen the strings. Tightening a string increases the stiffness, raising the natural frequency.

In a structure, increasing the stiffness to place the natural frequency above the forcing frequency helps reduce vibration.

Damping fixes

Damping can be added to reduce the severity of vibration when operating at or near a natural frequency. The plot below (*Figure 9*) shows the reduction in amplitude of the system response as damping increases.

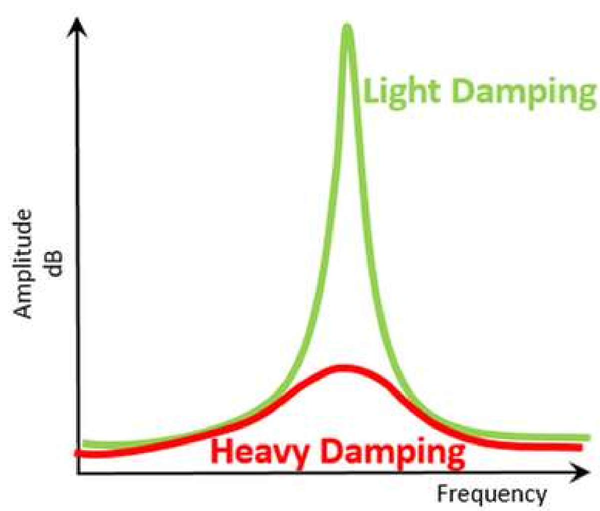


Figure 9: When damping is low (green), system amplitude response is high. When damping is high (red), system amplitude response is low.

Damping treatments are often used to reduce vibration. For example, many large bridges, such as London's Millennium Bridge, feature fluid viscous dampers to control vibration (*Figure 10*).

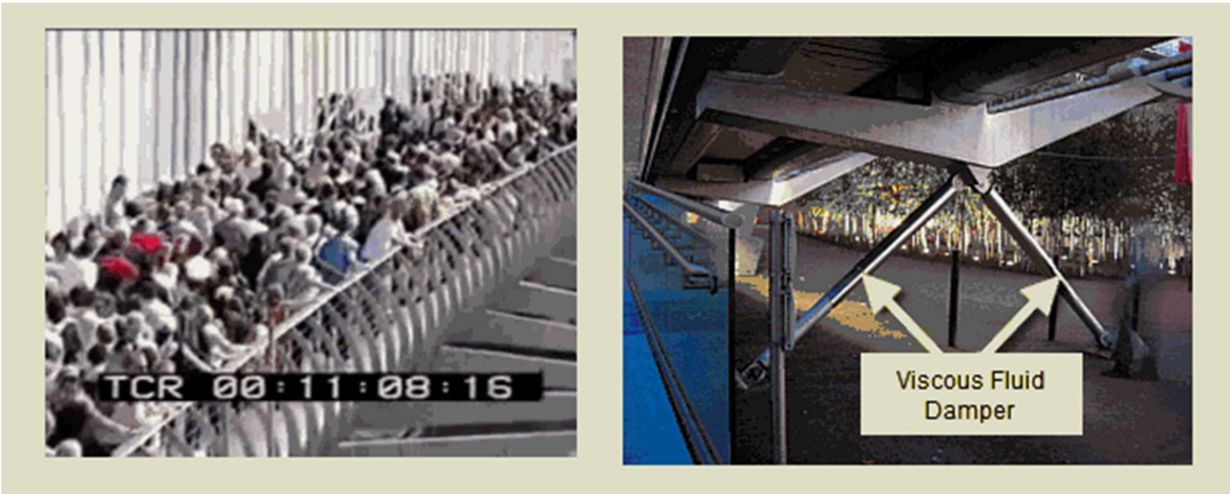


Figure 10: Fluid viscous dampers (right) are used to reduce unwanted vibration (left) on the Millennium Bridge.

The Millennium Bridge was opened to the public on June 10, 2000. Due to excessive vibration caused by pedestrian traffic, it was shut down after two days, retrofit with 37 fluid viscous dampers, and re-opened on February 22, 2002.

Tuned Absorber

A tuned mass-spring-damper system can be used to reduce the amplitude of vibration in a dynamic system. A tuned mass damper modification is created by adding an additional mass-spring system "tuned" to the natural frequency of an existing system (*Figure 11*).

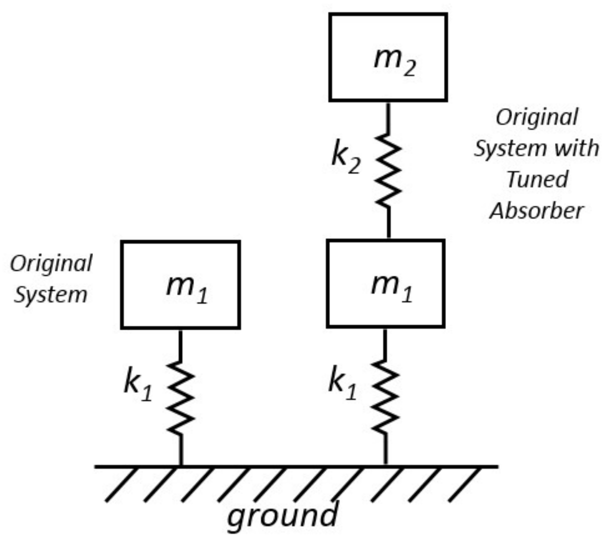


Figure 11: Left – Original system consisting of m1 and k1. Right – Original system with tuned absorber (m2, k2) applied.

Applying a tuned absorber (m2, k2) to an existing system (m1, k1) has two effects as shown in *Figure 12*:

- At the original natural frequency of the original system, the additional tuned mass-spring-damper will vibrate, but the original system will not move.
- The original natural frequency is split into two. One mode of vibration where the original and tuned system are in-phase, and one mode where the original and tuned systems are out of phase. The in-phase mode is at a lower frequency than the original system natural frequency, while the out-of-phase mode is a higher frequency that the original system natural frequency.

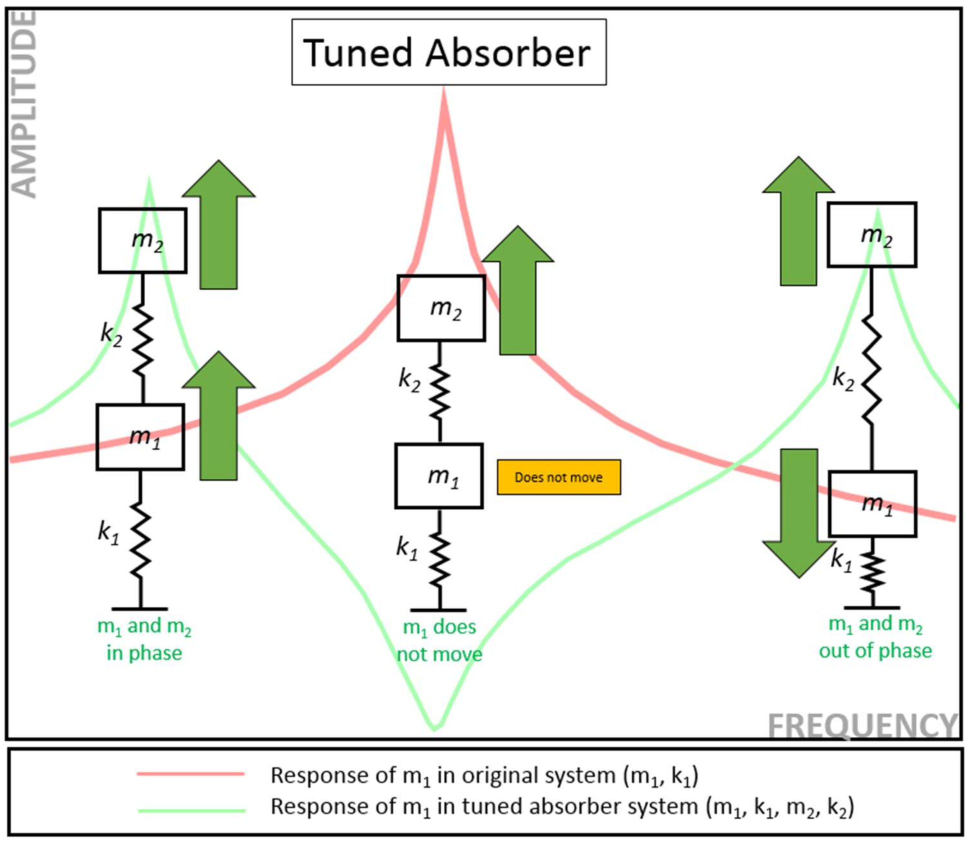


Figure 12: Original system (m1, k1) response (red) is reduced to zero (green) by introducing tuned absorber (m2, k2). New system has two modes: one above original system natural frequency, and one below.

The effects are only possible when the tuned absorber frequency is equal to the original system frequency.

How does this reduce vibration? Consider a vehicle body mode where the engine idle forces excite the bending resonance. The bumper can be turned into tuned mass-spring system so it can vibrate and cancel the bending mode. The bumper will vibrate at idle, but the body will not, reducing the vibration felt by the driver. The newly created in-phase mode will not be excited, since it is below the idle vibration. The out-of-phase mode, which is higher than the original frequency excited at idle, can be tuned to an engine speed that is not often used in operation.

One of the benefits of a tuned absorber approach is the additional mass and stiffness changes to the structure can be minimal. In the vehicle bumper example, the mass of the bumper was already part of the structure. Adding spring/stiffness elements to the bumper was a minimal change in the overall weight of the vehicle, which helps with fuel efficiency, etc.

Tuned dynamic absorbers also are used to help reduce the swaying vibration in buildings. The Taipei 101 skyscraper contains the world's largest and heaviest tuned mass dampers, at 660 metric tons (730 short tons). The damper is tuned to the swaying mode of the building (*Figure 13*).

Figure 13: Taipei 101 skyscraper (left) and tuned absorber (right).

The tuned absorber is viewable by the public on an indoor observation deck at the top of the skyscraper. It cost an estimated $4 million to build.

More about modifying stiffness/mass, adding damping, applying tuned absorbers to modal models in the Modification Prediction knowledge base article.

Examples of Resonance

Broughton Bridge

In 1831, the Broughton suspension bridge in England (*Figure 14*) collapsed when a column of soldiers crossed the bridge marching in step.

Figure 14: Broughton Bridge in 1883.

The bridge had a natural frequency near the frequency of the marching of the soldiers and began to vibrate violently. This caused a bolt in one of the supporting chains to fail which led to the collapse of the bridge.

Flutter in Aircraft

The flight envelope (Mach speed and altitude) of any aircraft is determined by the resonant frequencies of the wings and airframe structure. If the aircraft goes too fast, the aerodynamic forces can excite the natural frequencies causing a resonance, which in an aircraft is called flutter.

Beating Sound

Rolled the rear windows down while driving and heard a beating sound? That beating sound (sometimes also called buffeting) is amplified by a resonance of the vehicle air cavity (*Figure 16*). Even air can have a resonance!

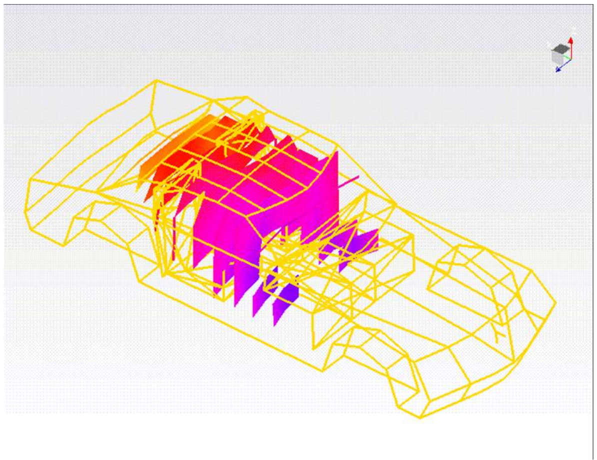


Figure 16: Experimentally measured acoustic mode of vehicle air cavity.

The air cavity inside the vehicle is trapped in a specific volume/geometry. As a result, the air cavity has natural frequencies which can be excited and cause increases in the sound level in the car.

Opera Singer and Glass

A popular example of resonance from movies has an opera singer hitting a specific note, causing glasses in the audience to shatter (*Figure 17*), due to resonance.

Figure 17: Glass shattering due to sound waves exciting resonance.

The TV show "Mythbusters" confirmed that this is possible on the episode "Breaking Glass" which was originally broadcast on May 18, 2005.

Brake Squeal

Ever pressed on the brake in a car and heard a high frequency squeal? This is resonance at work. A brake rotor mode, probably around 3000 Hz or so, was excited by braking (*Figure 18*).

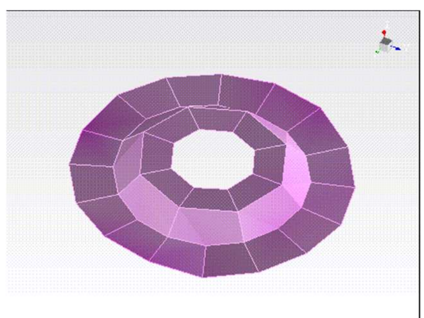


Figure 18: Brake rotor has a resonance which can create audible squeal during braking.

Brake rotors are symmetric structures, which can create multiple modes at the same frequency. To reduce squeal, the rotor design should be modified to keep the modes well separated so they cannot cross excite each other.

Helmholtz Resonator Air Induction

The German scientist, Hermann von Helmholtz (a friend of Ernest Werner Siemens), created the Helmholtz resonator in the 1850s. A Helmholtz resonator is an air-based tuned absorber that is designed for a specific frequency. When blowing over the lip of an empty bottle, the resonant frequency of the trapped volume of air can be heard.

The larger trapped volume of air acts like the spring (it gets compressed into the volume). The smaller volume at the mouth, which is open to the air, acts like a mass. See the diagram in *Figure 19*.

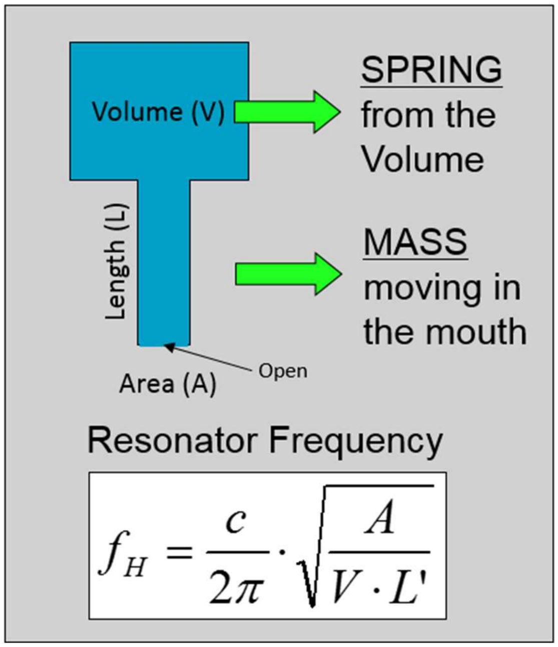


Figure 19: A Helmholtz resonator creates a tuned absorber (frequency fh) with a trapped volume of air (V) which acts like a spring, and an open volume of air (defined by A and L).

In ducted systems, the Helmholtz resonator can be used as tuned absorber to reduce sound levels at a specific frequency. In *Figure 20*, the air in the exhaust system has a 330 Hz resonant frequency. The sound pressure level at the outlet is greatly reduced by introducing a Helmholtz resonator attached to the duct. The resonator "absorbs" energy from the exhaust system and reduces sound at the outlet.

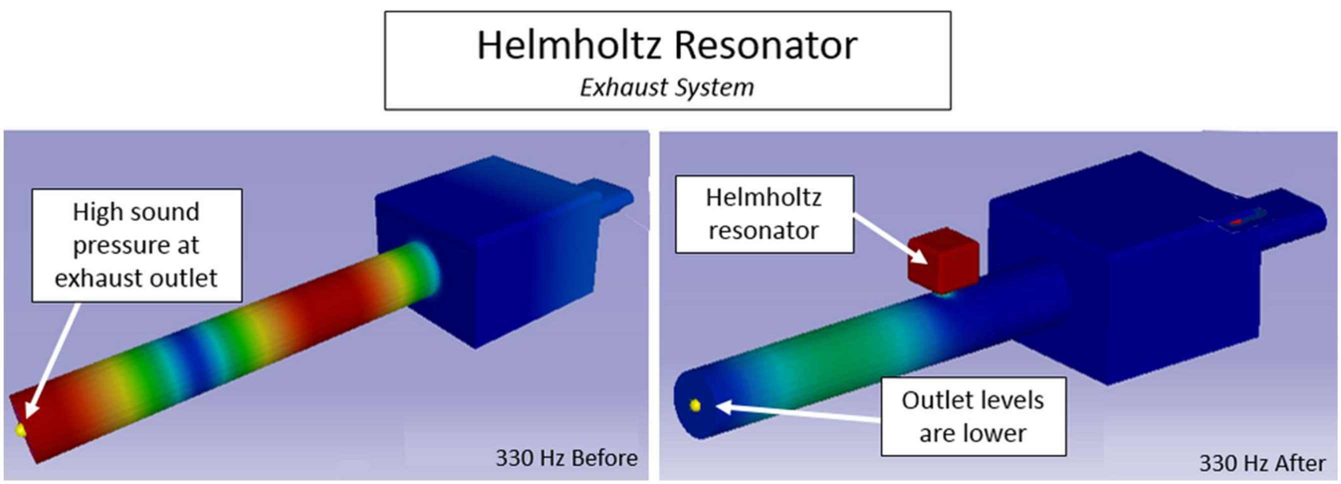


Figure 20: Left - A 330 Hz air cavity mode creates high pressure at outlet. Right - With a Helmholtz resonator installed, the outlet pressure is greatly reduced.

These resonators can be commonly found in air induction systems, HVAC ducts, and exhaust systems.

Final Thoughts: Resonance - Not always a bad thing!

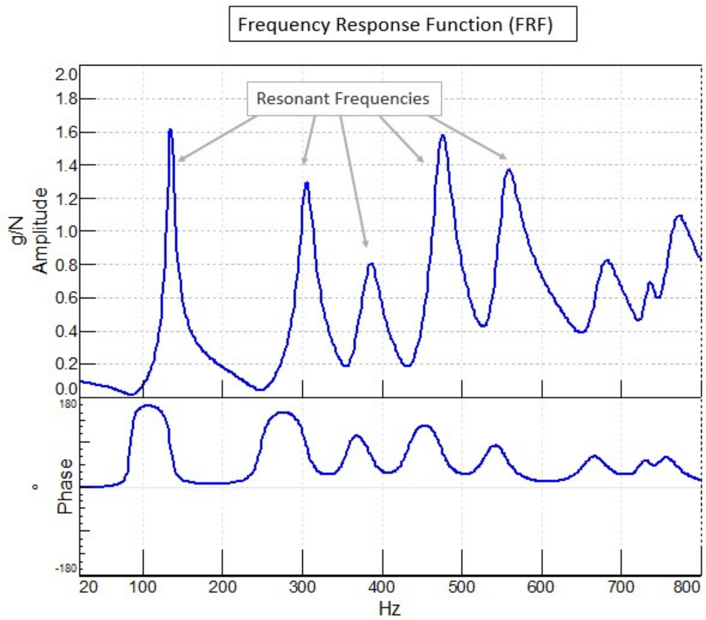
Resonance is not always a bad thing! Sometimes amplifying the response of a system can be useful to increase its performance. Some examples include:

- The human ear can hear sound better between 1000 and 4000 Hertz due to an acoustic resonance of the air in the ear canal.
- Tuning into a radio station (at least in the analog days) is done through resonance. The electrical circuit is tuned to the radio frequency by adjusting the frequency of a capacitor.
- Vibrating bowls used in manufacturing applications to sort parts rely on the resonance of the bowl to create large vibration with minimal energy input.
- Microwave ovens use the natural frequency of water molecules to heat items quicker than a traditional oven.
- In the medical field, magnetic resonance imaging (MRI) is used to view inside the human body without requiring invasive procedures. A MRI is tuned to resonate with certain tissues.

Frequency Response Function (FRF)

A Frequency Response Function (or FRF), in experimental modal analysis:

- Is a frequency based measurement function
- Used to identify the resonant frequencies, damping and mode shapes of a physical structure
- Sometimes referred to a “transfer function” between the input and output
- Expresses the frequency domain relationship between an input (x) and output (y) of a linear, time-invariant system



Bode Plot of Amplitude and Phase of a FRF function. Amplitude has peaks corresponding to natural frequencies/resonances of test object. Phase has shift at resonant frequency.

In a Frequency Response Function measurement the following can be observed:

- *Resonances* - Peaks indicate the presence of the natural frequencies of the structure under test
- *Damping* - Damping is proportional to the width of the peaks. The wider the peak, the heavier the damping
- *Mode Shape* – The amplitude and phase of multiple FRFs acquired to a common reference on a structure are used to determine the mode shape

In Experimental Modal Analysis

Many types of input excitations and response outputs can be used to calculate an experimental FRF. Some examples:

- Mechanical Systems: Inputs in force (Newtons) and outputs in Acceleration (g's), Velocity (m/s) or Displacement (meter)
- Acoustical Systems: Inputs in Q (Volume Acceleration) and outputs in Sound Pressure (Pascals)
- Combined Acoustic and Mechanical systems: Inputs in force (either Q or Newtons) and outputs in Sound Pressure (Pa), Acceleration (g's), etc.
- Rotational Mechanical Systems: Inputs in Torque (Nm) and output in Rotational Displacement (degrees)

For an experimental modal analysis on a mechanical structure, typically the input is force and output is acceleration, velocity or displacement.

Forces can be applied and measured via:

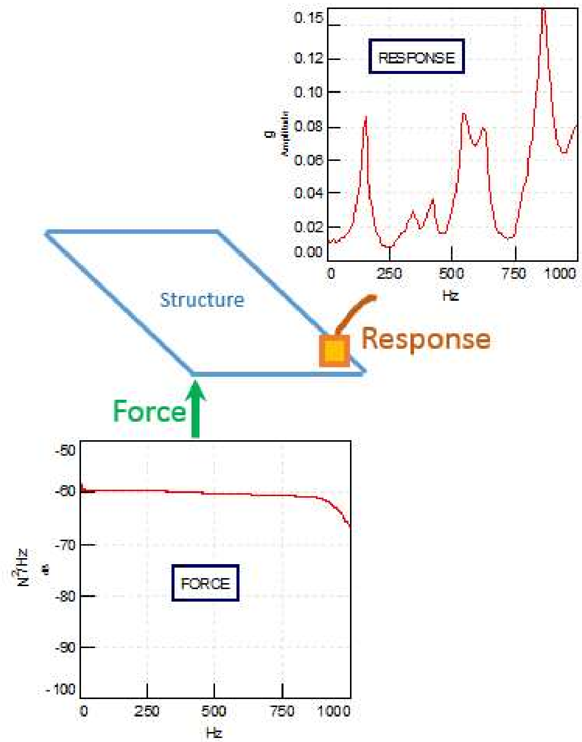
- Impact Hammers
- Electrodynamic Shakers

Responses can be measured by:

- Accelerometers: measure acceleration vibration
- Lasers: measure surface velocity
- String Pots, Photogrammetry: displacement

Generally, the input force spectrum (X) should be flat versus frequency, exciting all frequencies uniformly. When viewing the response (Y), the peaks in the response indicate the natural/resonant frequencies of the structure under test.

Because the FRF response is "normalized" to the input, the peaks in the resulting FRF function are resonant frequencies of the test object.



Imaginary FRFs and Mode Shapes

A FRF is a complex function which contains both an amplitude (the ratio of the input force to the response, for example: g/N) and phase (expressed in degrees, which indicates whether the response moves in and out of phase with the input).

Any function that has amplitude and phase can also be transformed to real and imaginary terms, as described the equation below:

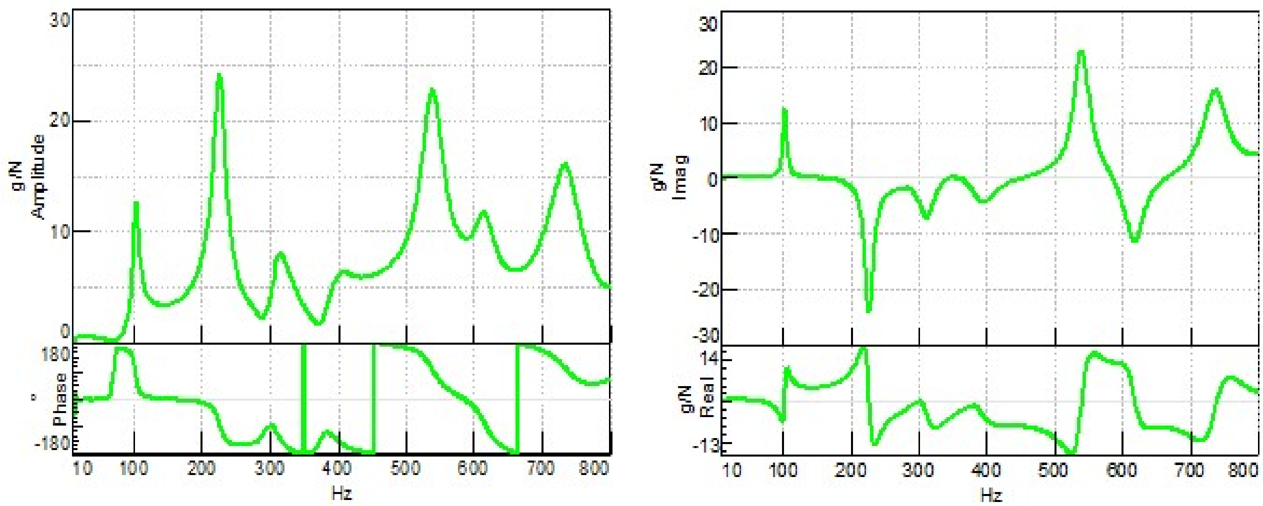
$$Amplitude = \sqrt{Imag^2 + Real^2}$$

$$Phase = \tan^{-1}(\frac{Imag}{Real})$$

After transforming the FRF from Amplitude & Phase to Real & Imaginary, some interesting things happen:

- The real part of the FRF will equal zero at natural/resonant frequencies

- The imaginary will have "peaks" either above or below zero which indicate resonant frequencies. The direction of the peaks can be used to determine the mode shape associated with the natural/resonant frequency

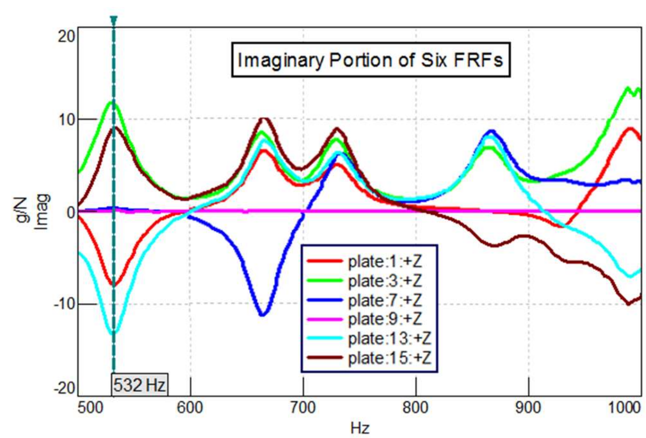


Left: FRF expressed in Amplitude and Phase, Right: FRF expressed in Real and Imaginary

If several FRFs are acquired at different locations on the structure, and they are all phased with respect to a common reference, the imaginary part of the FRFs can be used to plot the mode shape.

In the example below, six FRF measurements were taken on a simple metal plate hung in free-free boundary conditions. The six FRFs are located as follows:

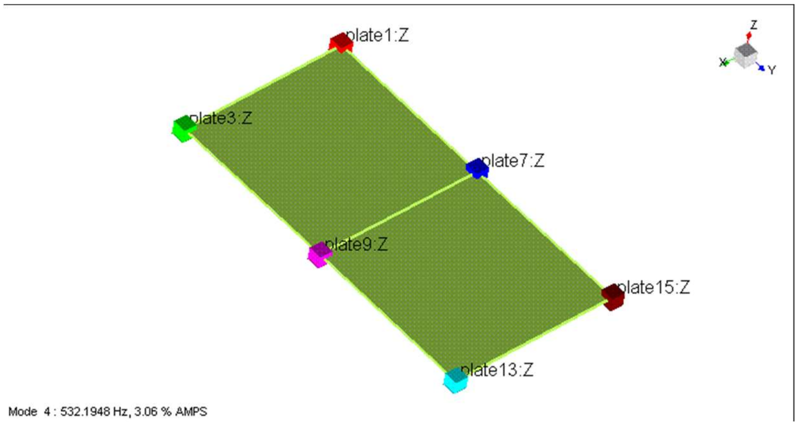
- Measurement points 1 and 3 are on one end of plate
- Measurement points 7 and 9 are in the center
- Measurement points 13 and 15 are on the other end, opposite points 1 and 3



Plot of the imaginary portion of six FRFs on simple plate

When plotting the imaginary portion of the FRF, and looking at 532 Hz:

- Measurement points 1 and 13 move in phase, and are opposite corners of the plate (red and cyan)
- Measurement points 7 and 9 have low amplitude (blue and magenta)
- Measurement points 3 and 15 are in phase on opposite corners of the plate (brown and green)

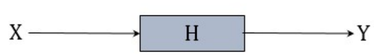


When plotting the imaginary values at 532 Hz on a stick figure model of the plate, it can be seen that the plate is in torsion.

Who knew that viewing "imaginary" FRFs could be so useful?!

Digital Signal Processing Terminology

In nomenclature, a FRF is typically represented by the single capital letter H. The input is X and output is Y. H, X and Y are all functions versus frequency.



The FRF is the crosspower (Sxy) of the input (x) and output (y) divided by the autopower (Sxx) of input.

$$H = \frac{S_{xy}}{S_{xx}}$$

The autopower Sxx is the complex conjugate of the input spectrum to itself, which becomes an all real function, containing no phase. The crosspower Sxy is the complex conjugate of the output spectrum and the input spectrum and contains both amplitude and phase.

Averaging FRF Measurements: Coherence & Estimators

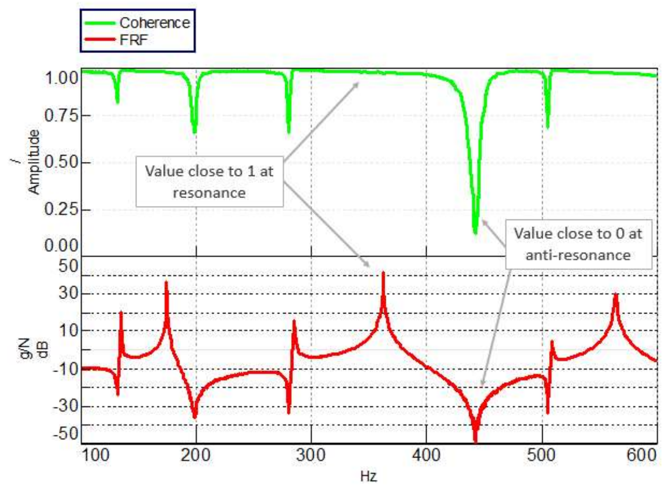
It is common practice to measure the FRF measurement several times to ensure that a reliable estimate of the structures transfer function is being measured. The repeatability of the individual FRFs is checked by estimating a coherence function, while the average is calculated using different estimator methods, depending on the desired end result.

Coherence

Coherence is function versus frequency that indicates how much of the output is due to the input in the FRF. It can be indicator of the quality of the FRF. It evaluates the consistency of the FRF from measurement to repeat of the same measurement.

The value of a coherence function ranges between 0 and 1:

- A value of 1 at a particular frequency indicates that the FRF amplitude and phase are very repeatable from measurement to measurement.
- A value of 0 indicates that opposite – the measurements are not repeatable, which is a possible "warning flag" that there is an error in the measurement setup.



When the amplitude of a FRF is very high, for example at a resonant frequency, the coherence will have a value close to 1.

When the amplitude of the FRF is very low, for example at an anti-resonance, the coherence will have a value closer to 0. This is because the signals are so low, their repeatability is made inconsistent by the noise floor of the instrumentation. This is acceptable/normal. When the coherence is closer to 0 than 1 at a resonant frequency, or across the entire frequency range, this indicates a problem with the measurement.
Problems could include:

- Instrumentation error – For example, ICP power is not being supplied to transducer that requires ICP power
- Inconsistent excitation – Structure is not being hit by an impact hammer consistent (for example, operator is tired and striking structure at different angles between impacts)
- Insufficient force – The structure is not being excited. For example, a very small hammer (example: size of pencil) on a large object (example: size of a bridge) with a large distance between excitation and response measurement

Note that if only one measurement is performed, the coherence will be a value of 1! The value will be one across the entire frequency range – giving the appearance of a "perfect" measurement. This is because at least two FRF measurements need to be take and compared to start to calculate a meaningful coherence function. Don't be fooled!

Estimators

When measuring a Frequency Response Function on a structure by inputting a 30 Hz forcing frequency. Using three different force levels, the following happens:

- Measurement #1 – Two Newtons of input force results in 10 g's of acceleration response: Ratio of response to input is 5.0 g/N

- Measurement #2 – One Newton of input force results in 5.1 g's of acceleration response: Ratio of response to input is 5.1 g/N
- Measurement #3 – Three Newtons of input force at 30 Hz result in 14.7 g's: Ratio of response to input is 4.9 g/N

Why the variation between measurements? Unlike generating a FRF from a Finite Element Model, measuring a FRF may not return the same value every time a measurement is taken: structures are not completely linear and there can be small amounts of instrumentation noise in the measurement.

The three measurements had similar results: 5.0, 5.1 and 4.9 g/N. Which one is correct?

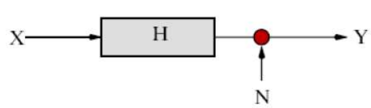
To determine the "correct" value, estimators are used for calculating the amplitude ratio (H) of the input to output of FRFs. There are three main FRF estimators in use today: H1, H2 and HV estimators. When trying to characterize a structure the following table of data was gathered over 5 individual FRF measurements at three different frequencies. The following is a simplified example for learning purposes. In a single FRF measurement, when looking at 3 different frequencies, the following may be observed over 5 individual measurements:

	133 Hz (resonance)		194 Hz (mid)		245 Hz (anti-resonance)	
	X	Y	X	Y	X	Y
Measurement 1	2.0	20.1	1.6	1.5	2.1	0.50
Measurement 2	1.0	10.0	1.0	1.1	1.1	0.27
Measurement 3	0.5	4.90	3.0	2.9	0.4	0.10
Measurement 4	3.0	30.0	0.5	0.5	3.2	0.80
Measurement 5	4.0	40.2	4.1	4.2	4.0	1.10

These X and Y pairs are plotted, a line is fit to the data. The slope of the line (typically g/N) will determine the amplitude of the FRF. The estimators affect how the data is fit and how much each data point is adjusted to create the best fit line.

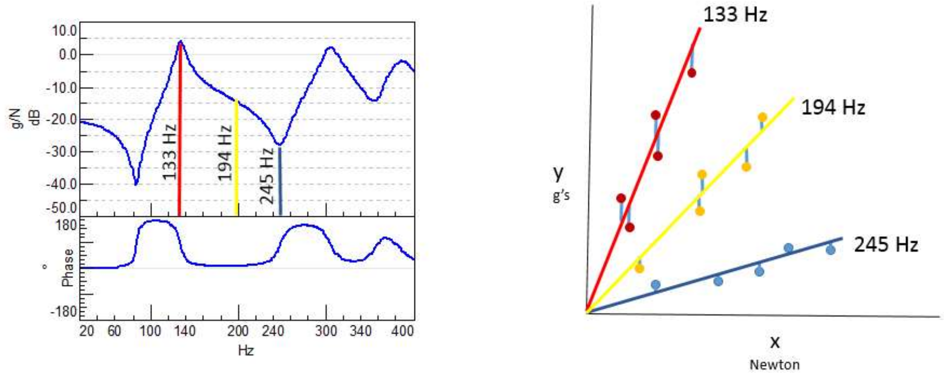
H1 Estimator

The most commonly used estimator is the H1-estimator, which assumes that there is no noise on the input and consequently that all the X measurements (the input) are accurate. All noise (N) is assumed to be on the output Y.



$$Y = HX + N$$

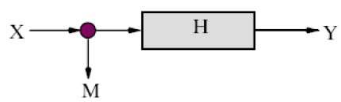
This estimator tends to give an underestimate of the FRF if there is noise on the input. H1 estimates the anti-resonances better than the resonances. Best results are obtained with this estimator when the inputs are uncorrelated.



Left: FRF function, Right: Graph of X and Y values from 5 separate measurements, and Y-only corrections for average

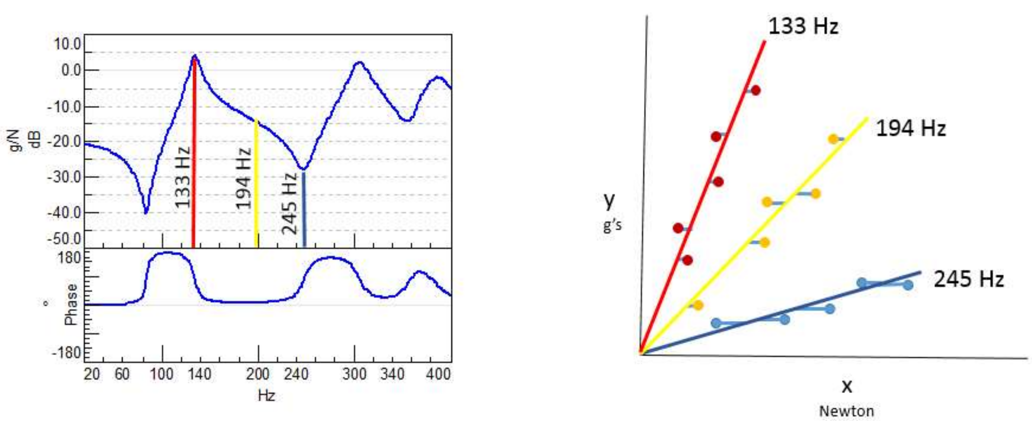
H2 Estimator

Alternatively, the H2 estimator can be used. This assumes that there is no noise on the output and consequently that all the Y measurements are accurate. Noise (M) is assumed to be only on input X.



$$Y = H(X - M)$$

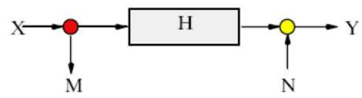
This estimator tends to give an overestimate of the FRF if there is noise on the output. This estimator estimates the resonances better than the anti-resonances. Notice the corrections are bigger for the 245 Hz anti-resonance frequency than for the 133 Hz resonance frequency.



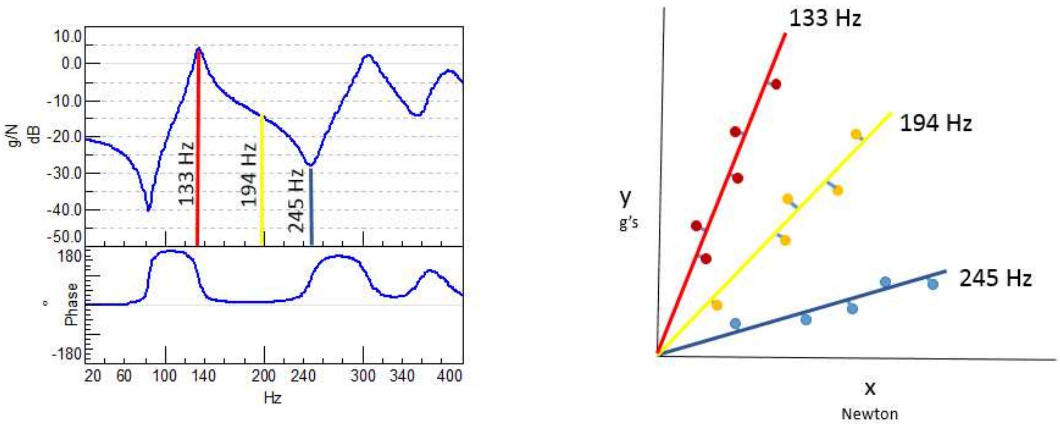
Left: FRF function, Right: Graph of X and Y values from 5 separate measurements, and X-only corrections for average

Hv Estimator

The Hv estimator provides the best overall estimate of the frequency function. It approximates to the H2 estimator at the resonances and the H1 estimator at the anti-resonances. It does however require more computational time than the other two, which is not an issue for today's computers. The Hv estimator assumes noise (M and N) is on both the X input and Y output.



Y−N =H (X−M)



Left: FRF function, Right: Graph of X and Y values from 5 separate measurements, and corrections.

Conclusion

Frequency Response Functions (FRFs) are used to measure and characterize the dynamic behavior of a structure.

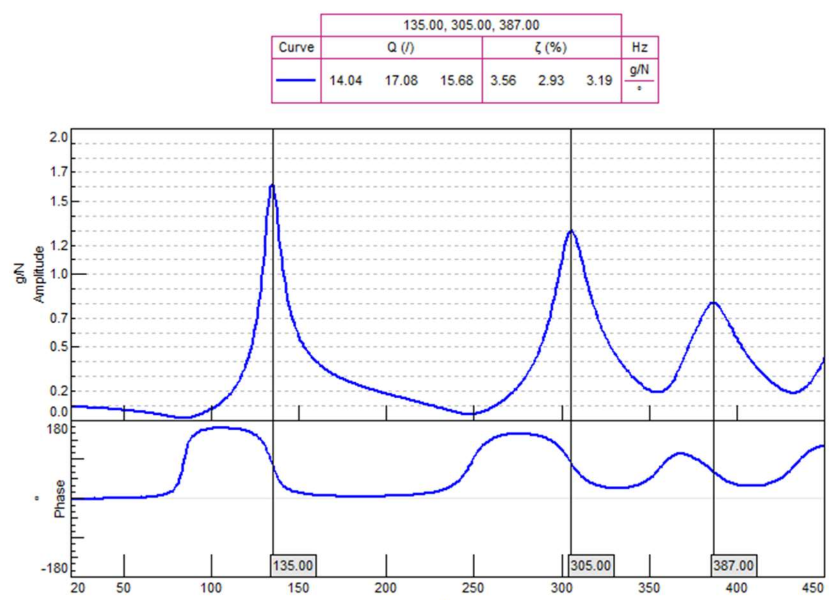
FRFs contain information about:

- Resonant frequencies
- Damping
- Mode shape

When creating an average FRF, coherence functions can give indications of FRF quality, while estimation methods are used to account for noise on the measurements.

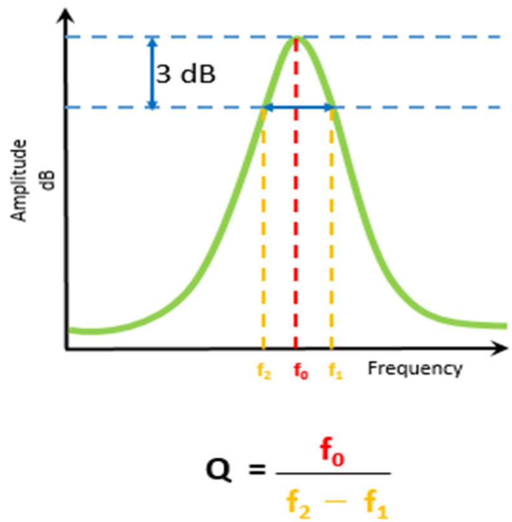
How to Calculate Damping from a FRF?

A classical method of determining the damping at a resonance in a Frequency Response Function (FRF) is to use the "3 dB method" (also called "half power method").



Bode plot of FRF Amplitude (Top) and Phase (Bottom). The damping values "Q" and ""Damping Ratio" are shown for three different peaks in the FRF.

In a FRF, the damping is proportional to the width of the resonant peak about the peak's center frequency. By looking at the 3 dB down from the peak level, one can determine the associated damping.



The "quality factor" (also known as "damping factor") or "Q" is found by the equation Q = $f_0/(f_2-f_1)$, where:

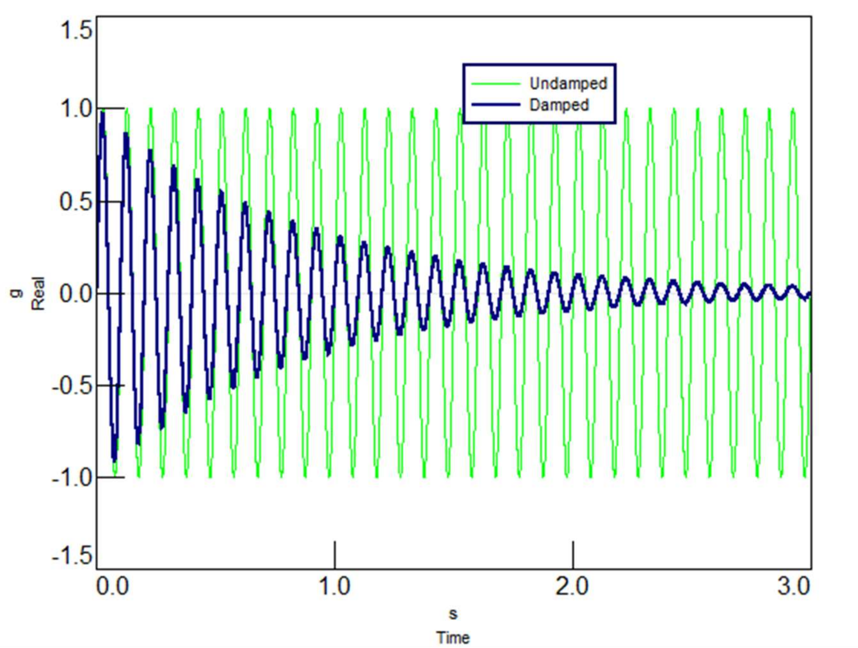
f_0 = frequency of resonant peak in Hertz

f_2 = frequency value, in Hertz, 3 dB down from peak value, higher than f0

f_1 = frequency value, in Hertz, 3 dB down from peak value, lower than f0

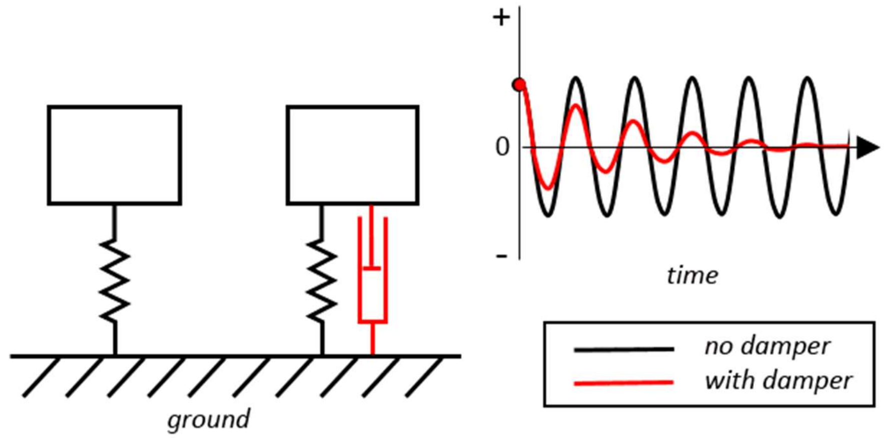
What is damping?

Damping is the energy dissipation properties of a material or system under cyclic stress.



Acceleration versus time of a damped vs undamped system response due to a load

If a mechanical system had no damping, once set in motion, it would remain in motion forever. Damping causes the system to gradually stop moving over time. The more damping present in a mechanical system, the shorter the time to stop moving.



Forms of Damping

Damping can be expressed in several different forms, including "loss factor", "damping factor", "percent critical damping", "quality factor", etc. Note that if the value of one of these damping forms is known, the other forms can be mathematically derived. It is just a matter of using the equations to transform the value to a different form.
For the purposes of this article, we will consider the "damping factor", "quality factor" and "Q" to be the same as described below:

$$\eta = \frac{1}{Q} = 2\zeta = \frac{\%Cr}{50} = \tan\phi = \frac{\Delta\omega_{3dB}}{\omega_o}$$

η = loss factor
Q = damping factor or quality factor
ζ = damping ratio
$\%Cr$ = percent of critical damping ($\%Cr = 100\% x\zeta$)
ϕ = phase angle between cyclic stress and strain

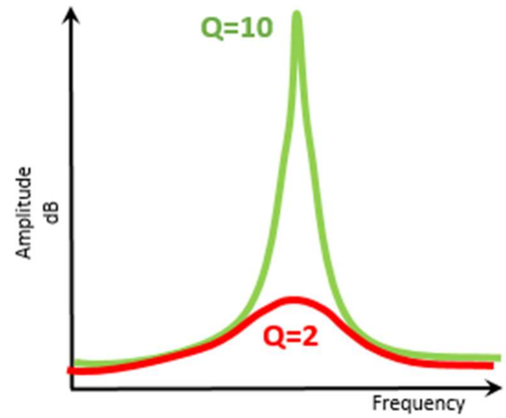
Why have different ways of expressing damping? Mostly, this is to make it easier to discuss differences in damping.

For example, perhaps two materials were tested, and the "percent critical damping" was 0.0123% in one case, and 0.0032% in the other case, which sounds like a small difference. In terms of "quality factor" or "damping factor", the difference is 4065 versus 15625, which sounds much bigger!

Instead of using long numbers with small decimal differences like percent critical damping, changing to the "quality factor" form makes the differences easier to understand.

More vs Less Damping

As the peaks in a FRF get wider relative to the peak, the damping increases (i.e., "more" damping). This means that any vibration set in motion in the structure would decay *faster* due to the increased damping.



Comparison of function with Q=2 (Red) versus Q=10 (Green). Red would be considered to have more damping than green.

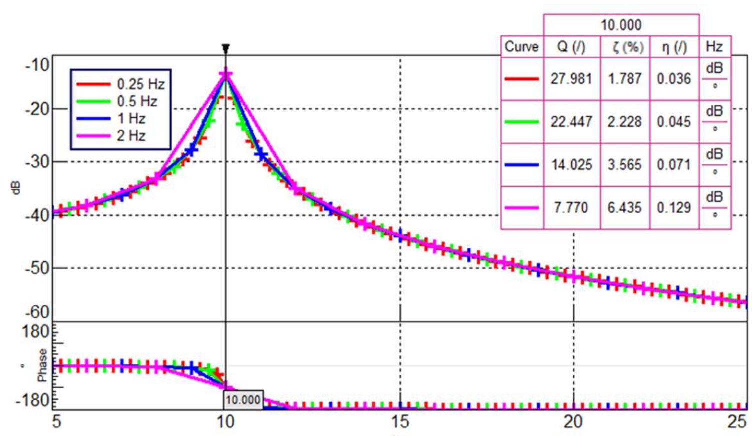
Depending on the form being used to express damping, the value may be higher or lower.
For example, the “quality factor” or “damping factor” will decrease with more damping, while “loss factor” and “percent critical damping” would increase with more damping.

Cautions

When calculating damping from a FRF, there are several items which can influence the final results:

Caution #1: Frequency Resolution

When using a digital FRF, the data curve is not continuous. It is broken into discrete data points at a fixed frequency interval or resolution. For example, this could be a 1.0 Hz spacing versus a 0.5 Hz spacing between data points. This will influence how the frequency values f0, f1 and f2 are determined.

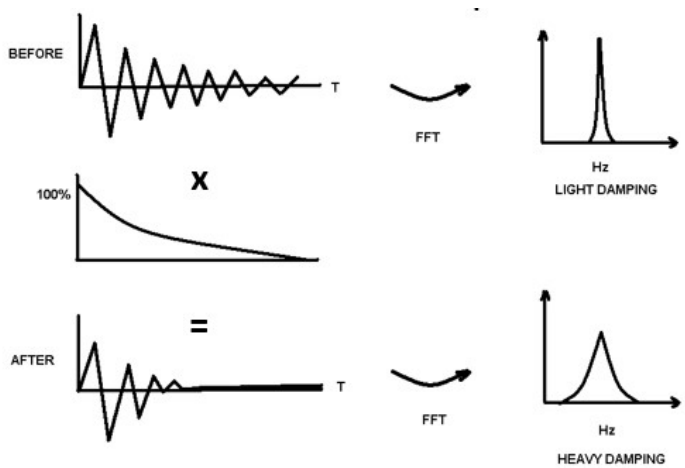


Bode plot of the same system response acquired with four different frequency resolutions: 0.25 (Red), 0.5 (Green), 1.0 (Blue), 2.0 (Magenta) Hz and corresponding change in damping values (legend).

Using a very fine frequency resolution is recommended when calculating damping using the 3dB method.

Caution #2: Window

Sometimes, when using a modal impact hammer to calculate an experimental FRF, a user may apply an Exponential Window to avoid leakage effects to the accelerometer signal. The accelerometer response is multiplied the exponential window causing it to decay quicker in time, thus increasing the apparent damping.

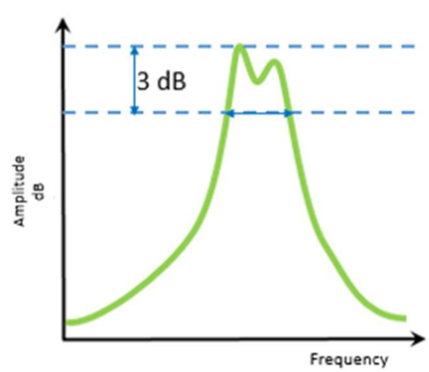


When an exponential window is applied to the time response of a FRF measurement, it increases the apparent damping in the FRF.

One should be aware the resulting FRF will yield higher damping estimates when using the "3 dB method". It is possible to back out the effects of the window and get the actual damping value. For example, in Simcenter Testlab, when performing a modal analysis curve fit, the exponential window affect is removed from the modal damping estimate automatically for the user.

Caution #3: Mode Spacing

If two modes are close to each other in frequency, it may be impossible to use the "3dB method" to determine the damping values.



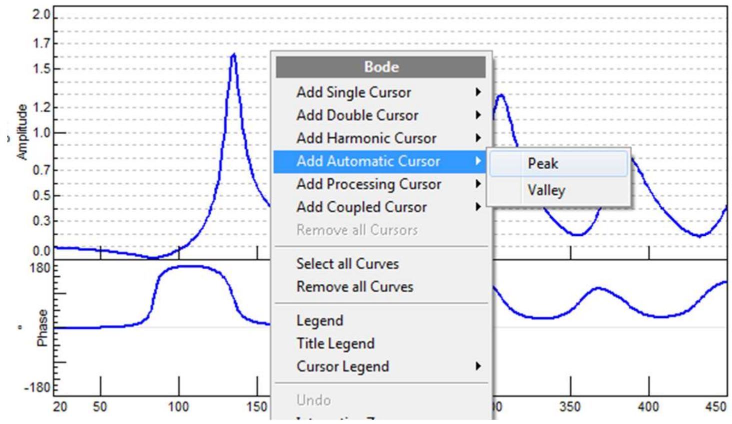
The proximity of two closely spaced modes makes the determination of Q impossible via the 3dB method.

If this is the case, there are two resonant peaks close together, which would make the peak wider than if each peak could be analyzed separately. In this case, one should use a modal curve fitter to determine the damping properly for each peak. A modal curve fitter can successfully separate the two modes influence on each other.

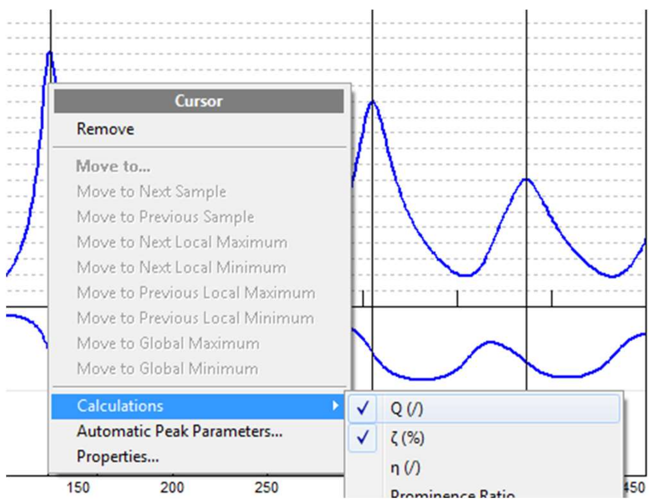
Simcenter Testlab Damping Cursor

After displaying a FRF, to calculate damping in Simcenter Testlab:

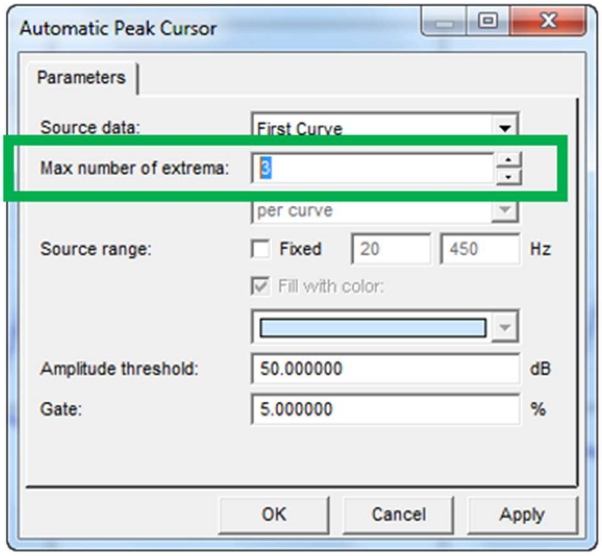
- Right click and add an "Automatic -> Peak Cursor" (or a regular SingleX cursor if one is confident in positioning it on the actual peak). Note: It is not necessary for amplitude to be in dB of the FRF, this will be accounted for correctly.



- After the cursor is on the peak value, right click on the cursor and select "Calculations -> Q" or "Calculations -> Damping Ratio" as desired



- Right click on cursor and select "Automatic Peak Parameters" and add as many peaks as desired in the "Max number of extrema" field.



- Right click on legend and choose “Copy Values” to export the values to Excel if desired

Simcenter Testlab Impact Testing

Minimal Equipment Needed

A minimum of two channels of data must be acquired to measure a structural FRF: an input force and the response output of a test object.

In an impact measurement, the input force is provided by a modal impact hammer, while the test object output response is measured with an accelerometer (*Figure 1*).

Figure 1: Impact measurement equipment

Here is a list of required equipment, some of which is shown in *Figure 1*:

- Simcenter SCADAS data acquisition system
- Computer with Simcenter Testlab Impact Testing
- Impact hammer

- Accelerometer(s)
- Appropriate cables
- Test object

Getting Started

To get started with Simcenter Testlab Impact Testing:

- Turn on the SCADAS frontend, make sure it is connected to the PC
- Double-click the Simcenter Testlab or Simcenter Testlab icon on the windows desktop (or navigate via Start menu)
- Open up the “Structures Acquisition” folder.

Double-click the Impact Testing icon, as shown in *Figure 2*.

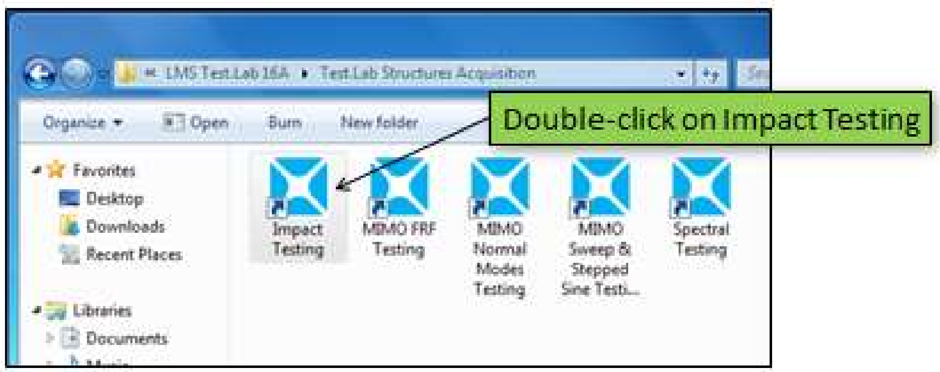


Figure 2: Double-click the Impact Testing icon to get started

Note: It is also possible to copy this icon onto your desktop to create a shortcut.

Once the software is open, a mostly grey screen is shown as in *Figure 3*. Click the white page icon in the top left to open a new project. The software will communicate to the SCADAS frontend to determine the number of available channels for the test, etc.

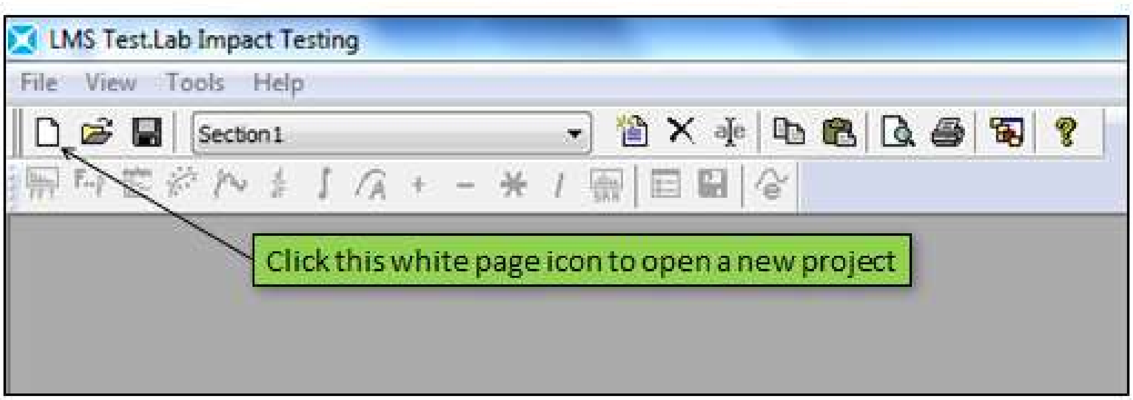


Figure 3: Click on the white page icon in the upper left to open a new project

After the new project is open, it is called "Project1.lms" by default. This is similar to how Microsoft PowerPoint starts with "Presentation1.pptx" or Microsoft Word starts with "Document1.docx". Choose "File -> Save As..." from the main menu to save a project to the desired name as shown in *Figure 4*. The project file, which will have a *.lms extension, can be stored in any directory.

Figure 4: Choose "File -> Save As..." to save the project file

There are several 'worksheets' along the bottom of the screen as shown in *Figure 4*. To setup and perform an impact measurement, one works through the worksheets from left to right, starting with the 'Documentation' worksheet.

In the 'Documentation' worksheet, one can store pictures of the test and create documentation. See the knowledge base article called 'What's up with the Documentation worksheet'.

Channel Setup

After documenting the test, go to the 'Channel Setup' worksheet as shown in *Figure 5*. An Excel-like table contains a list of all the channels in the connected SCADAS frontend. Each row corresponds to one channel.

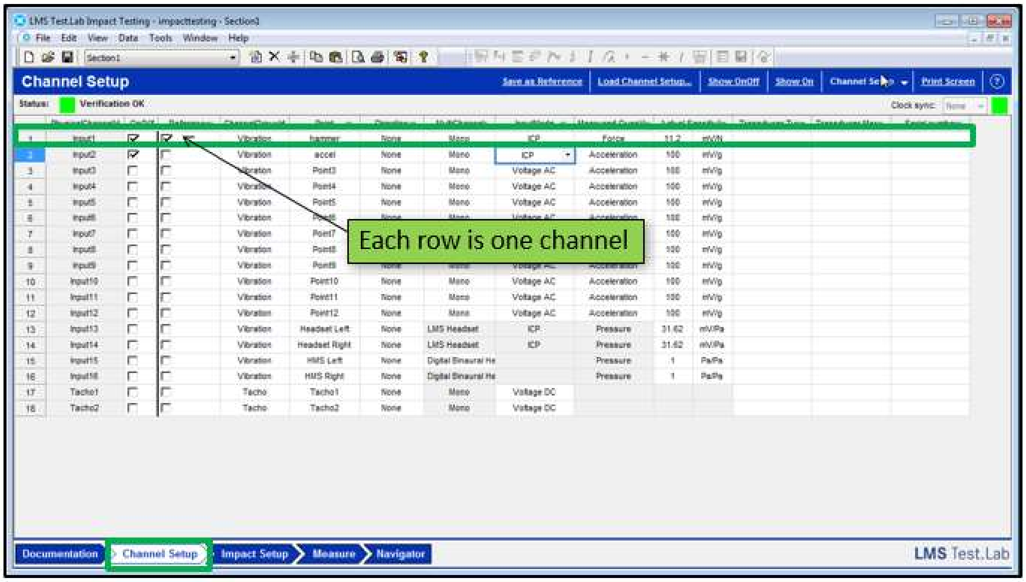


Figure 5: In the 'Channel Setup' worksheet, each row corresponds to one channel.

Assume a hammer is plugged into Channel 1, and an accelerometer is plugged into Channel 2. Enter following in the 'Channel Setup' worksheet as shown in *Figure 6*:

- Turn 'ON' two channels (Input1 and Input2).
- Turn 'ON' reference field for the hammer channel. The impact hammer should be marked as reference

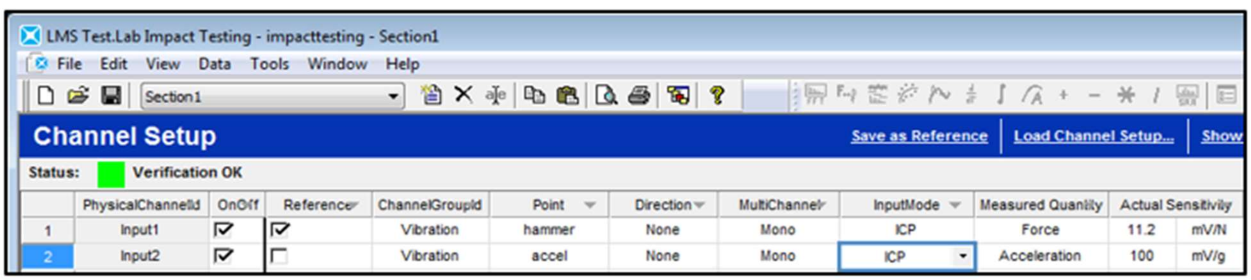


Figure 6: Setup the hammer and accelerometer information as shown

For 'Input1', the impact hammer channel, enter the following in the fields as shown in *Figure 6*:

- *ChannelGroupId:* Vibration
- *Point*: This field is the description of where the impact hammer is applied to the structure. This is a free description field where anything can be entered (in this example: 'hammer'). If the data will be used for a modal animation, the point should be filled in as component: number, for example: table:1 or frame:15
- *Direction:* Selections include +X, -X, +Y, -Y, +Z, -Z. The direction is only needed if the FRF data collected will be used in a modal animation with a geometry, otherwise it can be set to 'None'

- *InputMode:* If using an ICP or IEPE transducer, this field should be set to 'ICP'. This means the transducer will be powered by the SCADAS frontend directly. The transducer is therefore wired directly into the frontend with no external signal conditioners required
- *Measured Quantity:* Force
- *Actual Sensitivity:* Enter the calibration value found on the calibration sheet. For an impact hammer this will be in units of mV/N as shown in *Figure 7*.

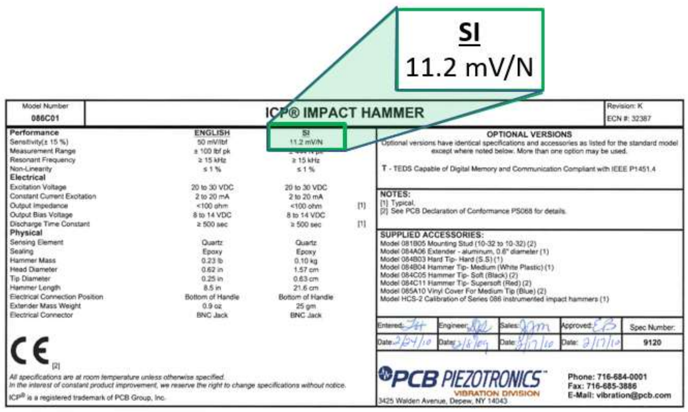


Figure 7: The sensitivity value of the transducer can be found on the calibration sheet

For 'Input2', the accelerometer channel, enter the following in the fields as shown in *Figure 6*:

- *ChannelGroupId:* Vibration
- *Point*: This field is the description of where the accelerometer is located on the structure. This is a free description field where anything can be entered (in this example: 'accel'). If the data will be used for a modal animation, the point should be filled in as component: number, for example: table:1 or frame:15
- *Direction:* Selections include +X, -X, +Y, -Y, +Z, -Z. The direction is only needed if the FRF data collected will be used in a modal animation with a geometry, otherwise it can be set to 'None'
- *Multi-Channel:* When using a triaxial accelerometer, the Multi-channel field can be activated to make setting up the directions easier. See the 'Cool Channel Setup Tricks for Accelerometers' knowledge base article
- *InputMode:* If using an ICP or IEPE transducer, this field should be set to 'ICP'. This means the transducer will be powered by the SCADAS frontend directly. The transducer is therefore wired directly into the frontend with no external signal conditioners required
- *Measured Quantity:* Acceleration
- *Actual Sensitivity:* Enter the calibration value found on the calibration sheet. For an accelerometer this will be in units of mV/g. If the accelerometer has a Transducer Electronic Data Sheet (TEDS) chip with the calibration value stored directly in it, it can be accessed by select "Read Teds" in the upper right corner as shown in *Figure 8*.

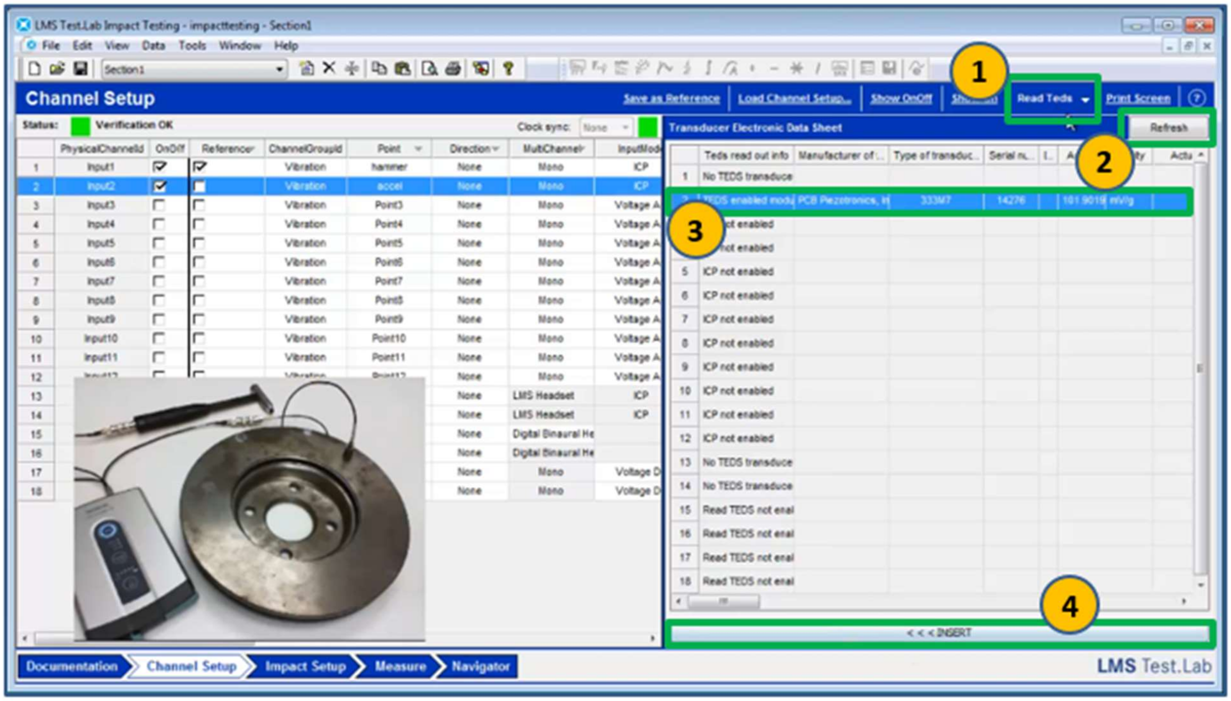


Figure 8: TEDS allows the reading the sensitivity, serial number, model number of the accelerometer directly from a chip embedded in the transducer

Transducers with TEDS capabilities make it easy to enter the calibration value, serial number, etc even when the accelerometers are already mounted on the structure:

1. Change 'Channel Setup' to 'Read TEDS'
2. Press the 'Refresh' button
3. Any TEDS transducers appear in the list
4. Press the 'INSERT' button at the bottom of the worksheet

Not all transducers have TEDS capability, or it may be optional when ordering. All Simcenter SCADAS frontends come with the ability to read TEDS as a standard feature.

Impact Measurement Setup

With the channel information entered, the impact measurement can be prepared. Select the 'Impact Setup' worksheet to do the following:

- Setting the trigger
- Checking the input frequency range
- Determining a window.

In this worksheet, the impact measurement can be setup by following the sub-worksheets at the top of the screen as shown in *Figure 9*.

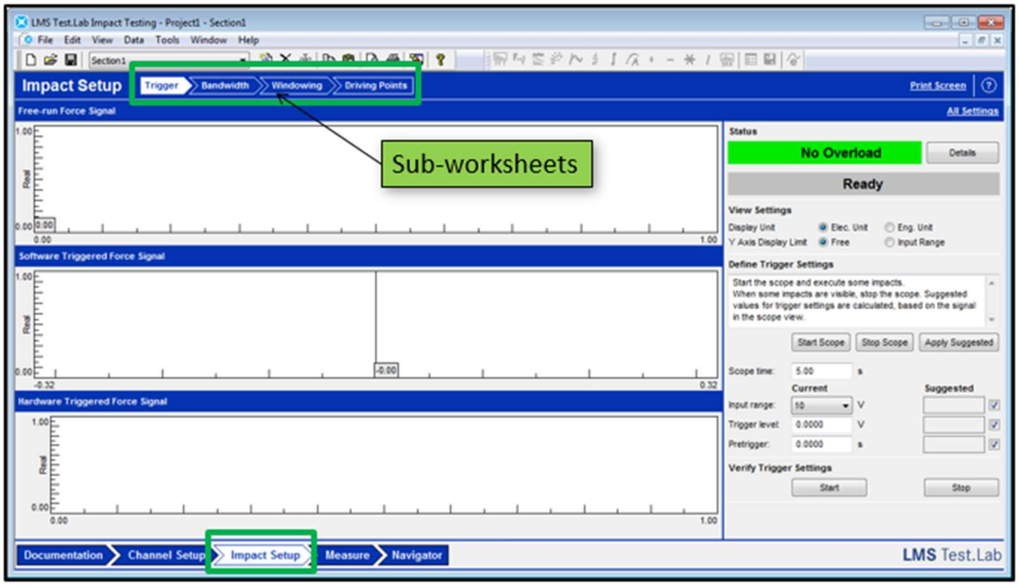


Figure 9: In the 'Impact Setup' worksheet, there are sub-worksheets to step thru an impact measurement setup. The sub-worksheets are at the top of the screen and called 'Trigger', 'Bandwidth', 'Windowing', and 'Driving Points'

The sub-worksheets, located at the top of the screen, are 'Trigger', 'Bandwidth', 'Windowing', and 'Driving Points'.

Trigger

The first step is to determine a trigger level that will make the measurement start automatically when the impact hammer strikes the object. In the 'Impact Setup' worksheet (*Figure 10*), there are instructions for setting up the hammer on the right side.

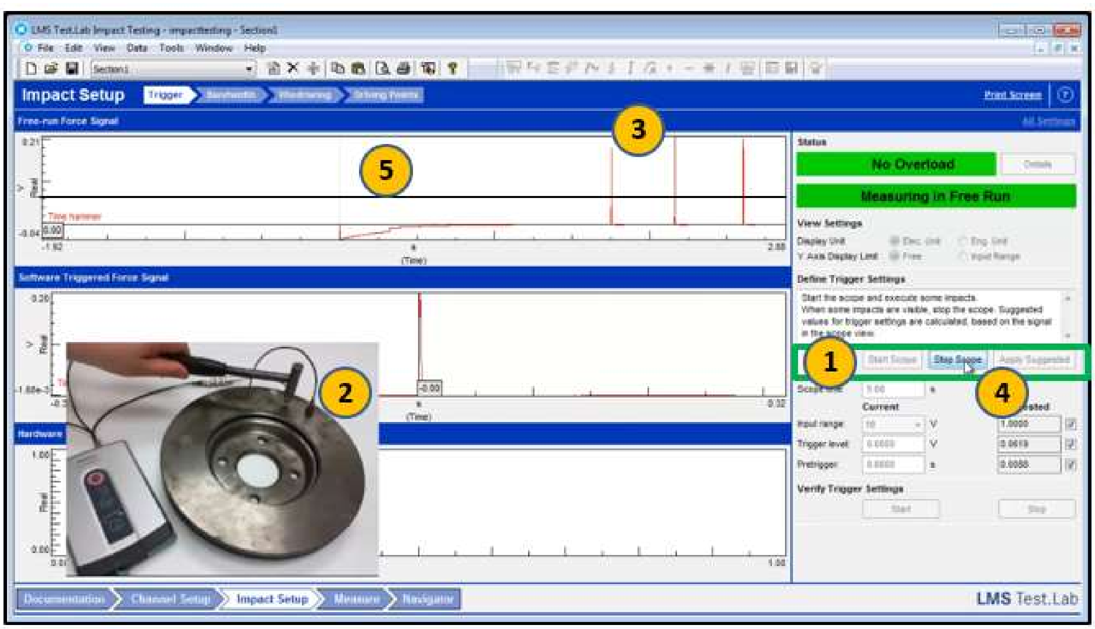


Figure 10: 'Trigger' sub-worksheet is used to set the trigger

1. Press the 'Start Scope' button
2. Practice hitting the test object several times with the impact hammer
3. Several impacts should show on screen
4. Press 'Stop Scope' and 'Apply Suggested' buttons
5. A black horizontal line is drawn on the screen which is the calculated trigger level

Note: Along with the trigger level being calculated, a pre-trigger is also set. The pre-trigger is a small amount of time before the measurement is triggered to ensure the entire impact is recorded.

Bandwidth

In the next step, called 'Bandwidth', the input spectrum of the impact hammer is evaluated. To get a good Frequency Response Function (FRF) measurement, the impact hammer force should be uniform, or the same level, across the desired frequency range.

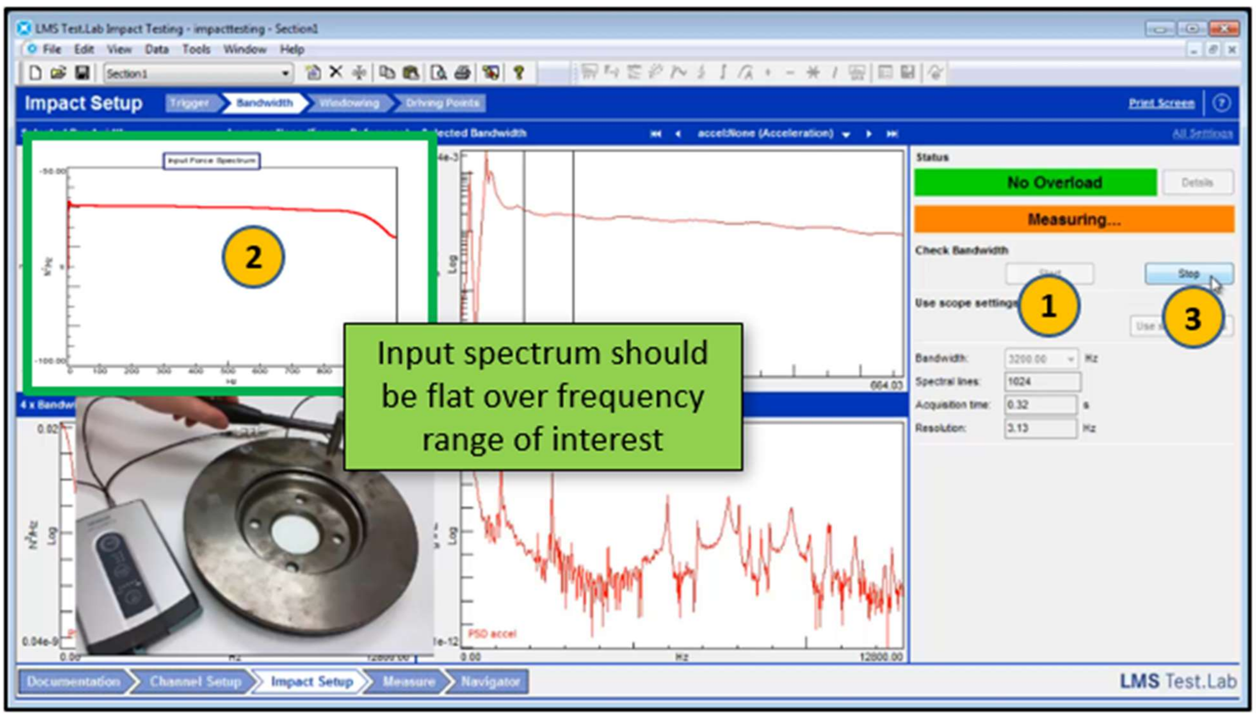


Figure 11: 'Bandwidth' sub-worksheet

Depending on the company standard, the input force spectrum should be flat, with a drop-off of no more than 3 to 10 dB. Note that the roll off of the top 20% of the frequency spectrum is due to the anti-aliasing filters in the SCADAS frontend.

1. Press the 'Start' button
2. Take two hits. In the upper left display, the input force spectrum is displayed.
3. Press 'Stop' button

If the force does not span the frequency range before it rolls off, the modal impact hammer tip can be changed. For example, a soft rubber tip can be replaced by a hard metal tip to excite a higher frequency range. See the Test Knowledge base article 'What modal impact hammer tip should I use?'.

Windowing

Next, enter the 'Windowing' sub-worksheet (*Figure 12*). Using a sample acquired measurement, a suitable window will be determined to assure that the measurement is not affected by leakage.

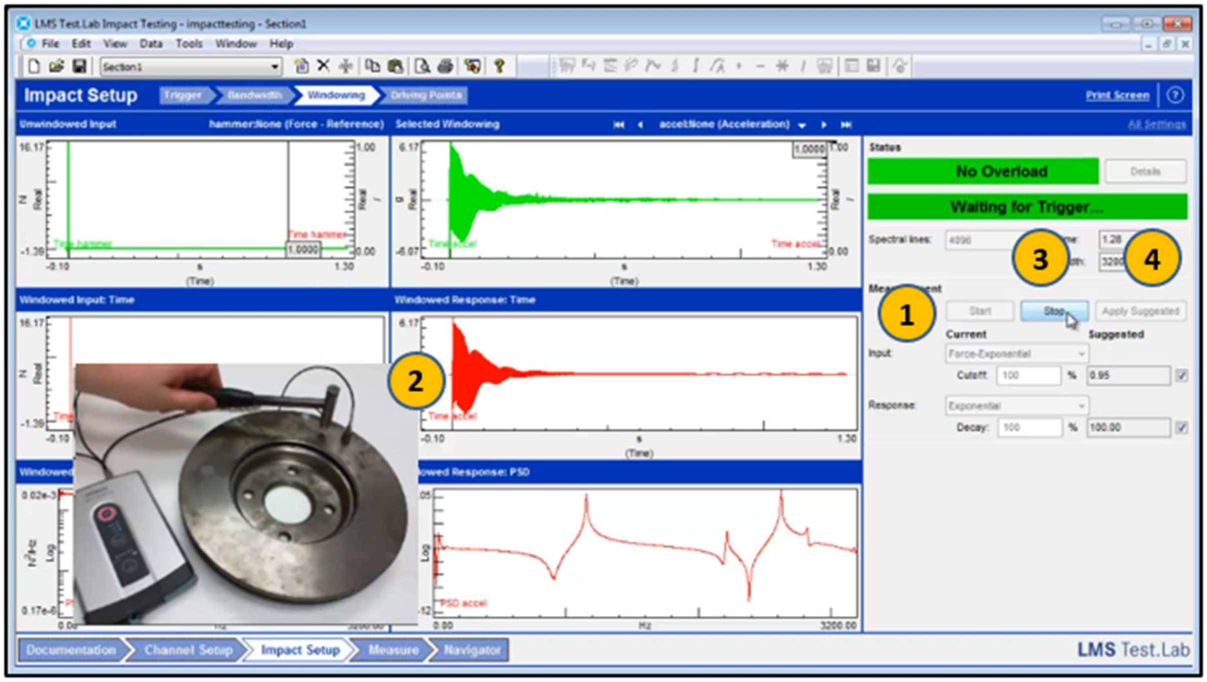


Figure 12: 'Windowing' sub-worksheet

Do the following:

1. Press the 'Start' button
2. Impact the test object once
3. Press the 'Stop' button
4. Press 'Apply Suggested' button

Ideally, no windows will be needed for the input or response. If the accelerometer response dies out completely in the measurement time frame, the software will be recommend an exponential window of 100%. An exponential window of 100% is equivalent to not applying a window, i.e., there is no reduction in the amplitude of the signal/window over the measurement time, as shown in the top graph of *Figure 13*.

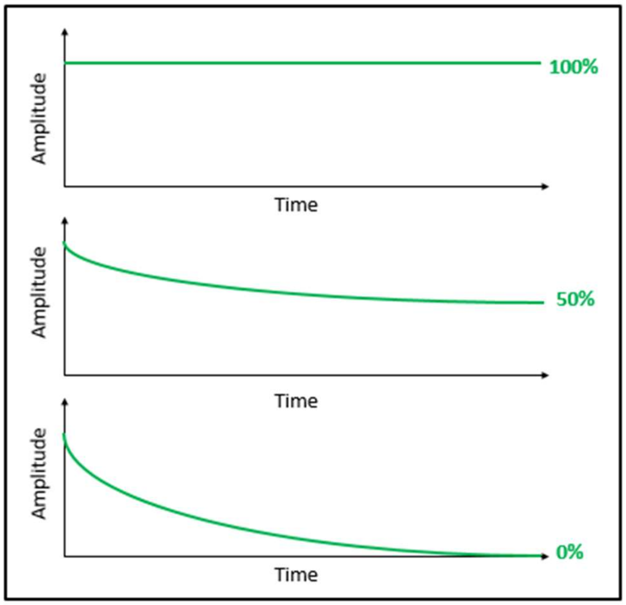


Figure 13: Various exponential windows and associated percentages

If the accelerometer response does not decay to zero within the measurement time frame, the response is multiplied by an exponential window to assure that it does decay to zero. An exponential window starts with a value of 1:

- A 100% exponential window still has a value of 1 at the end of measurement
- A 50% exponential window reduces the windowed signal amplitude by 50% at the end of the measurement
- A 0% exponential window reduces the amplitude of the windowed signal to zero by the end of the measurement

As shown in *Figure 14*, if an exponential window was applied, additional artificial damping is added to the FRF measurement.

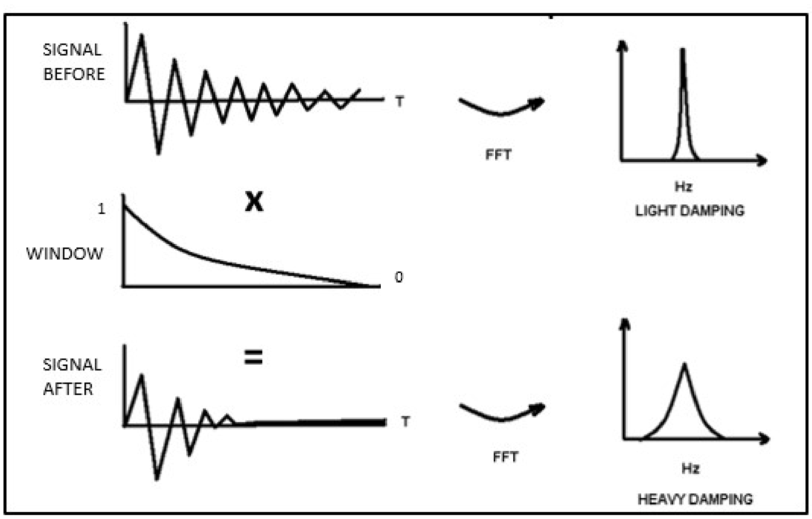


Figure 14: If used, an exponential window will add additional artificial damping to a measurement

This artificial damping is one reason to try and avoid a window. If the measurement does not reduce to zero by itself in the measurement time, it is preferable to increase the measurement time, rather than apply a window. See the Testing Knowledge Base article 'How to calculate damping from a FRF?' for more details.

Measure

With setup finished, the actual FRF measurement can be performed in the 'Measure' worksheet as shown in *Figure 15*.

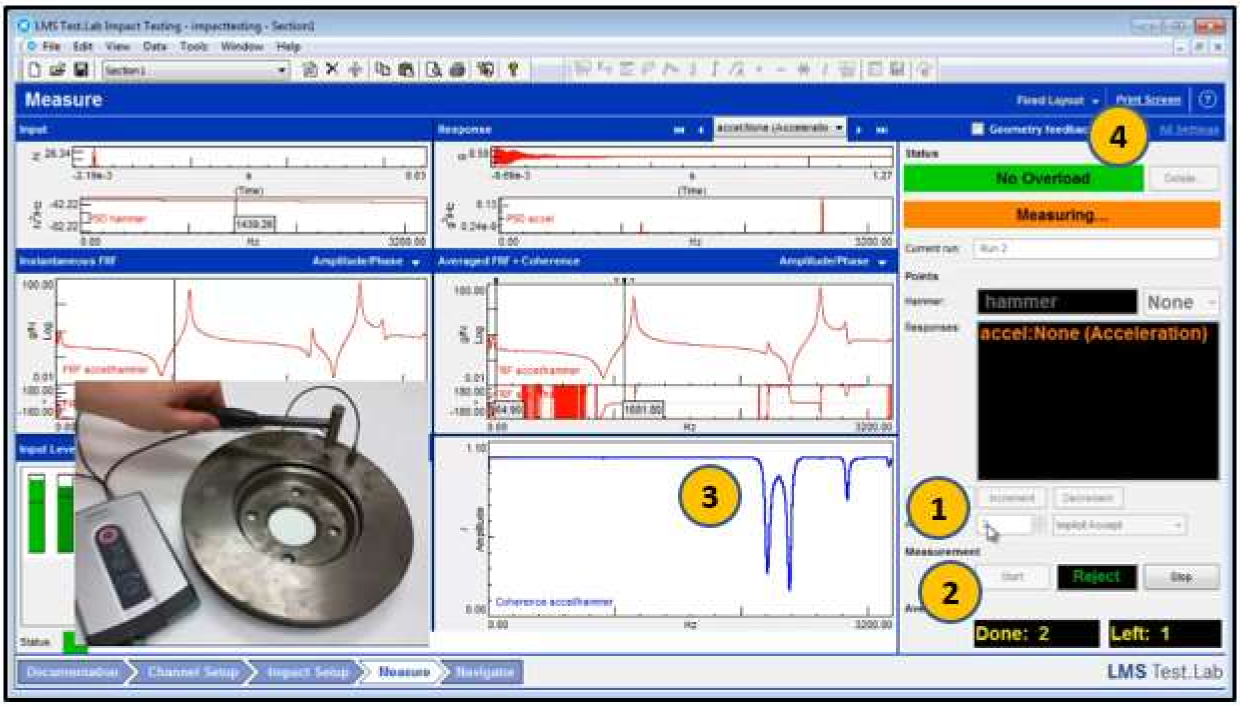


Figure 15: 'Measure' worksheet

In the 'Measure' worksheet:

1. Set the number of averages as desired
2. Press the 'Start' button. Hit the object.
3. Monitor the FRF and coherence after each hit. Measurements are automatically accepted by default. If an undesirable measurement occurs, press the 'Reject' button.
4. Optional: Under 'All settings', turn on double hit and overload detection/rejection if desired

When doing the FRF measurement, be sure to monitor the coherence function in the bottom right display. The coherence function indicates how much of the output is due to the input by checking the variation from measurement average to measurement average:

- A coherence of close to 1 indicates that the measurement is repeatable
- A coherence of 0 indicates the measurement is not repeatable

The majority of the frequency range should ideally have coherence value close to 1, with the only exceptions occurring at anti-resonance frequencies where the response is low and affected by the measurement system noise (*Figure 16*).

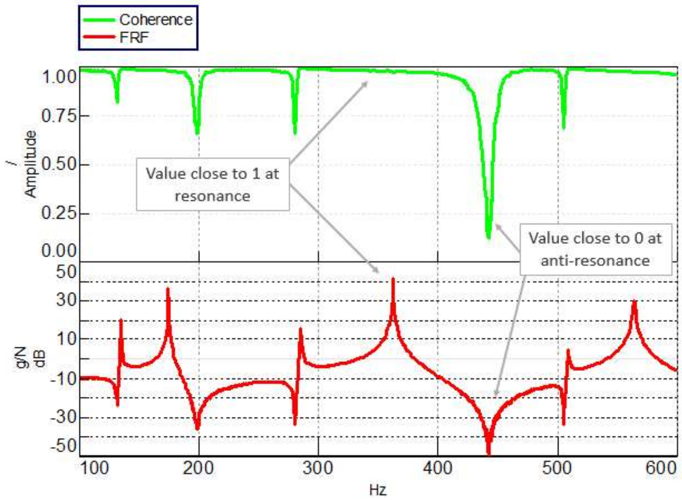


Figure 16 – Coherence (top display) should be close to 1, with the only exceptions occurring at anti-resonances in the FRF (bottom display)

For more information on coherence, see the knowledge base article 'What is a Frequency Response Function (FRF)?' When the averages are completed, the FRF measurement is automatically stored in the Simcenter Testlab project file.

All Settings Dialog

There are more settings available in both the Impact Setup and Measure worksheets by pressing 'All Settings' in the upper right hand corner as shown in *Figure 17*.

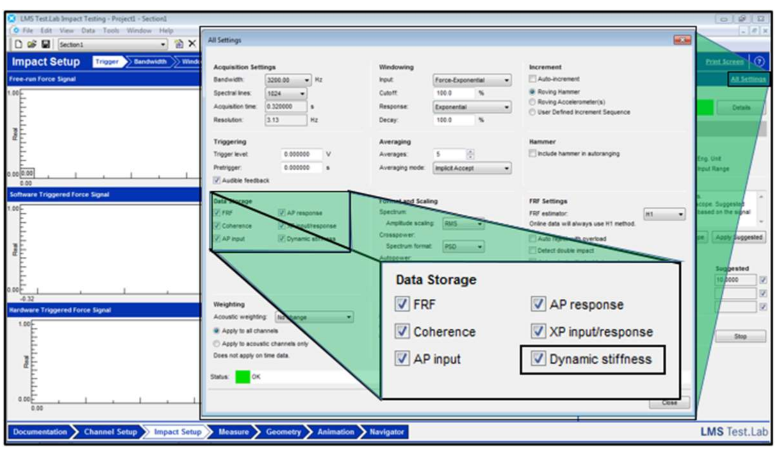


Figure 17 - By pressing the 'All Settings' in the upper right-hand corner, additional settings can be made, like measuring dynamic stiffness in addition to the FRF

Additional settings include:

- Measuring dynamic stiffness in addition to the FRF
- Enable/disable sounds during measurement
- Switch between roving hammer and roving accelerometer

Modal Impact Hammer Tips

What modal impact hammer tip should I use?

Getting high quality frequency response function (FRF) measurements is key to identifying the resonant frequencies of a structure. Using the appropriate hammer tip is a big part of getting a quality FRF measurement.

Background

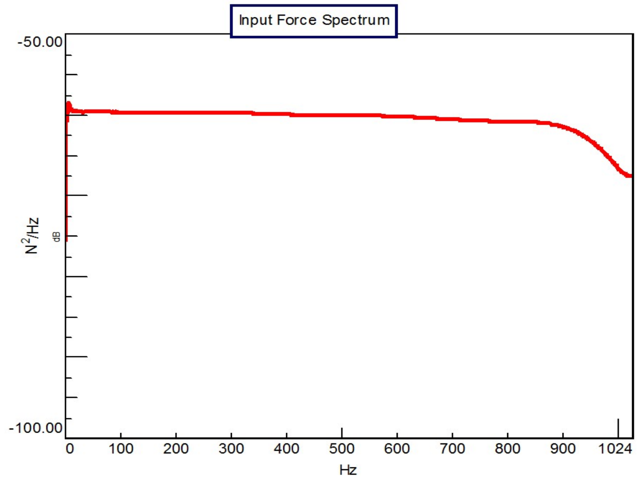
During a modal impact test, a Frequency Response Function (FRF) is calculated to determine the natural frequencies of the structure under test. A FRF is a measure of the systems output in response (usually acceleration, velocity or displacement) to a known input (usually force).

$$FRF = \frac{output}{input}$$

To calculate an experimental FRF function, both the input and output response signals are measured using sensors, like load cells and accelerometers.

For a good measurement the input force must:

- Excite a broad range of frequencies at high amplitude (eg, above the noise floor of the instrumentation)
- Have amplitude evenly distributed versus frequency



An ideal input force should be uniformly distributed vs. frequency

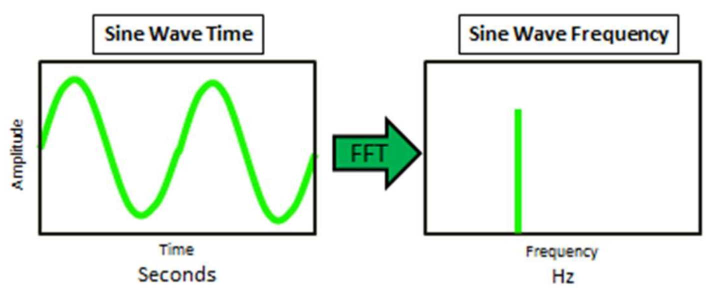
The general idea is that resonant frequencies can be easily identified by applying the same force level across the entire frequency range. Frequency peaks in response to the force correspond to resonant frequencies.

Time vs Frequency domain

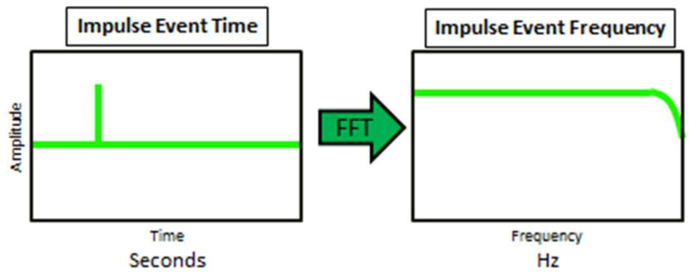
The width of the input force is controlled by the length of time of the impact pulse. The shorter the impulse duration, the broader the frequency domain response becomes.

In general, there is an inverse relationship between the time and frequency domain of a signal. Signals with short durations is time, have a broad response in frequency and vice versa.

For example, a sine wave, which is continuous in the time domain has a narrow frequency spectrum.



Short, transient pulses in the time domain, on the other hand, have a wide frequency spectrum.



Tip Adjustments

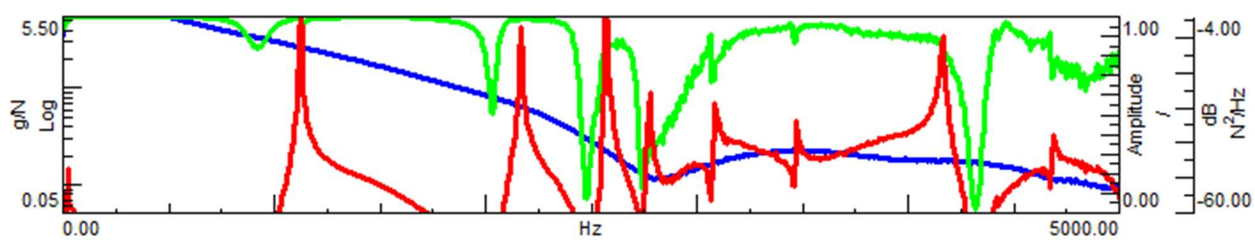
So, a short pulse is desired for a wide excitation frequency range, but how is this achieved in practice?

The input force frequency range can be controlled by changing the hammer tip in two ways:

- Hammer Mass – Decreasing the mass of the hammer tip causes the hammer to contact the structure for a shorter amount of time. The reduced mass allows the hammer to reverse directions more easily after hitting the structure, reducing the time it is in contact.
- Hammer Stiffness – Increasing the stiffness of the tip allows shortens the duration that the hammer is in contact with the structure. For example, one could replace a rubber tip with a metal tip.

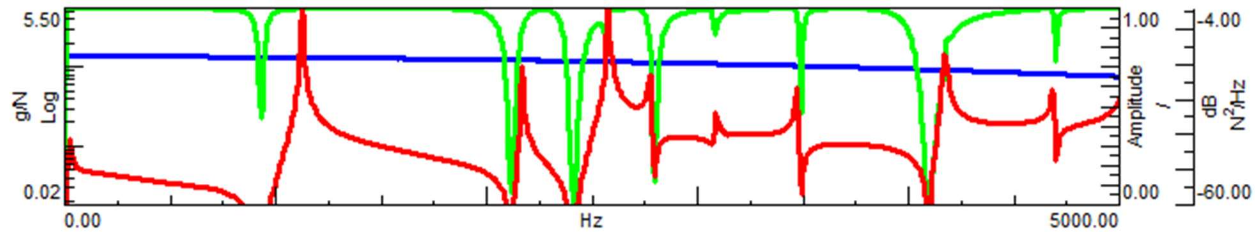
The desired result is a clean FRF over the full frequency range of interest and a relatively even input spectrum throughout that same frequency range.

If the FRF becomes noisy at higher frequencies and the input spectrum drops off significantly, this is an indication that our hammer tip may be too soft.



Tip "too" soft - Green: coherence, Red: FRF, Blue: input force spectrum

If all the modes of the structure are excited, far beyond the frequency range of interest, there may be noise on the FRF at lower frequencies which indicates that the hammer tip may be too hard. This would be caused by creating “out-of-band” overloads.

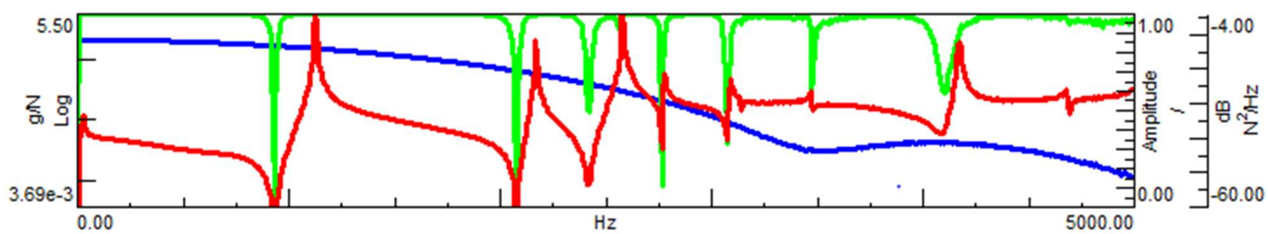


Tip "too" hard - Green: coherence, Red: FRF, Blue: input force spectrum

The mass of the hammer is also important for assuring that enough force is being input into the structure to excite it. So a heavier hammer tip may be needed to ensure that the force levels are high. For example, hitting a 50 ton boat with a one pound hammer will not excite the modes of the boat.

Conclusion

Like Goldie Locks looking for a bowl of porridge or a place to sleep, we are looking for hammer tip that is 'just right'. The correct tip will cause enough energy to excite the full frequency range of interest, but not significantly beyond. The correct tip will also ensure that enough force is being input into the structure to excite the modes of the structures.



Tip "just right" - Green: coherence, Red: FRF, Blue: input force spectrum

Roving Hammer vs Roving Accelerometer

"To rove or not to rove, that is the question" - Anonymous

When performing a multiple point modal test with a force impact hammer and accelerometer, there is a choice to move or "rove" either the hammer or the accelerometer between measurement locations.

Roving the impact hammer has some advantages over roving the accelerometer. When moving the accelerometer, one has to unmount and remount the accelerometer at the different measurement points. Moving the accelerometer also changes the mass distribution on the structure, which could alter the natural frequencies.

Moving the hammer is not without considerations however. Because the hammer is a single input and a triaxial accelerometer measures output is three directions, one must take care to ensure the resulting mode shapes are complete.

Question

I am doing a modal impact test on a structure with an impact hammer and a single triaxial accelerometer. There will be nine measurement locations as shown in *Figure 1*.

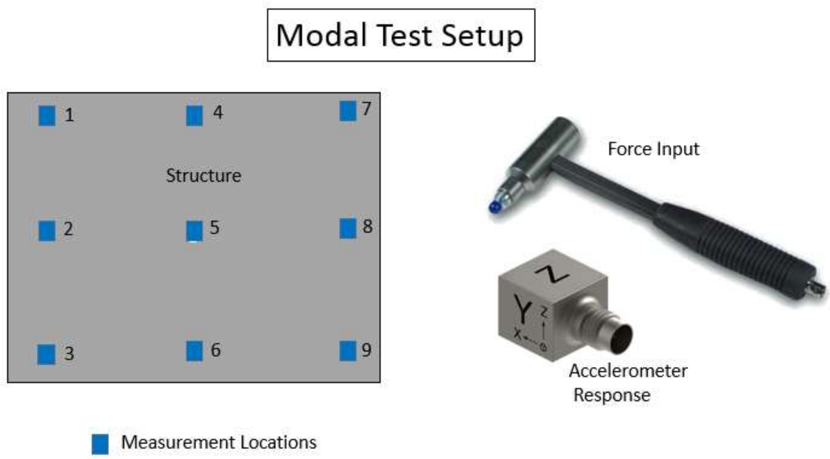


Figure 1: Modal test setup (with impact hammer and triaxial accelerometer) and structure (with nine measurement locations)

I want good mode shapes. Does roving the accelerometer versus roving the hammer make a difference in the modal results?

Answer

Yes! It can make a difference in the mode shape results if you are not careful. It is possible to perform the test and have incomplete mode shape information.

The accelerometer measures in 3 directions and the hammer measures in only one direction. If you move one versus the other, this difference needs to be taken into account.
Background

When performing a modal analysis test, consider the possible input references and possible output responses associated with the different measurement locations. A table of the possible inputs and outputs for a nine location test is shown in *Figure 2*.

Possible Inputs and Outputs

	A_{1x}	A_{1y}	A_{1z}	A_{2x}	A_{2y}	A_{2z}	…	A_{9x}	A_{9y}	A_{9z}
F_{1x}										
F_{1y}										
F_{1z}										
F_{2x}										
F_{2y}										
F_{2z}										
…										
F_{9x}										
F_{9y}										
F_{9z}										

Figure 2: For a nine measurement location modal test, there are 27 possible inputs and 27 possible outputs that can be measured.

When measuring the Frequency Response Functions (FRFs) between the nine measurement locations:

- There are 27 possible input locations for applying the input force. At any given location (1, 2, 3, 4, etc.) the force (F) can be applied in three possible directions: x, y, and z. Possible inputs are therefore F1x, F1y, F1z, F2x, F2y, F2z, etc.

- There are 27 possible output locations for measuring the acceleration response. At any given location (1, 2, 3, 4, etc) the acceleration (A) can be applied in three possible directions: x, y, and z. Possible outputs are therefore A1x, A1y, A1z, A2x, A2y, A2z, etc.

To have a proper mode shape, where the phasing of all the measurement points are aligned properly, the measurements must all have a *common reference*. A *common reference* means that either a complete row or complete column of the table shown in *Figure 3* must be measured.

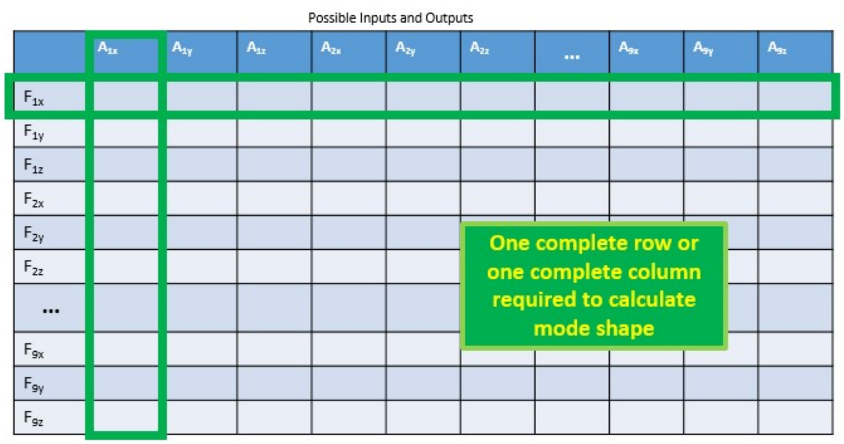


Figure 3: A complete row or complete column of the measurement table must be measured to ensure consistent phasing for the mode shape.

Roving Accelerometer

Consider the case where the accelerometer is moved between the nine different locations while the hammer is applied to the same location (F1z) as shown in *Figure 4*.

The accelerometer, which measures in three directions, is first located at measurement location 1. The A1x, A1y, A1z outputs are measured at one time with respect to F1z.

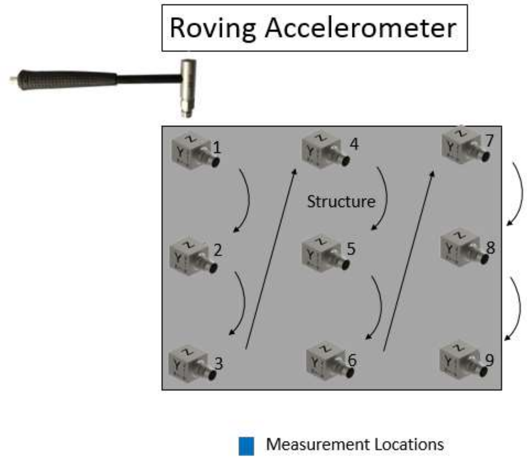


Figure 4: Roving accelerometer outputs with fixed hammer inputs

After measuring the three FRFs for location 1, the accelerometer is then moved to location 2. The three outputs of location 2 (A2x, A2y, A2z) are measured with respect to the hammer input at location 1 in the z direction. This continues for locations 3 thru 9.

After all nine locations are measured, one complete row of the table is filled in as shown in *Figure 5*.

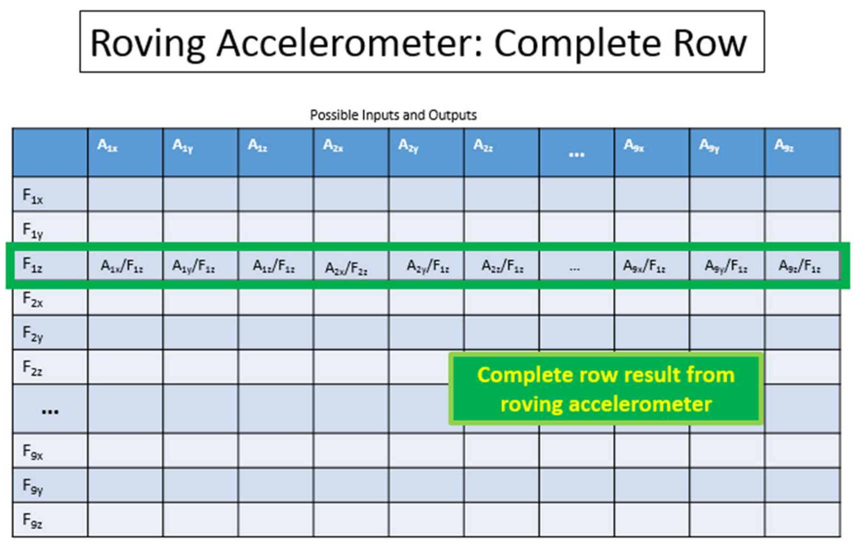


Figure 5: Roving accelerometer modal test has one complete row: all outputs measured with respect to one input

The complete row with common reference assures that the phasing of all the measurements are consistent, so that a proper mode shape can be calculated.

Roving Hammer

Instead of roving the accelerometer, the hammer could be roved while the accelerometer stays fixed.

In this case, the accelerometer measures output A1x, A1y, and A1z. The hammer impacts at F1z, which happens to be perpendicular to the surface of the test object. The hammer is then moved to F2z, F3z, etc. This happens until impacts are done at all nine locations as shown in *Figure 6*.

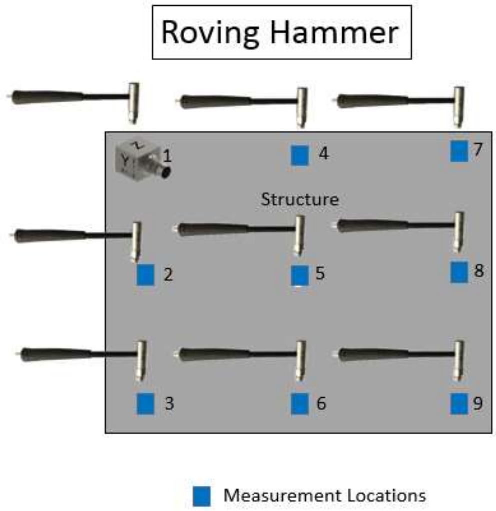


Figure 6: Roving impact hammer inputs with fixed accelerometer outputs

The result of this test is *NOT* a complete row or column in the table after the test is complete as shown in *Figure 7*.

Roving Hammer: Incomplete

Possible Inputs and Outputs

	A_{1x}	A_{1y}	A_{1z}	A_{2x}	A_{2y}	A_{2z}	...	A_{9x}	A_{9y}	A_{9z}
F_{1x}										
F_{1y}										
F_{1z}	A_{1x}/F_{1z}	A_{1y}/F_{1z}	A_{1z}/F_{1z}							
F_{2x}										
F_{2y}										
F_{2z}	A_{1x}/F_{2z}	A_{1y}/F_{2z}	A_{1z}/F_{2z}							
...										
F_{9x}										
F_{9y}										
F_{9z}	A_{1x}/F_{9z}	A_{1y}/F_{9z}	A_{1z}/F_{9z}							

NOT a complete row or column from roving hammer

Figure 7: Roving hammer modal test DOES NOT have a complete row or column

There is NOT a complete row or column with the roving impact hammer test. The impact hammer was only used in one direction at each different measurement location, instead of being used in all three directions. This is an easy oversight when using a modal impact hammer.

When trying to use an impact hammer on a flat surface, it is very difficult to impact in other than the perpendicular direction to the surface. This is why it is often easy to overlook the need to impact in all three directions when performing an experimental modal analysis.

If the impact hammer was used to input in all three directions (x, y and z) at each location, then three complete columns would have been created. There were 27 possible inputs and 27 possible outputs

(9 locations, three directions each) in this modal test. This means to get a complete row or column, 27 FRF measurement functions must be done at a minimum:

- With the roving accelerometer case, only nine actual measurements were required. Each time the accelerometer was moved, three measurements of the associated row in the table were completed.
- In the roving impact hammer case, twenty-seven separate measurements must be done. For a given modal impact hammer input, only one measurement is added to a single column in the table.

If the modal impact hammer is applied in all three directions at each measurement location, a proper modal test would be done, with three complete columns of the table.

Conclusion

While roving a hammer versus roving an accelerometer may outwardly appear to be the same for a modal test, the results can yield an incomplete mode shape. It is important to take care to make sure a complete set of outputs with respect to consistent input are measured.

Modal Testing: Driving Point Survey

A driving point survey is an important part of any modal analysis test. By making several driving point measurements at various points around our structure we can "survey" the structure to help determine the optimal location for excitation. Consider the three driving point FRFs in Figure 1 below.

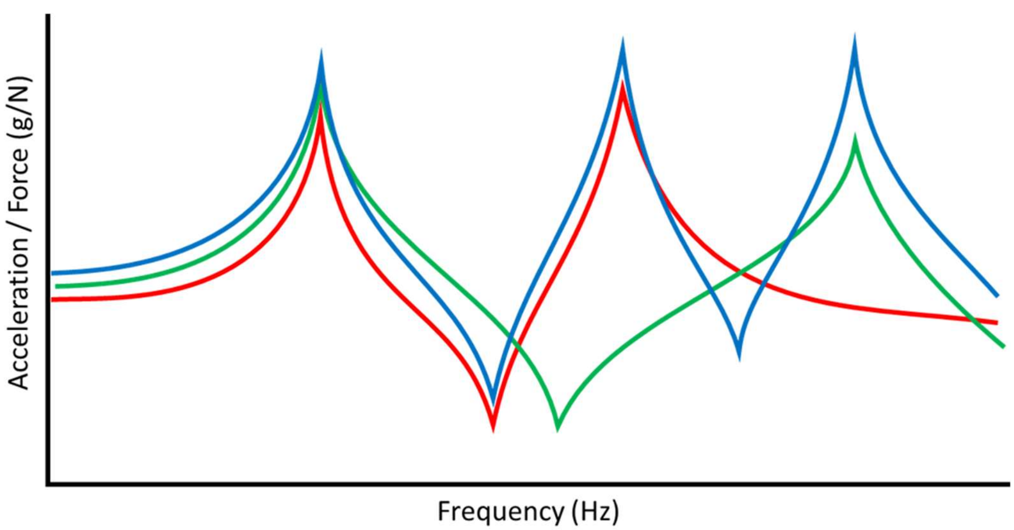


Figure 1: Three different driving point FRFs from the same structure.

Which FRF in Figure 1 contains the most information about the natural frequencies of the structure? It is clear the blue FRF shows three peaks, while the red and the green FRFs each only show two peaks. By looking at the set of FRFs and comparing them, we can conclude that the location used to measure the blue FRF is the best driving point of the three, because the blue FRF contains the most peaks. This example highlights why performing a driving point survey is an important step in a modal test!

This article will explain:

- What a driving point measurement is
- Why driving point measurements are important
- How to perform a driving point survey in Simcenter Testlab Impact Testing

What is a "driving point" measurement?

A driving point measurement is a dynamic measurement where the force input from the hammer (or shaker) and the response output from the accelerometer are measured at the same point on the structure, and in the same direction. (See Figure 2)

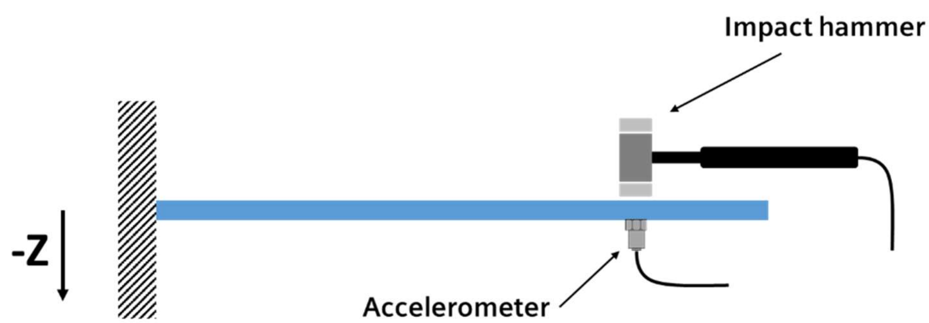


Figure 2: A driving point measurement on a cantilever beam. Both the force and response are being measured at the same location on the structure, and in the same direction (-Z).

However, sometimes it may not be physically possible to collocate the impact and the response measurement as shown in Figure 2, and a compromise must be made. For instance, imagine making a driving point measurement on a fuel tank, or other sealed vessel, where the interior surface is not accessible (see Figure 3). In this case, the accelerometer needs to be mounted on the same surface we will impact with the hammer and should be located as close as possible to the impact location. It is important that the accelerometer be located where it will not interfere with the impact itself. Under no circumstances should the impact hammer be striking the accelerometer, or making any direct contact, as this will distort the FRF we are trying to measure (and likely overload the signal coming from the accel).

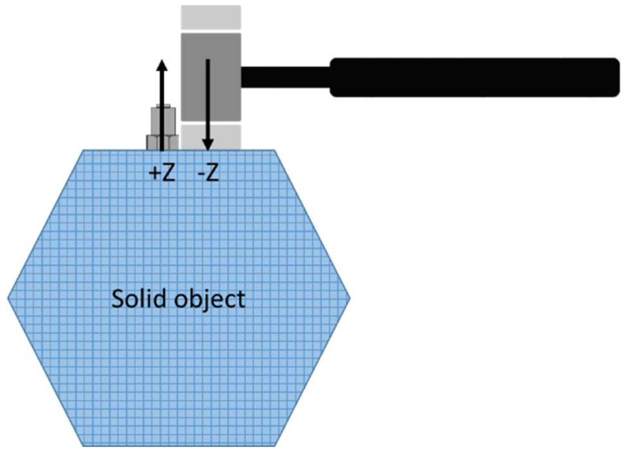


Figure 3 - When necessary, place accelerometer next to, and as close as possible, to the impact location.

While it is critical that the force and the response be measured at the same location and direction, the polarity of the two measurement directions can be different. For example, the hammer can be measuring in the -Z direction, while the accelerometer is measuring in the +Z direction. This is often the case in situations where the accel is mounted next to the impact location like is shown in Figure 3. As long as both hammer and accel signals are measuring along the same axis, the driving point measurement is valid.

Why is a driving point measurement important?

Very simply - a driving point FRF shows all of the modes of a structure that are excited by impacting at a particular location. If impacting at a point on a structure does not adequately excite a particular mode shape, the peak corresponding to that mode will be missing from the FRF. Why? To find out, let's take a look at how the FRFs from Figure 1 were generated.

Consider the cantilever beam in Figure 4a below. By placing the accelerometer at point R and impacting at that same location, we get the red FRF shown in Figure 4b. Next, we move the accelerometer and impact at point G, which generates the green FRF. Lastly, we move the accelerometer and impact to location B, and measure the blue FRF. Again, we notice that there are three peaks in the blue FRF, and only two in the red and green FRFs.

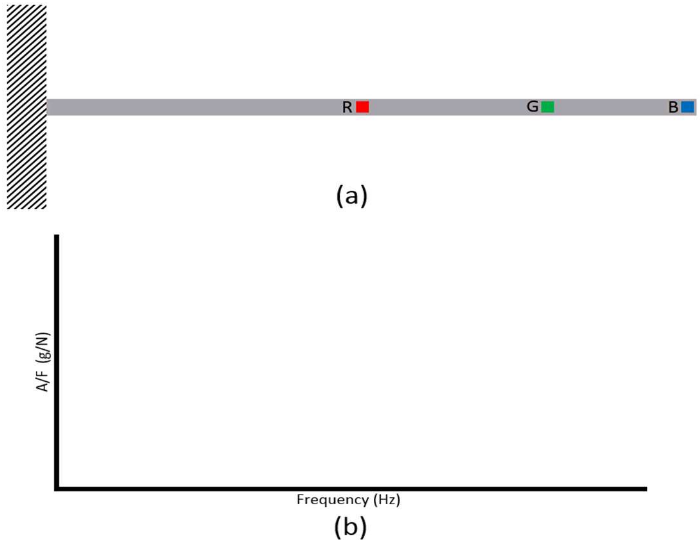


Figure 4: (a) Cantilever beam with three driving points –"R", "G", and "B"; (b) driving point FRFs from each location

To understand why this is, let's take a look at the first three modes of vibration for our cantilever beam in Figure 5 below.

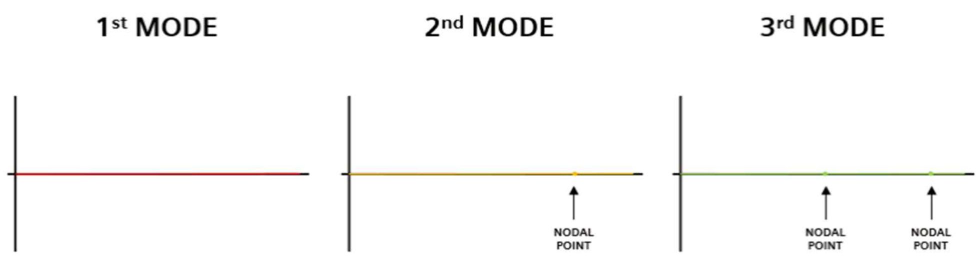


Figure 5: The first three mode shapes of a cantilever beam. The higher mode shapes feature points that do not move called nodes.

In the first mode, the entire beam participates in the deflection pattern, all points are moving. In the 2nd and 3rd mode however, there are points that do not move at all. These points that do not participate (ie- move) in the deflection shape are called nodes, or nodal points.

- Points that do not participate in a mode shape are called "nodes" or "nodal points".

Overlaying our cantilever beam with the first three mode shapes (Figure 6), it becomes clear that the Red impact location is on top of a node in the 3rd mode shape, and the Green impact location is at a node of the 2nd mode.

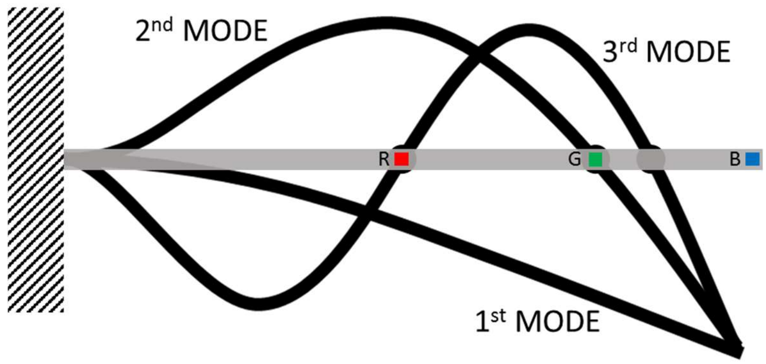


Figure 6: Nodal points in the 2nd and 3rd modes occur at Green and Red driving points respectively. These modes will not be excited by impacting at these locations!

Because nodal points do not participate in mode shapes, impacting our structure at the Red and Green locations will not adequately excite the mode shapes that have nodes at these locations. As a result, the peaks corresponding to the missing modes will not appear in the FRF.

This is the most important function of a driving point measurement: *A driving point FRF shows us which mode shapes are being adequately excited by impacting our structure at that particular location*. The modes that are excited by impacting at that location will create peaks in our driving point FRF, the modes that are not excited by that impact location will not.

- A driving point FRF shows which modes are being excited by impacting at that location.

Driving Point Survey

By making several driving point measurements at various locations around the structure and comparing the FRFs, we can tell which location will best excite the modes we are interested in. This is known as a "driving point survey."

Performing a driving point survey helps to avoid using a nodal point for excitation. For a realistic structure with 2 or 3 critical dimensions, this becomes even more important because the nodal "points" from the simple cantilever beam example turn into nodal "lines" as shown in Figure 7 below.

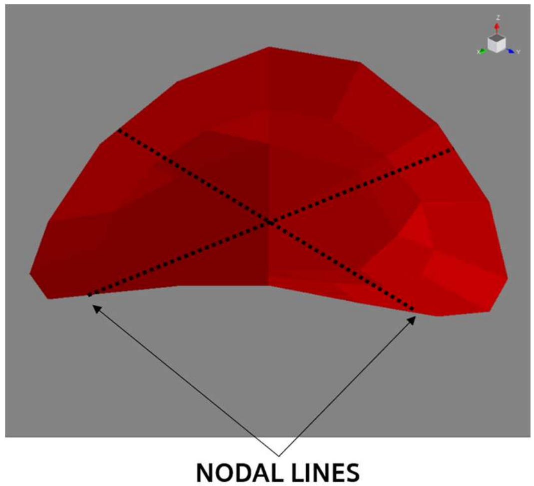


Figure 7: Mode of a circular disc featuring two nodal lines. Points along these lines do not participate in the mode shape.

However, by performing a driving point survey we gather the information needed to avoid nodal lines and ensure the modes of interest will be properly excited and measured.
Driving point measurements are important for other reasons as well. As we learned in this Modal Tips Article, in order to view mode shapes, it is necessary to measure either a complete row or complete column of our measurement matrix. The driving point measurements represent the diagonal of the test matrix and will always be part of this comprehensive modal survey data set. Driving point measurements also allow us to properly scale mode shapes, and calculate modal mass and stiffness for a structure. Unless we are only interested in resonant frequencies of a structure, a driving point measurement is always required.

Using Simcenter Testlab Impact Setup: Driving Points

After completing the setup of the trigger, bandwidth, and windowing using the steps in the Impact Setup workflow, the next step is the driving point survey.
One tip that makes the Driving Points process a little simpler is to move the Geometry tab in front of the Channel Setup tab in the Testlab workflow as shown in Figure 8.

Figure 8: Move Geometry ahead of Channel Setup tab to make the Driving Point survey easier.

Setting up our test structure in Geometry is a good thing to do before we begin acquiring data, as we can use it to name the input channels and ensure our data Point IDs match our geometry Point IDs. This is one of the reasons moving Geometry before Channel Setup can be helpful.
For this example, the test structure is a flat rectangular plate, made up of 15 nodes, as shown in Figure 9 below.

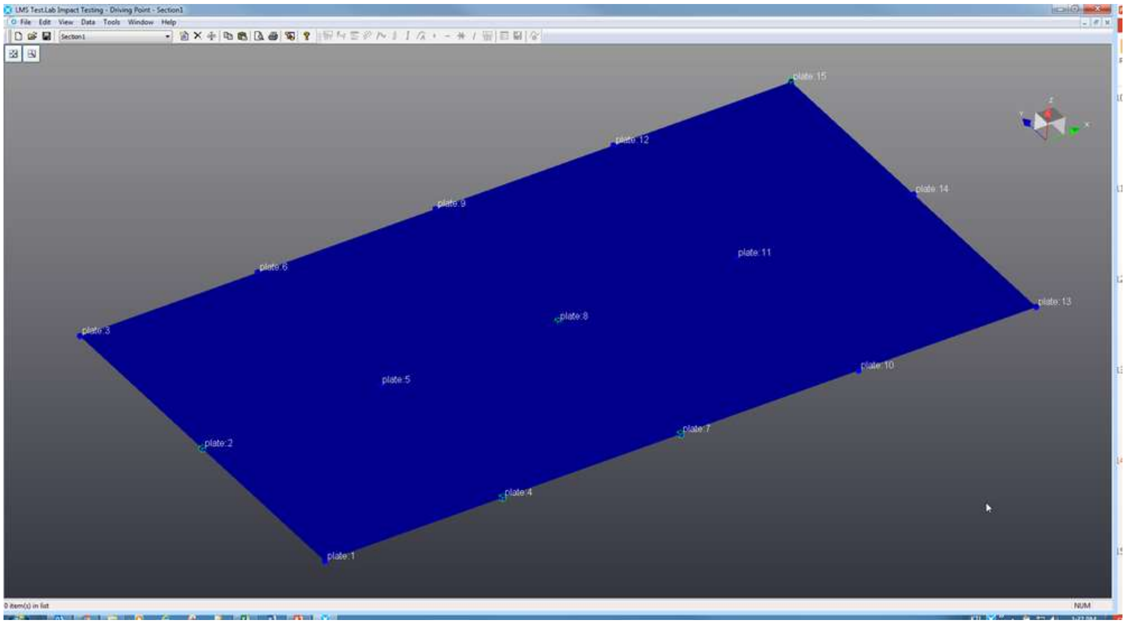


Figure 9: Test Geometry of a flat plate.

For this example, let's say we need to know the first 8 modes of this plate, their natural frequencies and mode shape description. Since I am unfamiliar with this plate, I select several points around the structure at which to make driving point measurements. The selected points are plate:2, plate:4, plate:7, plate:8, and plate:15, as shown below in Figure 10.

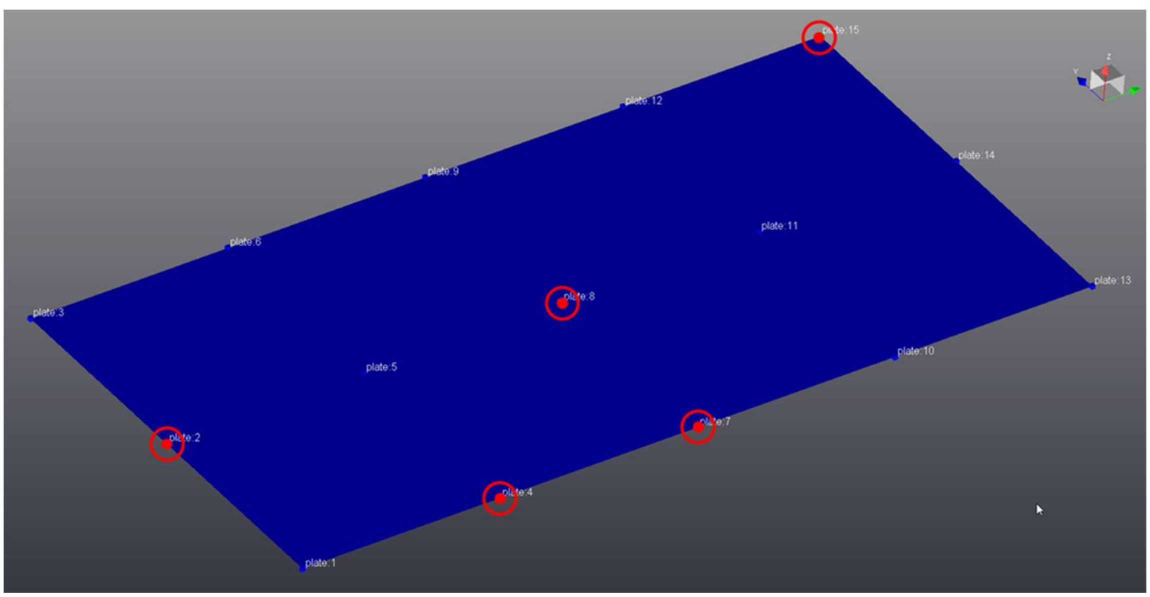


Figure 10: Point IDs selected to act as driving points.

Now that our test geometry is done, and our driving point locations are identified, we can move to the Channel Setup tab and begin the driving point survey. At this point we should have 2 channels turned on – a Force channel (Input 1) and an Acceleration channel (Input 2). Channel Setup is shown in Figure 11 below.

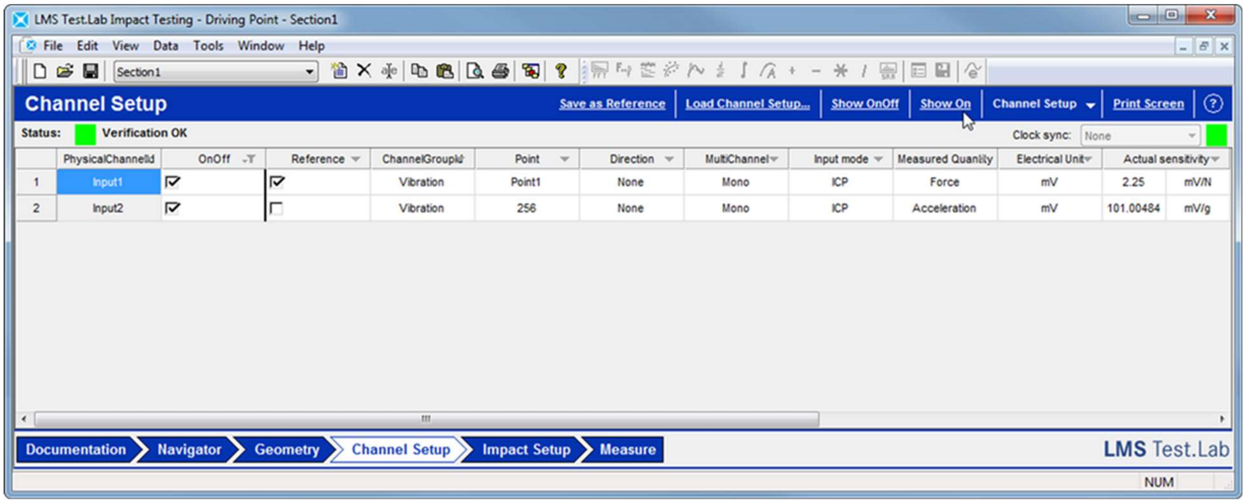


Figure 11: Channel Setup for our impact test.

Next, we will incorporate our geometry into Channel Setup by clicking on the down arrow next to "Channel Setup" in the blue bar near the top of the window as shown in Figure 12, and click on "Use Geometry".

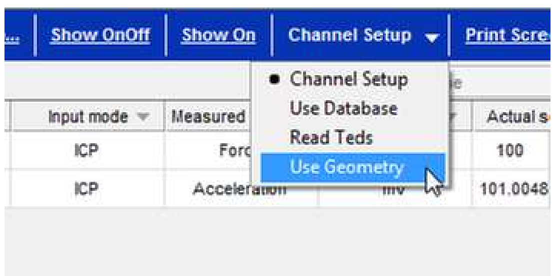


Figure 12: Select "Use Geometry" to incorporate Geometry Point IDs into Channel Setup.

This will open up a new window pane in Channel Setup for our geometry. Click on "Refresh" to show the structure (Figure 13).

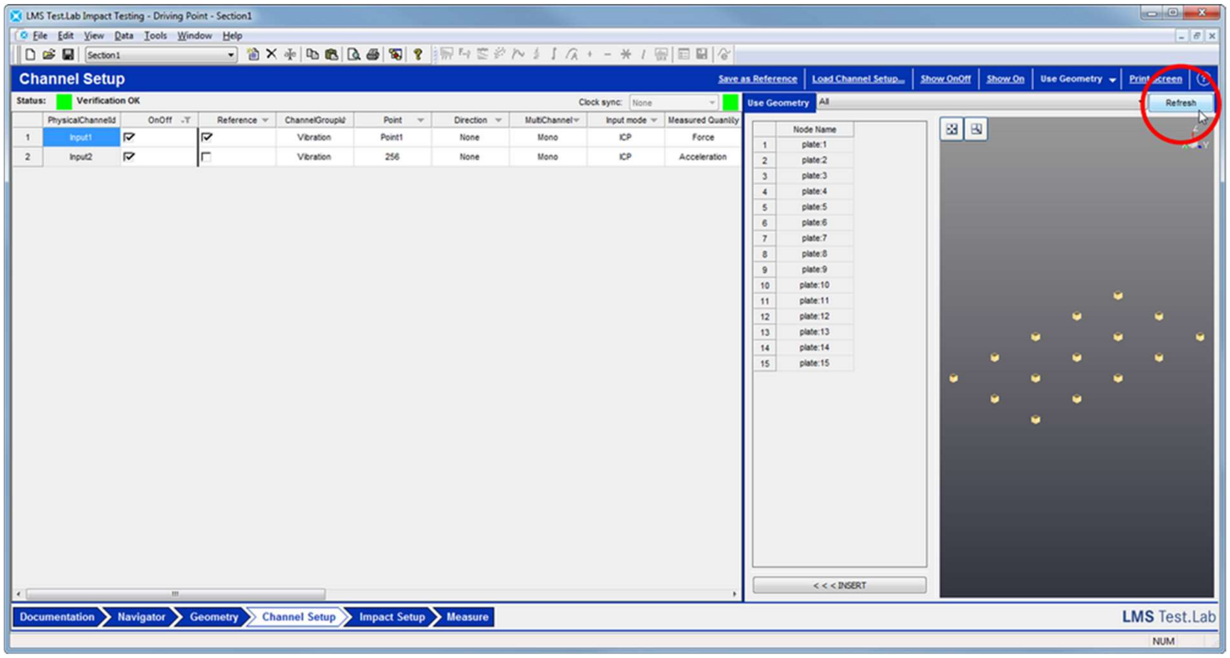


Figure 13: "Use Geometry" in Channel Setup to aid in naming input channels.

Our first driving point will be at the point named "plate:2" as shown in Figure 14 below. I can select this point in the geometry by clicking on the node icon or by highlighting it in the point ID list by clicking on the row header. Next, select the destination channels on the left half of the screen, again by clicking on the row headers for Input1 & Input 2 while holding the [SHIFT] key. Clicking "Insert" at the bottom of the screen will move the selected Point ID (plate:2) over to our two channel Point IDs (Figure 15).

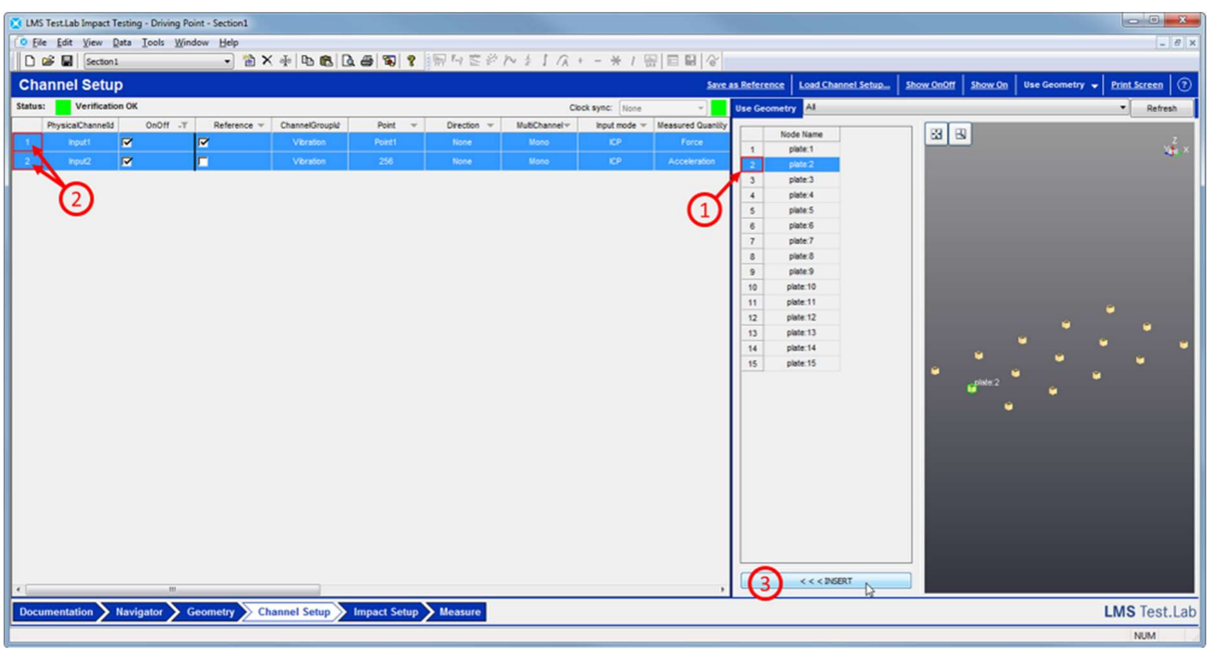


Figure 14: Select the first test point in the geometry and insert the ID on input channels.

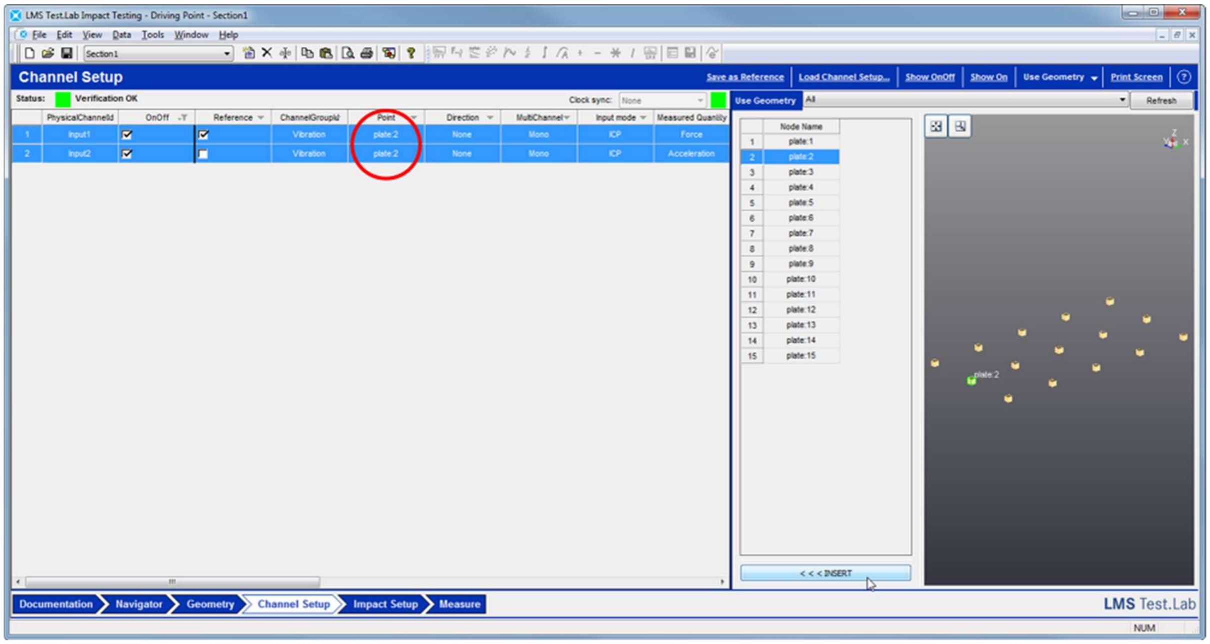


Figure 15: Channels 1 & 2 renamed to "plate:2" to match the geometry point ID!

With the measurement channels correctly labelled for the first driving point (plate:2), we can move to the Impact Setup tab in the Testlab workflow, and into the Driving Points page (Figure 16).

Figure 16: Driving Points is the last step in the Impact Setup worksheet.

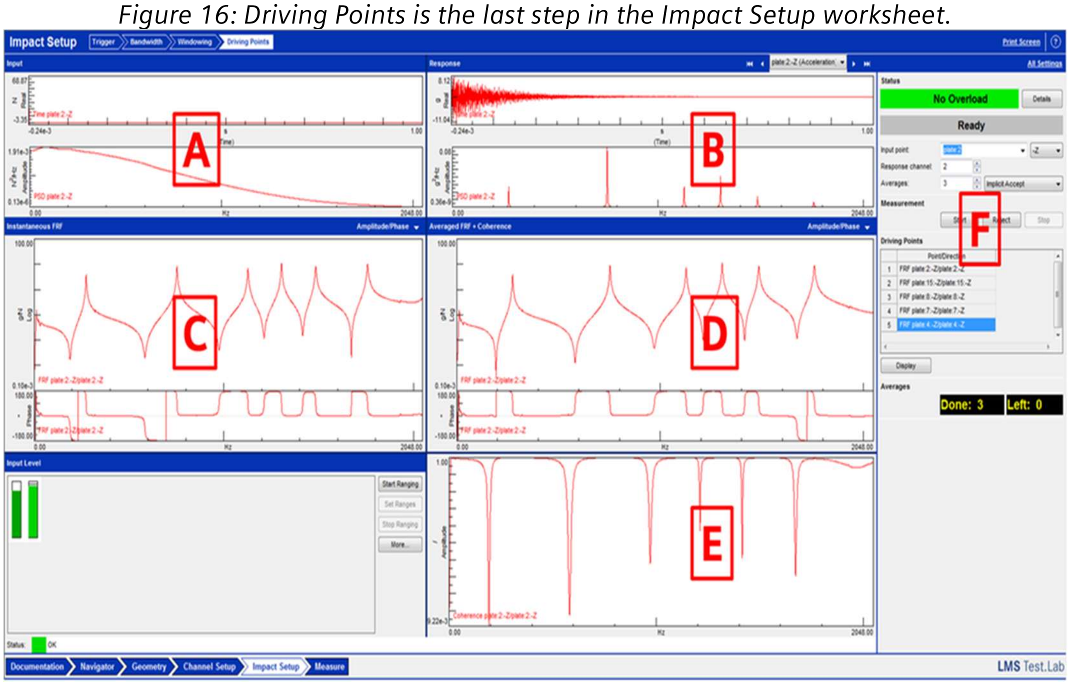


Figure 17: Various displays of the Driving Points worksheet in Impact Setup.

The Driving Points worksheet contains multiple displays, highlighting different aspects of our driving point measurement. These areas are listed below:

A: Time history (upper) and Autopower PSD (lower) of force channel

B: Time history (upper) and Autopower PSD (lower) of response channel

C: Instantaneous (current) driving point FRF Magnitude (upper) and Phase (lower)

D: Average of driving point FRFs Magnitude (upper) and Phase (lower)

E: Coherence

F: Control & Setup area for measurement

If we look in the area labeled "F" in Figure 17 we see that the "Input point" is already filled in as "plate:2". This is because we selected it in Channel Setup. The "Response channel" is set to "2" to indicate that our accelerometer is plugged into Channel 2 on our frontend.

We can select the number of averages we'd like to use for our driving point FRF, as well as whether we want to implicitly accept each average, or if we'd like to explicitly accept (via a popup window) after each average. These settings are shown in detail in Figure 18 below. Click on "Start" to acquire the driving point measurements for the first point.

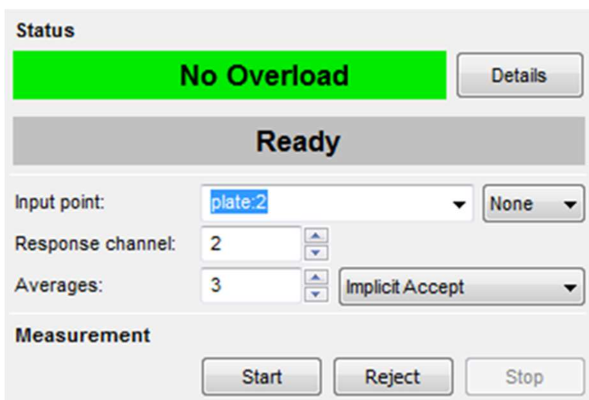


Figure 18: Input point will automatically read the Point ID from Channel Setup for the driving point measurement.

After we have performed the 3 averages, the driving point measurement name will appear in the "Driving Points" box in area "F", and we are ready to move the accelerometer to our next location, "plate:4". To update the channel information, we repeat the steps shown in Figure 14: first select the node in the geometry, then insert the name into channels 1 & 2. Once the channel setup info is updated, return to the Driving Points worksheet, and it will appear as shown in Figure 19.

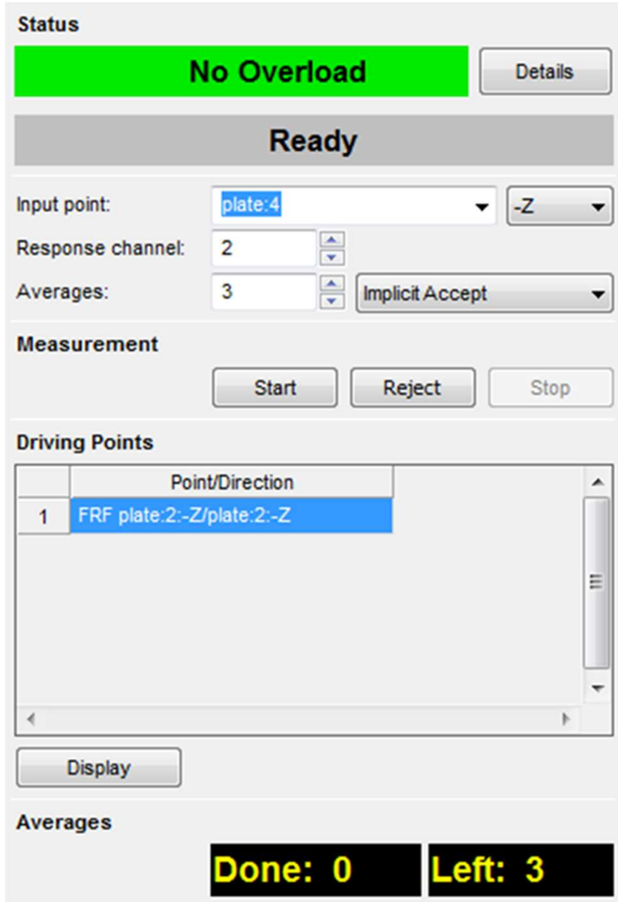


Figure 19: Use Channel Setup to update the naming of the next driving point measurement.

Repeat this process for the rest of the planned driving points. When finished there will be a total of 5 FRFs (Figure 20). By selecting them all and clicking on “Display” we will see all 5 curves overlaid (see Figure 21 below).

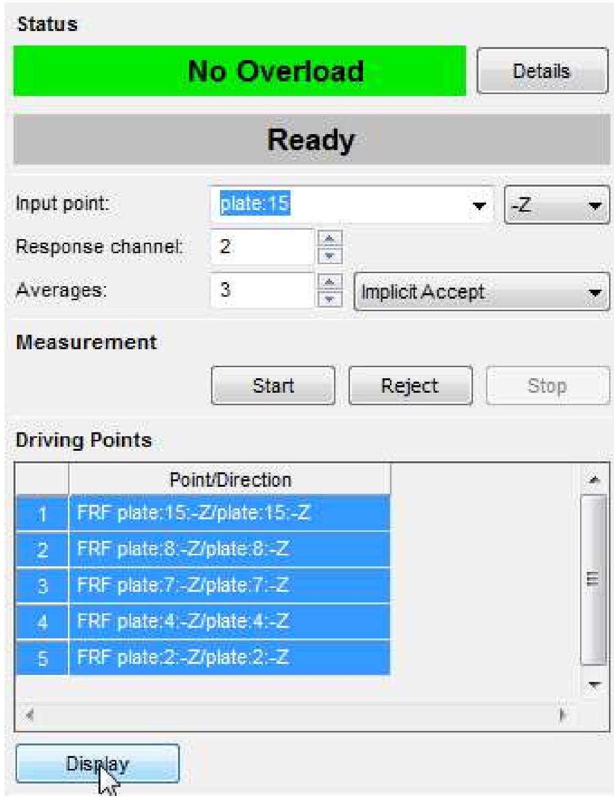


Figure 20: Select all driving point FRFs and click "Display" to compare the FRFs.

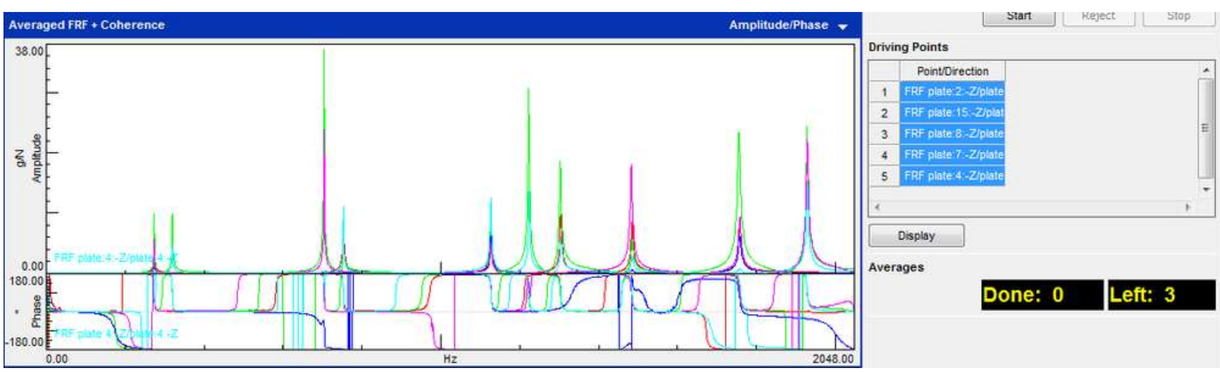


Figure 21: Driving point measurements overlaid

The driving point measurements will also appear in our Project, in a folder called "Driving Points", and can be plotted in Navigator. By comparing all five driving point FRFs we can quickly see some big differences between the excitation locations (Figure 22).

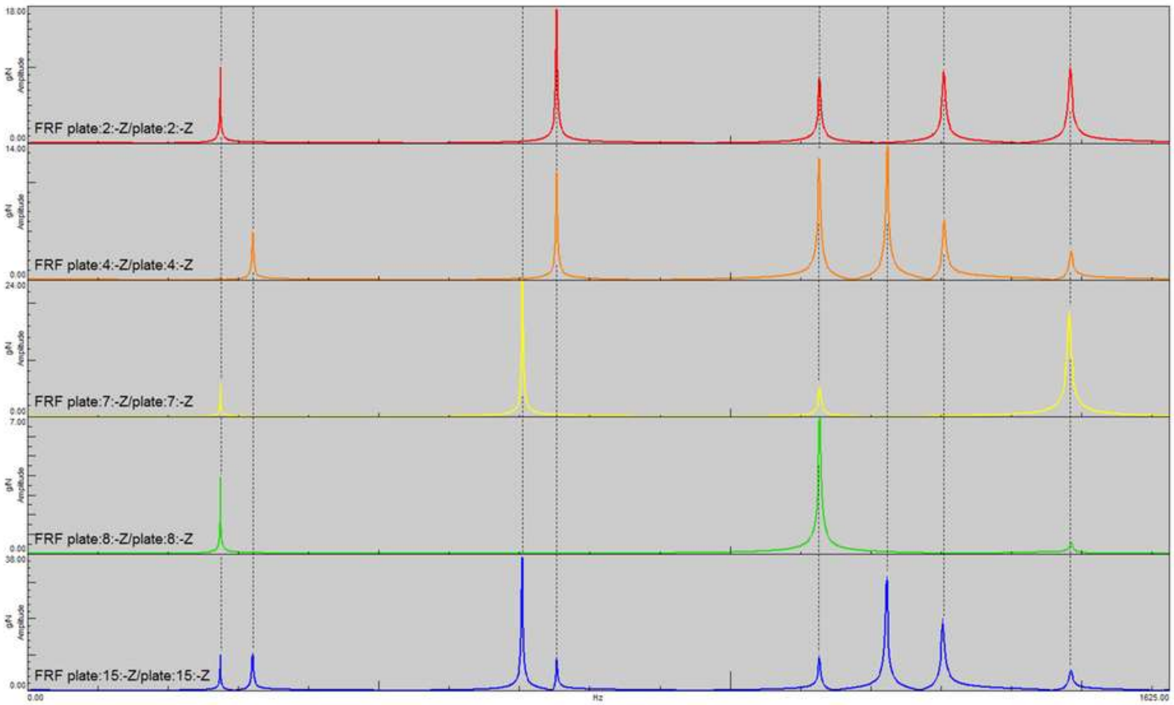


Figure 22: All 5 driving point FRFs in a multi-trace display. Some driving point locations did not excite all modes!

Each driving point location features a different number of peaks in the FRF. This indicates that only certain modes are being excited by impacting our structure at those locations. By making a table of the resonant peaks in each FRF, we can more easily see which driving point location will work best for this structure (Figure 23).

	plate:2	plate:4	plate:7	plate:8	plate:15	
Mode 1	274		274	274	274	bending
Mode 2		320			320	torsion
Mode 3			704		704	2nd torsion
Mode 4	753	753			753	2nd bending
Mode 5	1127	1127	1128	1128	1127	longitudinal bending
Mode 6		1224			1223	3rd torsion
Mode 7	1304	1305			1303	2nd longitudinal torsion
Mode 8	1484	1485	1483	1485	1485	3rd bending
# Missing	3	2	4	5	0	

Figure 23: Driving point location comparison. Only plate:15 excited all 8 modes of interest.

The red boxes in Figure 23 indicate that there is no peak at that frequency in the FRF (orange indicates weak excitation). Once we count up the modes and resonant frequencies found in each of the 5 driving point FRFs, we quickly see that only the FRF measured at plate:15 excited all 8 modes of interest. Every other impact location missed at least 2 modes we are interested in. By looking at the mode shape descriptions in Figure 23 for each of the modes, it becomes clearer why certain modes are not excited by impacts at certain locations. For example, consider the torsion mode, Mode #2 in Figure 23. Only points that are not on the two center lines of the plate will participate in this mode shape. In this case, only plate:4 & plate:15 are not on a center-line. (See Figure 24)

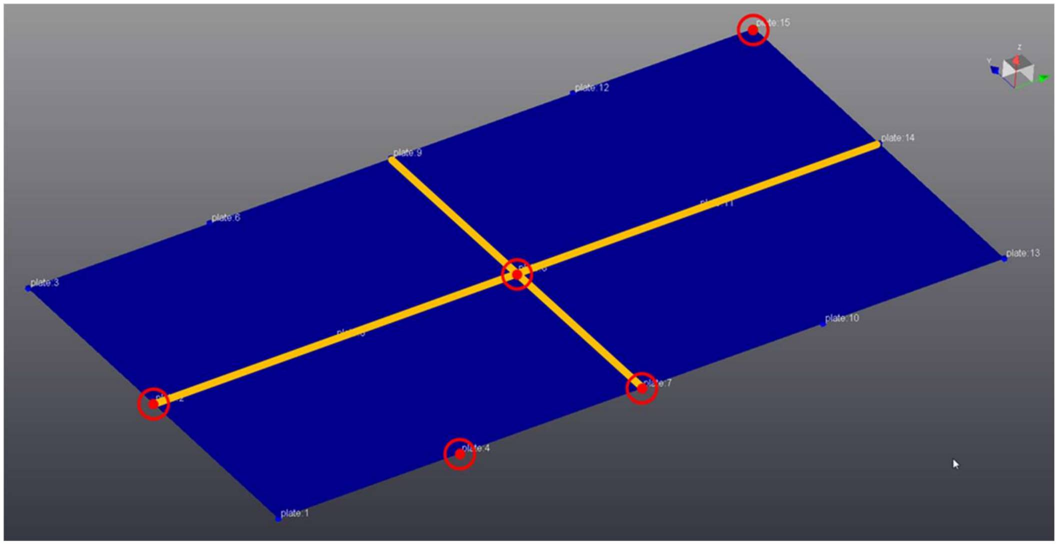


Figure 24: Points that are on a center-line of the plate will not participate in the torsion mode.

In general, it is best to avoid centerlines of our structure, focusing on corners and impact locations at the extreme ends and edges of the structure. This will typically excite the most modes and avoid nodal lines of the first few modes, which are generally of primary interest. However, on a new or unfamiliar structure, it is always in the best interest of the tester to perform a driving point survey. This will help to ensure that the structure is adequately excited, and the critical mode shapes of interest are measured.

Modal Assurance Criterion (MAC)

The Modal Assurance Criterion Analysis (MAC) analysis is used to determine the similarity of two mode shapes:

- If the mode shapes are identical (i.e., all points move the same) the MAC will have a value of one or 100% as show in *Figure 1*.
- If the mode shapes are very different, the MAC value will be close to zero, as shown in *Figure 2*.

If a mode shape was compared to itself, the Modal Assurance Criterion value should be one or 100%.

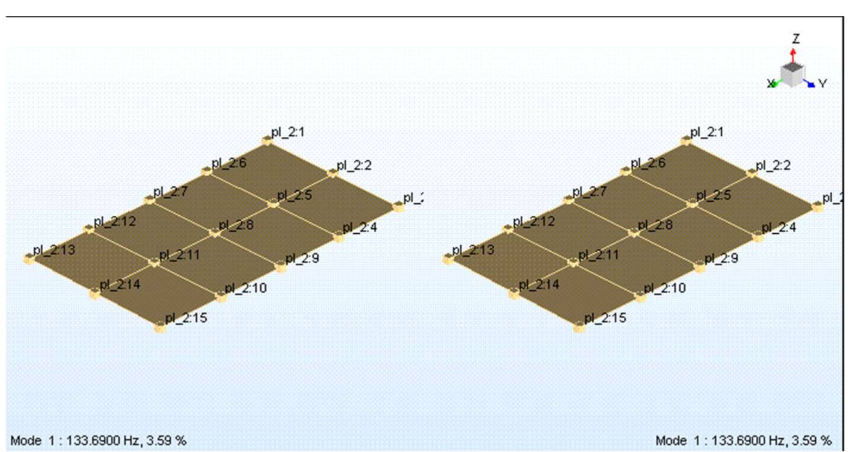


Figure 1: Mode shapes with 100% MAC value

For modes with different shapes, the MAC is less than 1. Shapes that are very different will have a value close to zero as shown in *Figure 2*.

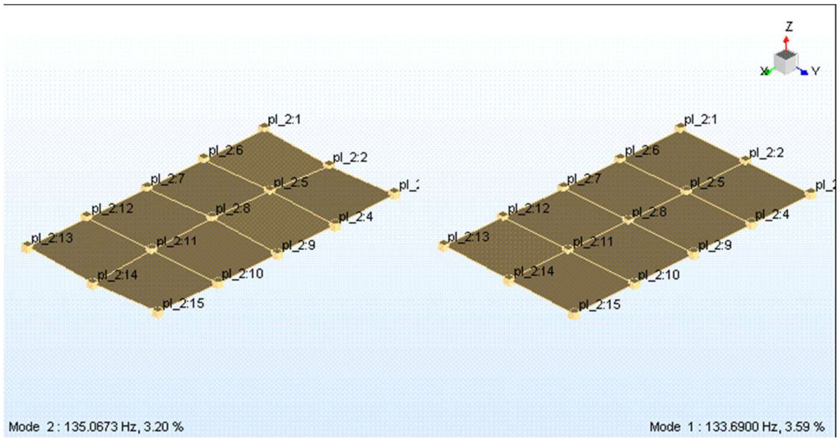


Figure 2: Mode shapes with 1.5% MAC value

Mode shapes that are used in the comparison can originate from a Finite Element Analysis or from an experimental modal analysis.
In a typical MAC analysis, one might make a 'MAC Matrix'. A 'MAC Matrix' is a series of bar graphs of MAC values, that each range from 0 to 100% as shown in *Figure 3*.

In the case of *Figure 3*, this is a mode set compared to itself. The mode set contains nine different individual modes, so 81 different MAC values are being calculated. About half the values are redundant –e.g., the MAC value between mode 1 and 3 is the same as between mode 3 and 1.

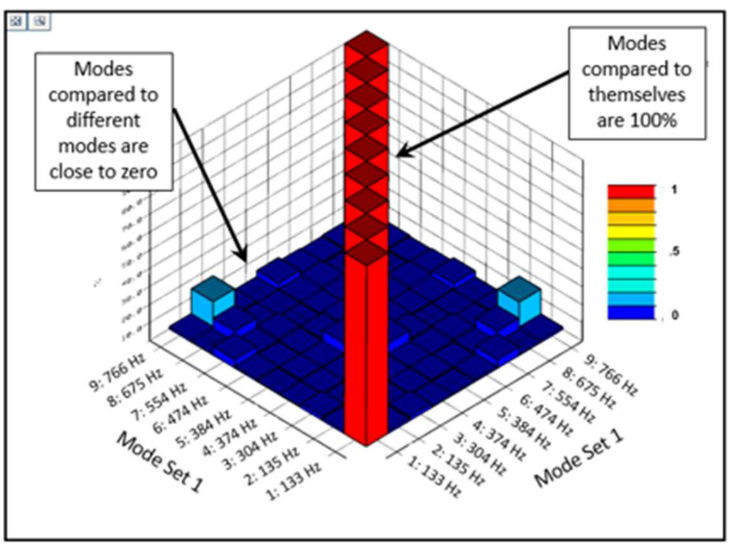


Figure 3: MAC matrix comparing a set of 9 modes. Red values are 100% MAC values. Dark blue is less than 10% MAC value.

In *Figure 3*, the first mode shape at 133 Hz is identical to itself, hence a single red bar of a value of 1. Along the diagonal, every mode is identical to itself, 1 to 1 (133 Hz), 2 to 2 (135 Hz), 3 to 3 (304 Hz), etc.

Off of the diagonal, the MAC values are very low. Ideally, each mode should be uniquely observed and have a different shape than the other modes. This is the case for this mode set. The highest off diagonal mode pair is mode 2 compared to mode 9 (and vice versa 9 to 2) with a MAC value of 20%. All the other off-diagonal mode pairs are below 20%.

Modal Assurance Criterion Equation

The MAC value between two modes is essentially the normalized dot product of the complex modal vector at each common nodes (i.e., points), as shown in *Equation 1*. It can also be thought of as the square of correlation between two modal vectors φr and φs.

$$MAC(\{\varphi_r\},\{\varphi_s\}) = \frac{|\{\varphi_r\}^{*t}\{\varphi_s\}|^2}{(\{\varphi_r\}^{*t}\{\varphi_r\})(\{\varphi_s\}^{*t}\{\varphi_s\})}$$

Equation 1: Modal Assurance Criterion equation for comparing two mode shapes

If a linear relationship exists (i.e., the vectors move the same way) between the two complex vectors, the MAC value will be near to one. If they are linearly independent, the MAC value will be small (near zero).
A complex vector simply includes both amplitude and phase, whereas a real vector is real part only. In *Equation 1*, it is also clear that the MAC is not sensitive to scaling, so if all mode shape components are multiplied with the same factor, the MAC will not be affected.

If an experimental modal analysis had 20 different nodes where measurements were made, the mode shape components at all 20 nodes are taken into account to calculate the MAC value, but more importance will be attributed to the higher amplitude node locations.

Use Cases

A Modal Assurance Criterion (or MAC) analysis can be used in several different ways:

- FEA-Test comparison – A MAC can be used to compare modes from an experimental modal analysis test to a Finite Element Analysis (FEA) and an object as shown in *Figure 4*. It will indicate if the same mode shapes are found in both the test and FEA analysis.
- FEA-FEA comparison – Several assumptions can be made in the creation of a FEA analysis: Young's Modulus, boundary conditions, and mass density values to name a few. A MAC analysis can determine the degree to which these assumptions affect the resulting mode shapes.
- Test-Test comparison – A MAC analysis can flag potential issues with the modal analysis results. Usually MAC will identify modes and areas that could benefit from acquiring more data points on the structure.

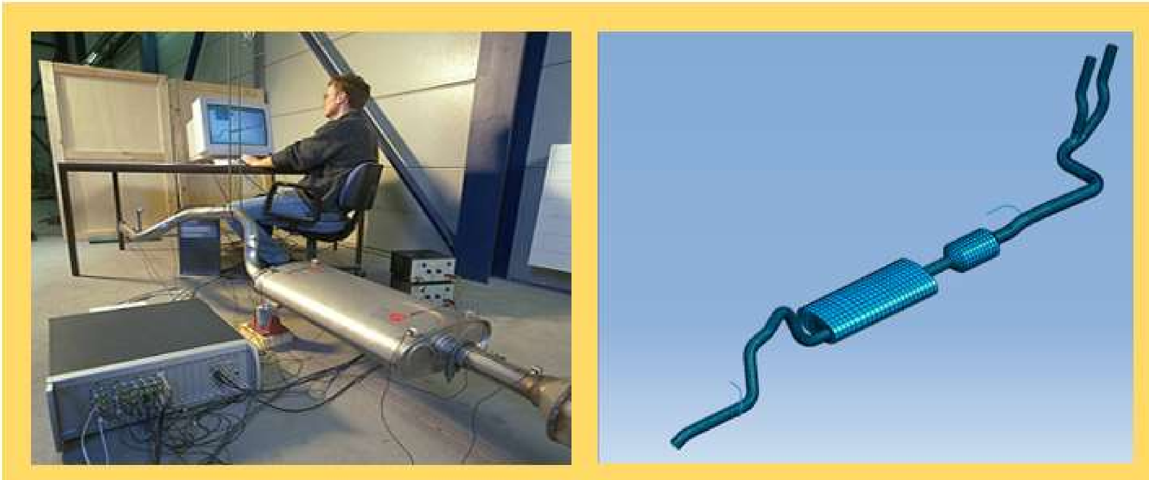


Figure 4: Experimental modal analysis (left) versus a finite element analysis (right) of an exhaust system

A MAC analysis is only looking at the mode shape, it does not compare the frequency value.

Experimental Modal Analysis Application Example

When performing an experimental modal analysis, the test operator must decide the number of points (i.e., nodes) to be measured. Determining the proper number is critical to the success of the test. If not enough points are measured, then the mode shape will not be identified properly.

Consider the case of an experimental modal analysis performed on a rectangular plate, suspended with free-free boundary conditions. Frequency Response Function (FRF) data was acquired at 6 locations on the plate. The FRF data was analyzed and a mode set extracted. After performing a MAC on the resulting mode set, not all the off-diagonal MAC values are close to zero (*Figure 5*).

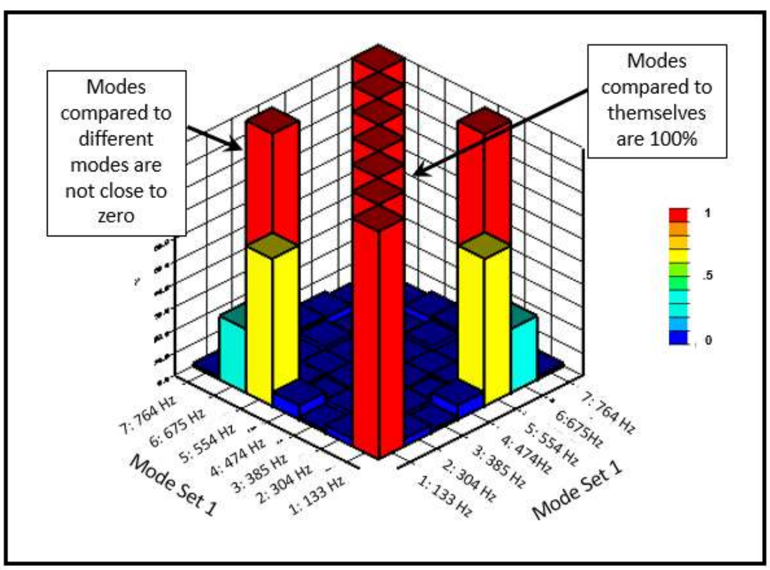


Figure 5: For a 6 point experimental modal analysis of a plate, the off-diagonal modes are not all close to zero. Modes 3 (385 Hz) compared to mode 7 (764 Hz) has a MAC value of over 90% which is unusual.

Upon closer inspection of modes 3 and 7, the shapes themselves are unexpected (*Figure 6*). They appear to be rigid body modes. Rigid body modes on a free-free suspended structure are normally around 0 Hz. There are 6 of them: translation in X, Y, Z and rotation in X, Y, and Z.

At frequencies like 385 Hz and 764 Hz, flexible modes of the structure are expected. In this case, only measuring at 6 locations on the structure leads to “spatial aliasing”. There are not enough points to capture the modes correctly.

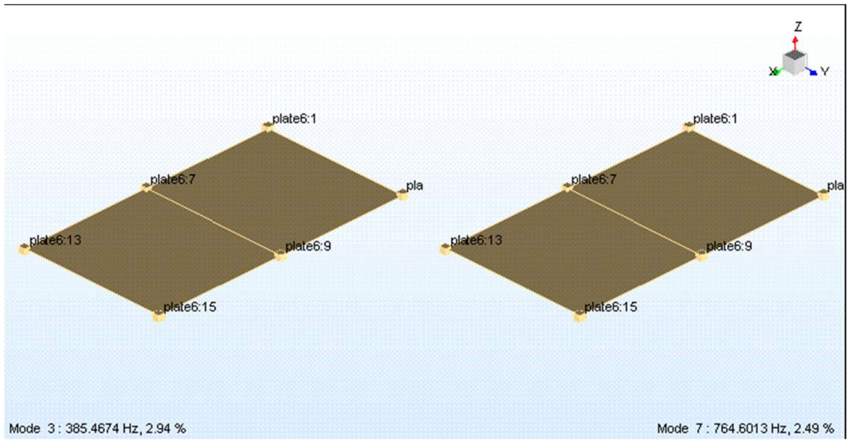


Figure 6: Both 385 Hz and 764 Hz appear to be rigid body modes, which is not expected.

Acquiring an additional 9 points leads to better results. With 15 total points, the mode shapes look completely different (*Figure 7*).

When viewing modes 3 and 7 again, but with 15 measured points, one can better appreciate how measuring only 6 points created the spatial aliasing error (*Figure 6 versus Figure 7*).

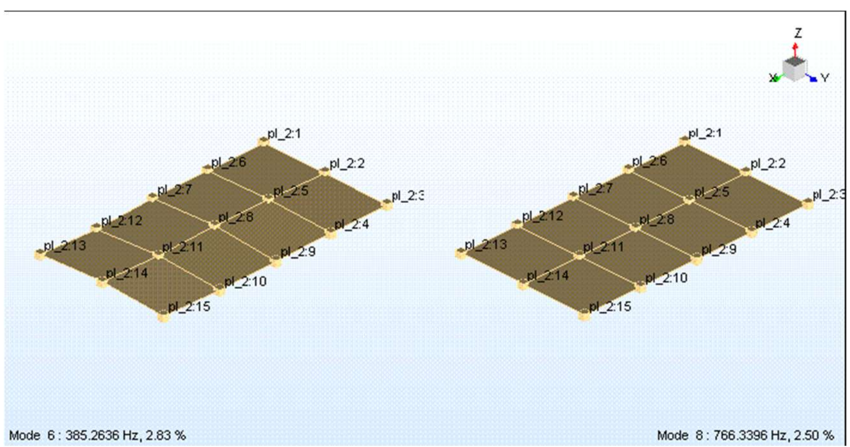


Figure 7: 15 point modal: 385 Hz on left, 766 Hz on right

When comparing *Figure 6* (6 point modal) versus *Figure 7* (15 point modal):

- The 385 Hz mode was a bending mode along the center of the plate as seen in the 15 Point modal analysis. Because points were only acquired on the edges in the 6 point modal, the entire bending was missed, resulting in a mode shape that appeared to be a vertical translation rigid body mode.
- The 766 Hz mode was actually a triple bending along the length of the plate as seen the 15 point modal analysis. The 6 point modal missed all the key bending areas, resulting in an apparent vertical translation rigid body mode.

The MAC analysis of the plate structure experimental modal analysis of the 15 point modal analysis is much improved as shown in *Figure 8*. The off-diagonal MAC values are much closer to zero.

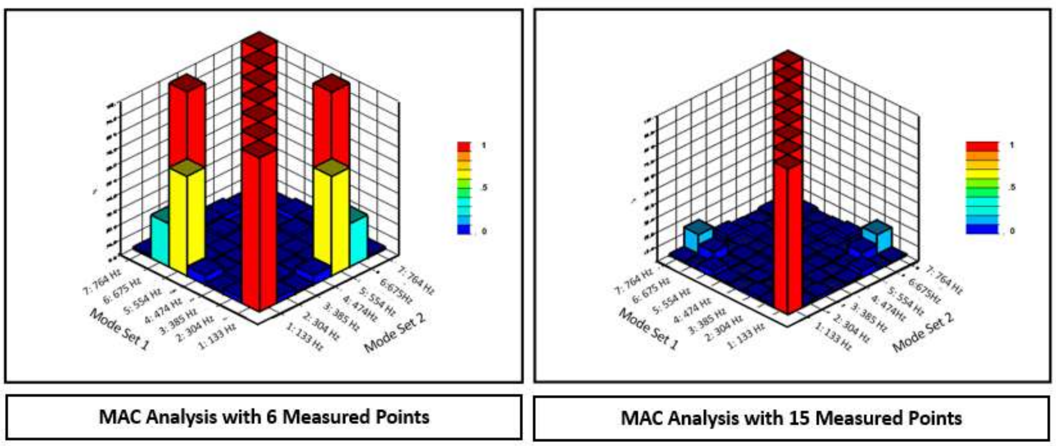


Figure 8: MAC analysis with 6 measured points (left) versus 15 measured points (right)

In experimental modal analysis, the data measured in the 6 point modal analysis is not "wrong". The FRF measurements at these nodes were no different in the 6 point modal versus the 15 point, since the physical structure being tested did not change. There were simply not enough measurement points to determine the complete mode shape. This is different than a Finite Element modal analysis were the number of nodes does determine the dynamic behavior.

In this case, a Modal Assurance Criterion (MAC) analysis flagged a problem with an inadequate set of measurement points. Because the off-diagonal MAC values in the MAC matrix were not low, the error was easy to find.

FEA-Test Application Example

An experimental modal analysis was done on an exhaust system and compared to a finite element modal analysis of the same exhaust (*Figure 9*).

Figure 9: Experimental modal analysis (left) versus a finite element analysis (right) of an exhaust system

After collecting Frequency Response Functions (FRFs) on the exhaust system, a MAC analysis was done between the first thirteen experimental test modes and the first thirteen finite element analysis modes. The results are shown in *Figure 10*.

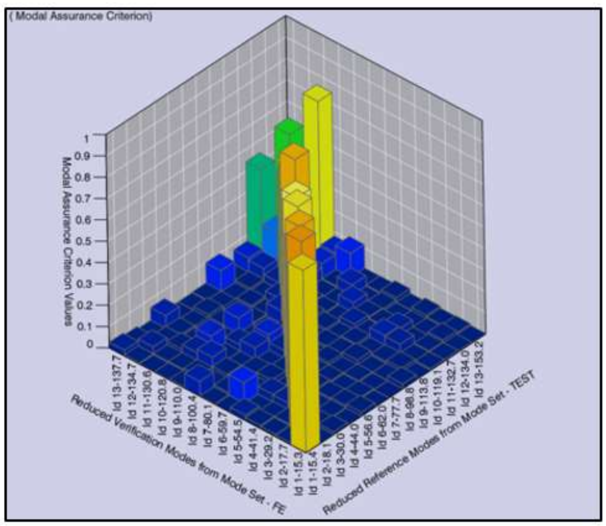


Figure 9: MAC Matrix comparing exhaust system finite element modes and experimental modes

Looking at the diagonal of the MAC matrix:

- MAC values are not 100%, because the two sets of modes are not identical.
- Modes 9 thru 13 are less than 75%.
- Modes 11 and 13 are not in the same order between the two mode sets

In this case, the MAC analysis indicates that there is room for improvement in the correlation of the test and FEA.

Using a variant of the MAC analysis called a 'MAC Contribution Analysis', the nodal points which most reduced the average MAC below 100% can be identified. Performing this analysis, it was found that nodal points of the Y pipe part of the exhaust were most responsible for the decrease.

Using this information, a visual inspection of the actual exhaust found that welds were present at the joint of the Y pipe (*Figure 10*). These welds were not physically represented in the finite element model, which was created from a CAD geometry. This can happen because when weld locations are only indicated on CAD drawings, but not physically present.

Figure 10: Welds at the Y pipe joint of the physical exhaust system (left) had to be introduced into the finite element model (right) to achieve better correlation.

After introducing appropriate elements at the Y joint, the finite element model was recalculated and compared to the test results (*Figure 11*).

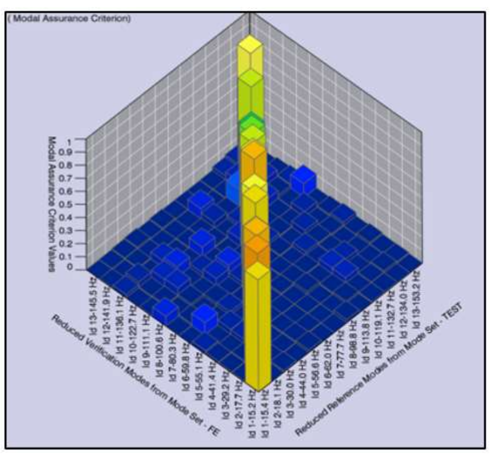


Figure 11: MAC analysis with updated FEA results versus test results

While not perfect, the following improvements were observed:

- Lowest MAC value along the diagonal improved from 69% to 80%
- Swapping of modes 11 and 13 was eliminated

Using the MAC and 'MAC Contribution Analysis' as guidance, the Finite Element model was significantly improved. This process can be continued until the results match within desired limits.

The most important part of this process is the actual "detective" work. Figuring out that welds need to be added to the FEA model (which are not in the base CAD parts) is an important lesson for future modelling projects. The MAC and 'MAC Contribution Analysis' are tools to be used in this process.

Calculating MAC in Simcenter Testlab

To calculate a Modal Assurance Criterion in Simcenter Testlab (formerly LMS Test.Lab), use the 'Modal Validation' worksheet of Simcenter Testlab Modal Analysis (*Figure 12*).

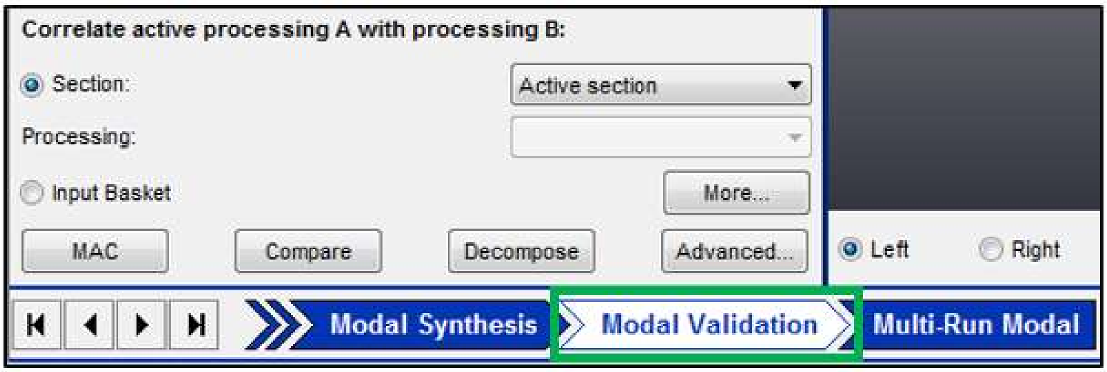


Figure 12: To calculate MAC analysis, open the 'Modal Validation' worksheet

A MAC analysis can be performed:

- 'MAC' - Between two different mode sets
- 'Auto - MAC' - A mode set to itself

Auto-MAC

To calculate a MAC analysis within a single mode set, select the "Processing" set that contains the modes in the upper left of the 'Modal Validation' worksheet (*Figure 13*).

Figure 13: To calculate MAC of one mode set to itself, use "Auto-MAC"

Then press the "Auto-MAC" button in the middle left of the screen. After pressing the "Auto-MAC" button, a table of MAC values is created in the upper right.

The default view of the table is called "Table/Geometry". Rather than looking at a table, a set of bars can be viewed by selecting "Matrix/Geometry" in the upper right (*Figure 14*).

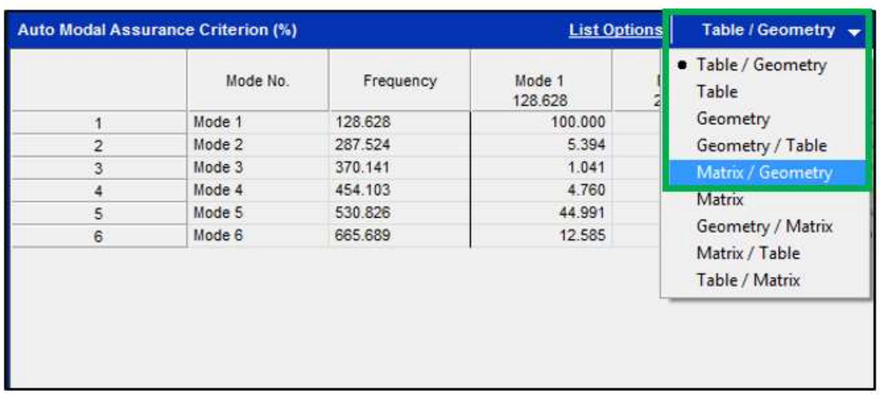


Figure 14: Switch the MAC table view from the default "Table/Geometry" to "Matrix/Geometry".

In the resulting Matrix view, one can click on the bars to see the MAC value for any mode pair. The corresponding mode shapes are automatically displayed below (*Figure 15*).

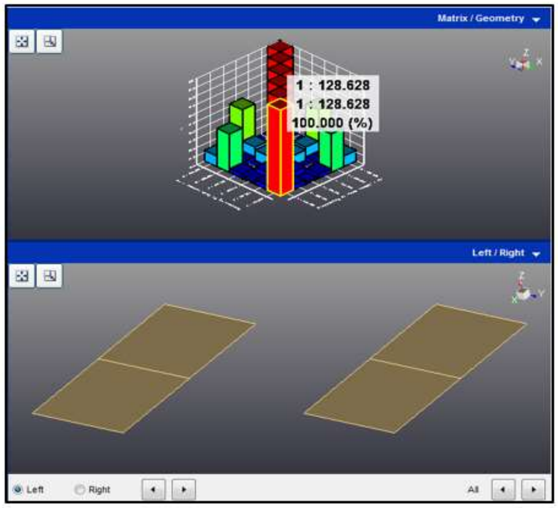


Figure 15: The MAC matrix view is a graphical representation of MAC values.MAC Comparison

A MAC analysis can also be performed between two different modes sets. This is done by selecting two different "Processing" sets (*Figure 16*).

Figure 16: For comparison between two separate mode sets, used the "MAC" button after selecting the appropriate mode sets.

To do a comparison between two different mode sets, each needs to be identified in a separate location on the left side of the 'Modal Validation' worksheet:

- First Mode Set – Select at the top via the "Active Processing" dropdown.
- Second Mode Set – Select at the bottom with either the "Section" toggle or "Input Basket" toggle.

Press the "MAC" button to perform the analysis.

Conclusion

A Modal Assurance Criterion (or MAC) analysis can be used for FEA-Test, FEA-FEA and Test-Test comparisons of modes. By analyzing a MAC matrix, an engineer can improve the quality of an experimental modal test, verify finite element models, and update FEA models with test data.

Using Pseudo-Random for High Quality FRF Measurements

Modal Testing

In structural testing, especially for Modal Testing, the quality of the Frequency Response Function (FRF) is very important. For correlation with an FE model, a good test modal model starts with a good, high fidelity FRF. The quality of the FRF is indicated in the Coherence function.

Typical Signals used for excitation

The types of excitation signals vary quite a bit, but are classified as either a random-stochastic type of signal, or one of sinusoidal nature. There are tradeoffs for both, and how they may be applied, but the signal-to-noise ratio (SNR) of the sinusoidal types is usually much better, resulting in a better FRF.

Avoid leakage in the Fourier Transform

A fundamental requirement for an accurate Fourier Transform is that the measured data sample is periodic within the acquisition window. Some signals are naturally periodic, like a burst-random, and some need to have a window to force the periodicity, which will always introduce some distortion in the measured data. For a sinusoid that lines up on the Fourier grid, this is a naturally periodic signal, and with high SNR.

Sine

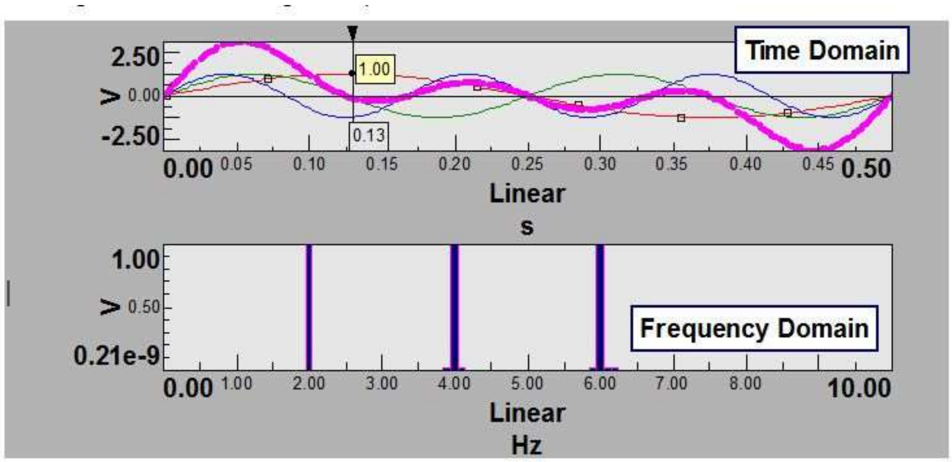
- Naturally periodic at spectral lines
- Concentrate energy in spectral bands
- High SNR

Burst Random

- Naturally periodic
- Broad energy across frequencies
- Low SNR

Multi-sines

Taking a number of these periodic sinusoids and adding them together are called “multi-sines”. Take for example a signal at 2, 4, and 6 Hertz, add them together, you end up with a signal as shown in the upper display below. With an acquisition window of .5 seconds, you end up with a frequency resolution on the Fourier grid of 2Hz, and so all signals are periodic.



The individual 1 Volt sinusoids that are aligned on the Fourier grid are naturally periodic in the time domain and the amplitude can be accurately calculated with the Fourier Transform to the frequency domain.

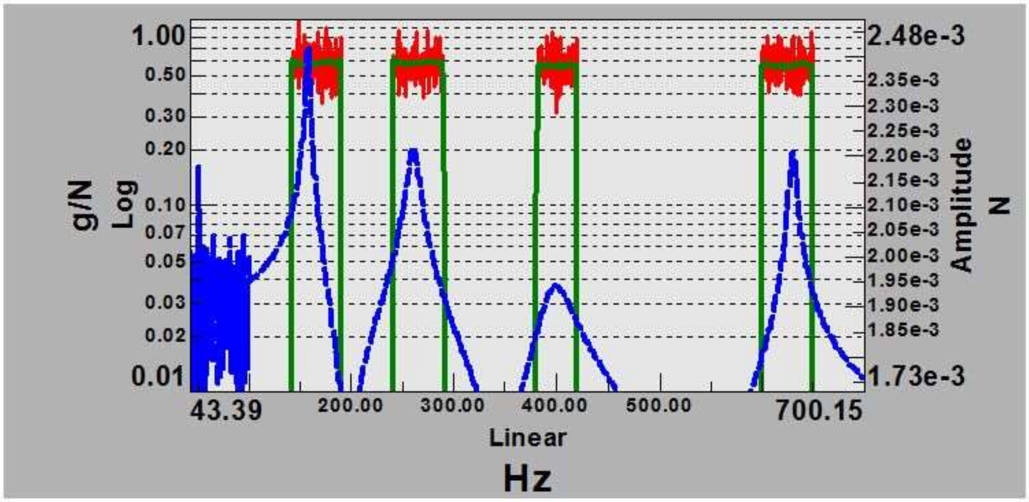
The picture below shows the difference of a Pseudo-random vs Broadband random and how they are represented in the frequency domain.

Different forms of “Multi-sines”

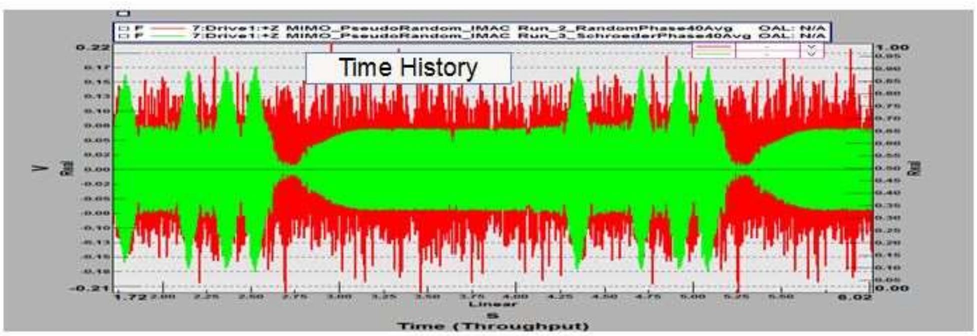
There are many combinations of “Multi-sines”, 2 of which are used for FRF excitation signals.

1. Periodic Random - Random Amplitude/Random Phase
2. Pseudo Random - Fixed Amplitude/Random Phase with a Constant or Shaped Spectrum to Concentrate energy around the resonances

With the Pseudo-random signal, there is control on the amplitude to define the sinusoids used to excite the structure. The amplitudes can be shaped to correspond with higher amplitudes where the structural resonance occurs. This puts more of the shaker excitation energy where it is most needed. By manipulating the phases of the Pseudo-random signal with “Schroder” sines, this can reduce the crest factor, (the peak to RMS ratio) of the output voltage to the amplifier/shaker combination. This will keep the time and spectral excitation signals closer to the reference spectrum. The benefit of this can be seen in the green trace of the display. In this case you can raise the excitation level higher, but still avoiding overloads, which can ruin a measurement.



The blue trace is the FRF which indicates where the signal should be concentrated. The frequency domain version of the standard Pseudo-random signal (Red trace) and the Schroder version of the Pseudo-random signal (Green trace).



The display shows the time domain version of the random-phase version (standard) of the Pseudo-random signal (Red trace), and one that has been applied with Schroder phase (Green trace). There is a 50% reduction in peak amplitudes.

Spectral averaging

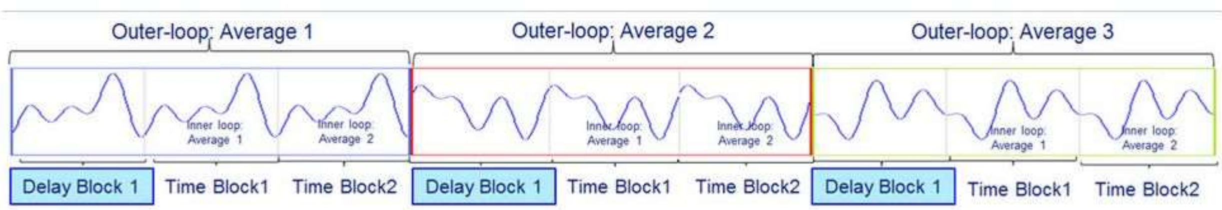
To measure a good representation of the signal, then a certain number of averages are required. The signal-to-noise has a big part in how many are required, and since sinusoidal signals have a better SNR, they will generally require less number of averages, saving testing time. For random data averaging in the frequency domain, there is a choice of linear or exponential averaging, whether using random with windowing, or more commonly using burst random.

Spectral Averaging can be either linear averaging, where each average is weighted equally, or exponential averaging where the later averages are weighted higher.

Cyclic Averaging

A time domain version of averaging is called cyclic averaging. There are a couple of cases where this is done.

- Case 1: Semi-stationary conditions
  - "Adaptive resampling"
  - Sampling based on RPM, also called Order Tracking
- Case 2: Stationary conditions
  - Synchronous sampling - Time average starts at beginning of time block or event

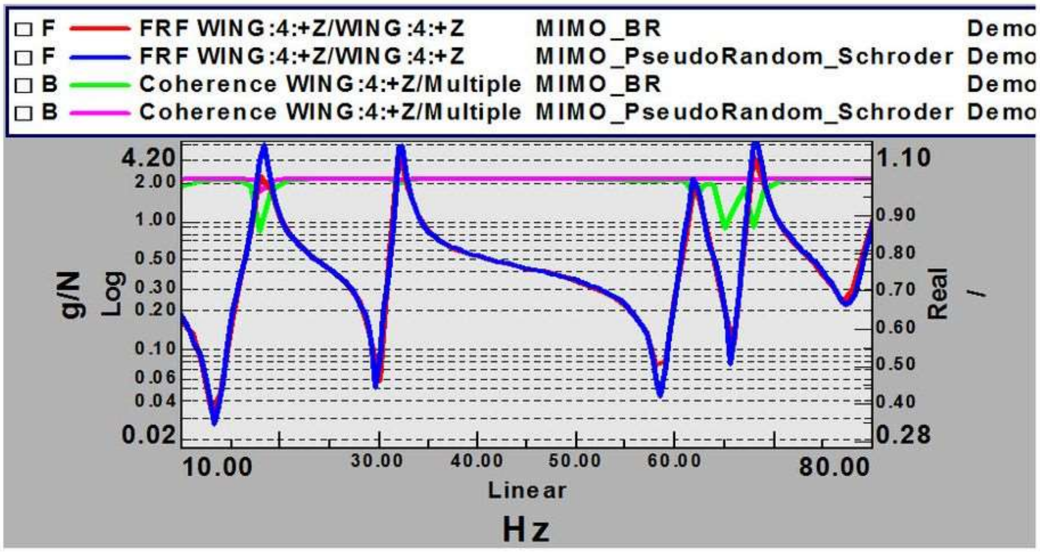


Cyclic Averaging

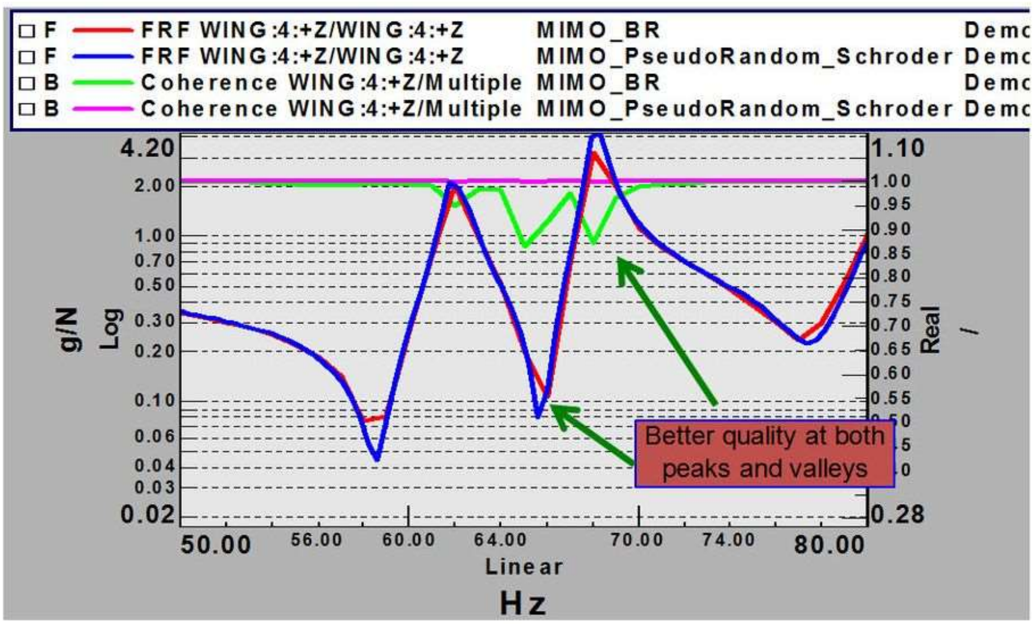
Cyclic averaging can be done when there is a repeated time signal exciting the structure, and the averaging is done "cyclically" in time, corresponding with the period of excitation. After some "delay blocks" have been sent, the transients have a chance to decay, and the structure has some time to respond periodically to that signal. The averaging is started after a number of these "Delay blocks", and then repeated at the same time after each excitation signal. After a number of these so-called "inner-loop" averages, then a number of "outer-loop" averages are taken.

Case Studies

To show the benefit of this Pseudo-random technique on a real structure, the following data is shown for a set of "Driving-point" FRF's made on an airplane. The following graphs use the Coherence function as a measure of the FRF quality. A value of 1.0 is a perfect coherence, and hence good quality FRF. You can see there are some lower dips in the coherence for the Burst Random case compared with the Pseudo-random with Schroder Sines case.



There are dips in the coherence function at some resonance peaks. This is exactly where an accurate measurement is the most important.



This plot is a zoomed version of the previous plot. This shows that the peak values of the FRF are better with Pseudo-random. It also shows anti-resonance values are better quality too, which is important for matrix inversion.

Psuedo-Random – Summary

- Fixed Amplitude/Random Phase
- Schroder Optimization to minimize Crest Factor → Higher SNR, and reduce the occurence of overloads
- Intrinsically Periodic →NO Leakage, NO Windows
- Very light transient effects if one or more delay blocks are used for cyclic averaging
- *Should give the best FRF Estimate (See Ref.)*

Operational Deflection Shape (ODS)

Operational Deflection Shapes (or ODS) analysis gives additional insight into noise or vibration problems that individual measurements alone do not.

An operational deflection shape is an animation of the vibration pattern in a structure. Both the amplitude and phase of vibration measurements are animated.

Figure 1 is an operational deflection shape of a vibration issue in a truck that occurs at cruising speeds. The vibration is felt in the steering wheel and seat by the truck occupant.

The vibration is measured at several *different* points or locations on the structure using accelerometers. In *Figure 1*, each blue cube represents a location where an accelerometer was used to measure vibration on a pickup truck.

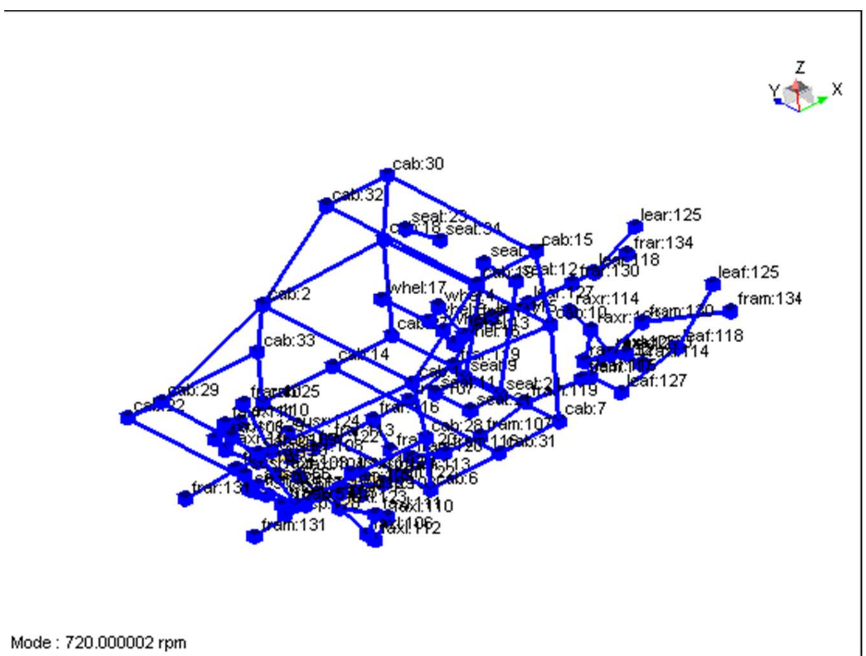


Figure 1: Operational deflection shape of pickup truck shows axle mode is cause of vibration problem

In the animation of *Figure 1*, an axle resonance excited by the rotation of the tires is the root cause of the unwanted vibration in the pickup truck. This is obvious by looking at the animation. Viewing vibration measurements (1st order wheel vibration) as shown in Figure 2 does not readily lead to the same conclusion.

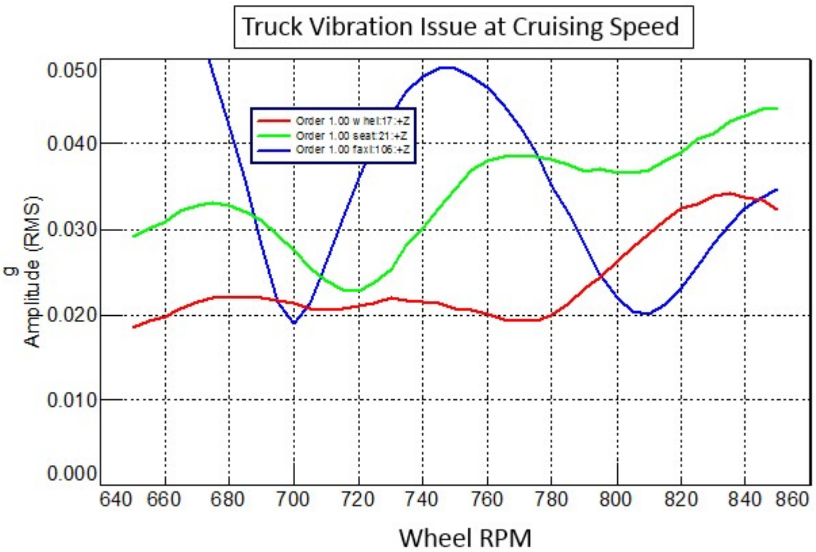


Figure 2: Vibration measurements do not show the axle tramp mode creating the vibration issue

For example, the fact that the two wheels of the axle are out of phase (called “axle tramp mode”) cannot be determined from these two dimensional (2D) plots.

Previous to performing the operational deflection shape, the manufacturer had tried to balance the tires in hopes of eliminating the vibration. The operational deflection shape animation clearly shows why balancing would not be the most effective strategy in reducing the vibration experienced by the driver.

When it comes to diagnosing a vibration issue, the old adage “a picture is worth a thousand words” could be rewritten as “an operational deflection shape is worth viewing a thousand individual measurements”!

What can be animated? What is required?

To perform an operational deflection shape analysis, three steps are required:

- Geometry: Create a geometry of the test object
- Measurement: Acquire data with consistent phasing
- Analysis: Create animation utilizing the geometry and measurement data

Any type of measurement (orders, spectrums, time) can be animated. The key is that the phase relationship between all the channels is preserved during the measurement. To preserve the phase properly, the measurements can be performed via two methods:

- *Measure all channels simultaneously* – Good for animating all types of measurement data, including time data. May required a high number of channels.
- *Phase reference channel* – Allows multiple low channel count measurements to be pieced together for a complete animation. A single reference accelerometer is kept in a fixed

location during each measurement rove. This technique is only useful for frequency/order domain measurements, not time domain.

Simcenter Testlab Operational Deflection Shapes

Here are instructions for performing an Operational Deflection Shape analysis in Simcenter Testlab

Geometry

Based on key vibration locations to be a measured, a geometrical representation of the test object should be created. The geometrical representation consists of test nodes, and connections between nodes, as shown in *Figure 3*.

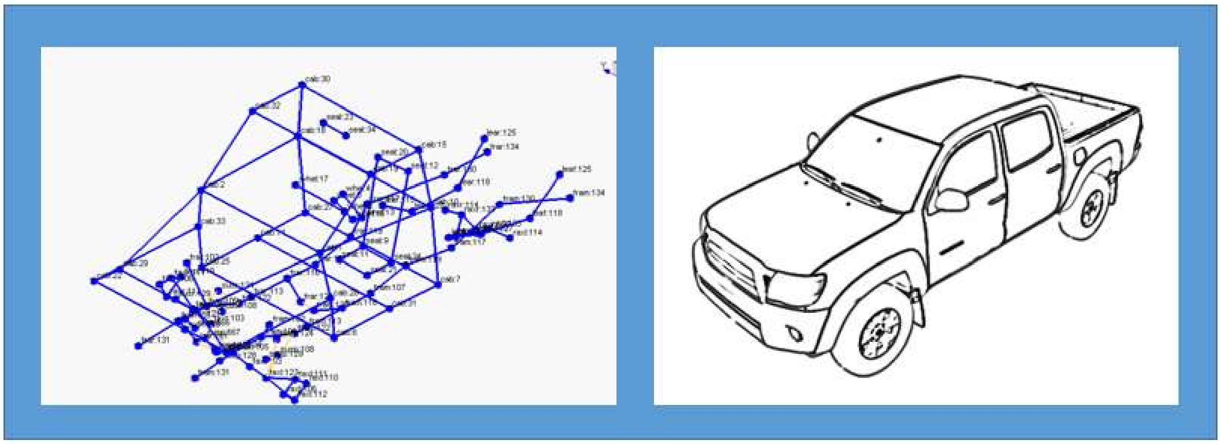


Figure 3: Left – Geometrical representation of test nodes, Right – Drawing of test object

It is also possible to import a CAD model as the basis for the geometry, rather than create one from scratch.

When creating a geometry in Simcenter Testlab, the following will need to be defined:

- *Components* – Test objects can have different components. For example, a truck could have components corresponding to body, wheels, seats, hood, roof, etc.
- *Measurement nodes or locations* – Nodes are the exact locations where accelerometers are placed on the test object.
- *Connections* – User defines connections between nodes. These are for visual interpretation only, and do not create any physical restraints that modify the physically measured motion.

To make a geometry, select "Tools -> Add-ins" from the main menu as shown in *Figure 4*.

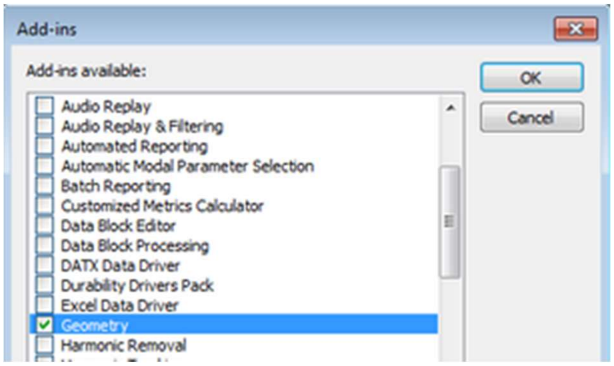


Figure 4: Tools -> Add-ins -> Geometry

This creates a new worksheet called "Geometry" as shown in *Figure 5*. The geometry add-in is only required to create and build a geometry. Once the geometry is made, the add-in may be turned off to conserve tokens, even when performing an operational deflection shape analysis.

Figure 5: Geometry Worksheet is added to workflow at bottom of screen

Click on the 'Geometry' worksheet. Across the top of the 'Geometry' worksheet are sub-worksheets as shown in *Figure 6*.

Moving thru the sub-worksheets from left to right goes through the steps needed to build a geometry. The sub-worksheets are in the order needed to create a geometry: Components, Nodes, and Lines.

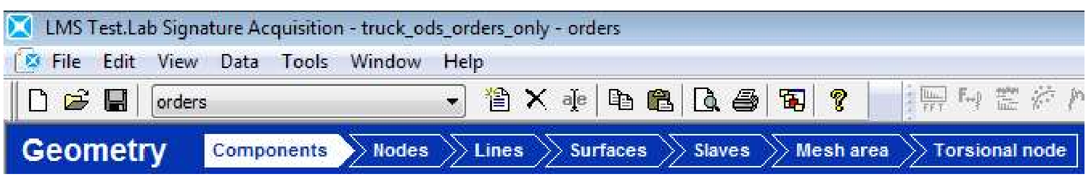


Figure 6: Geometry sub-worksheets build a geometry be moving left to right

In the first sub-worksheet called 'Components', enter the component names desired for the test object as shown in *Figure 7*. A single test geometry can consist of different components, for the example of the truck, the components include body, rails, axle, seat, steering wheel, etc.

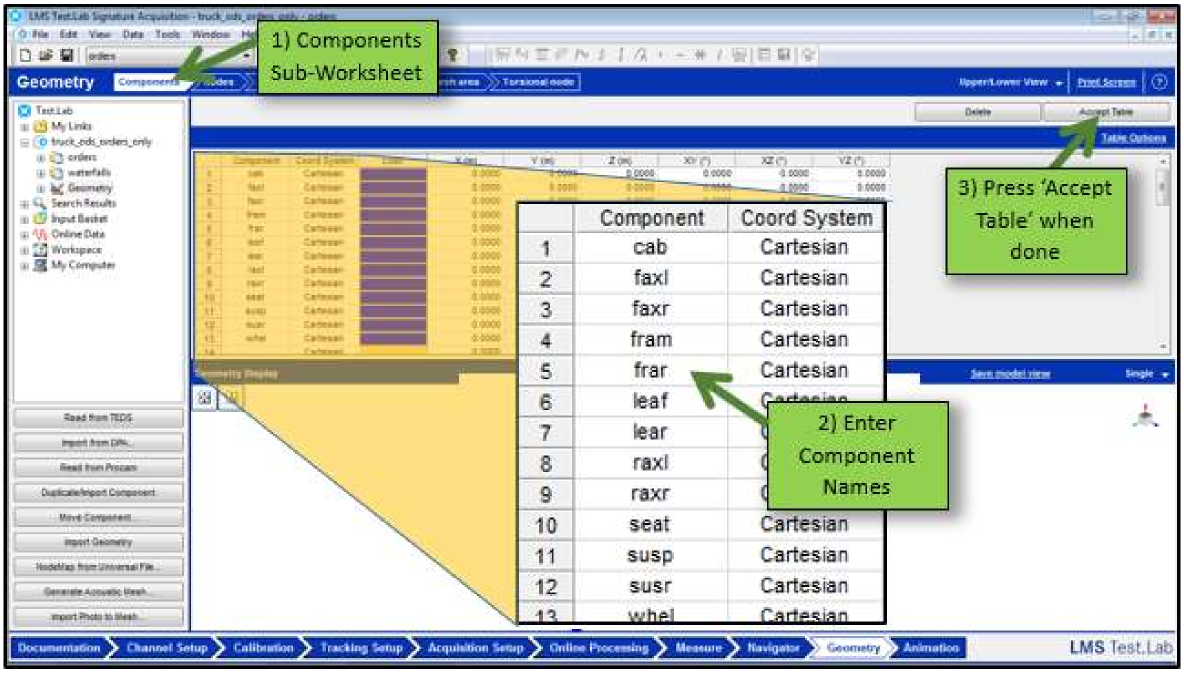


Figure 7: Component sub-worksheet in Geometry

The steps for creating components are:

1. Click on 'Components' sub worksheet.
2. Type the desired component names in the Component column. Each component can be assigned unique colors or co-ordinate systems.
3. Press 'Accept Table' in the upper right when finished.

Now select the next sub-worksheet called 'Nodes' to add measurement points to the components as shown in *Figure 8*. Each node corresponds to a point or location on the structure where an accelerometer will be mounted to measure vibration.

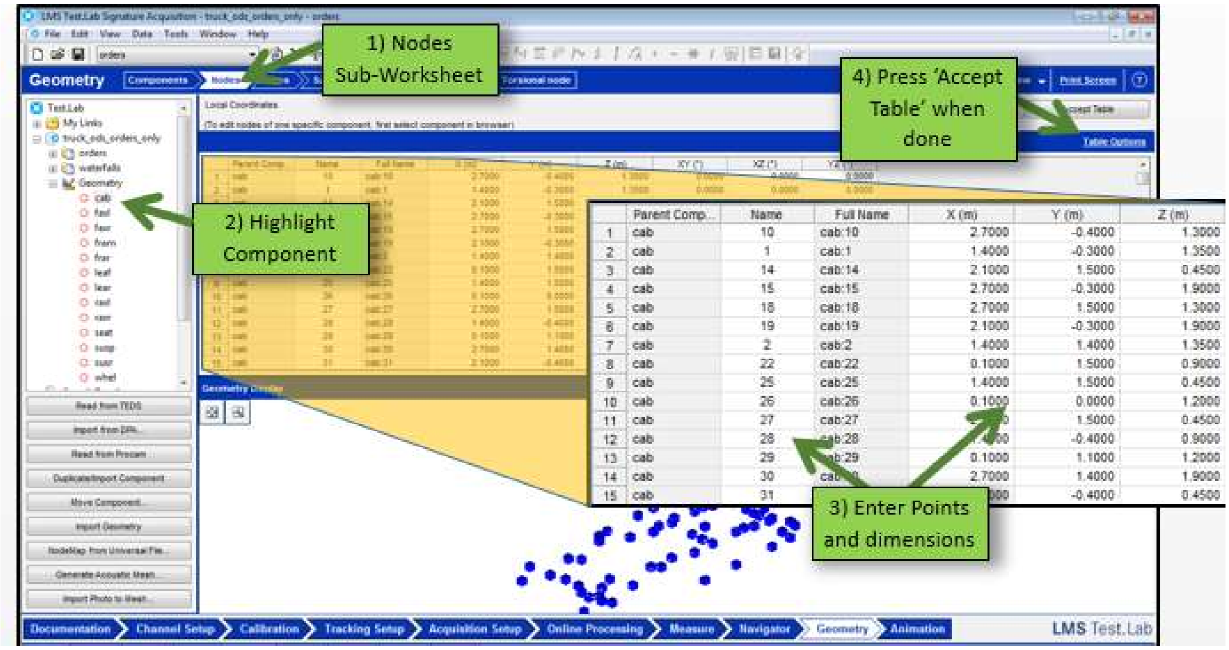


Figure 8: Nodes sub-worksheet in Geometry

To create nodes on the components:

1. Select the 'Nodes' sub-worksheet
2. On the left side, highlight the component to add nodes (ie, accelerometer measurement locations). This can be repeated for each component as needed.
3. After highlighting the desired component, enter the point number under the 'Name' column and X, Y, and Z dimensions (expected in meters by default). These dimensions should be entered according to a 'right hand rule' convention.
4. Press the 'Accept Table' button in the upper right when finished.

To add connections between the nodes, press the 'Lines' worksheet as shown in *Figure 9*.

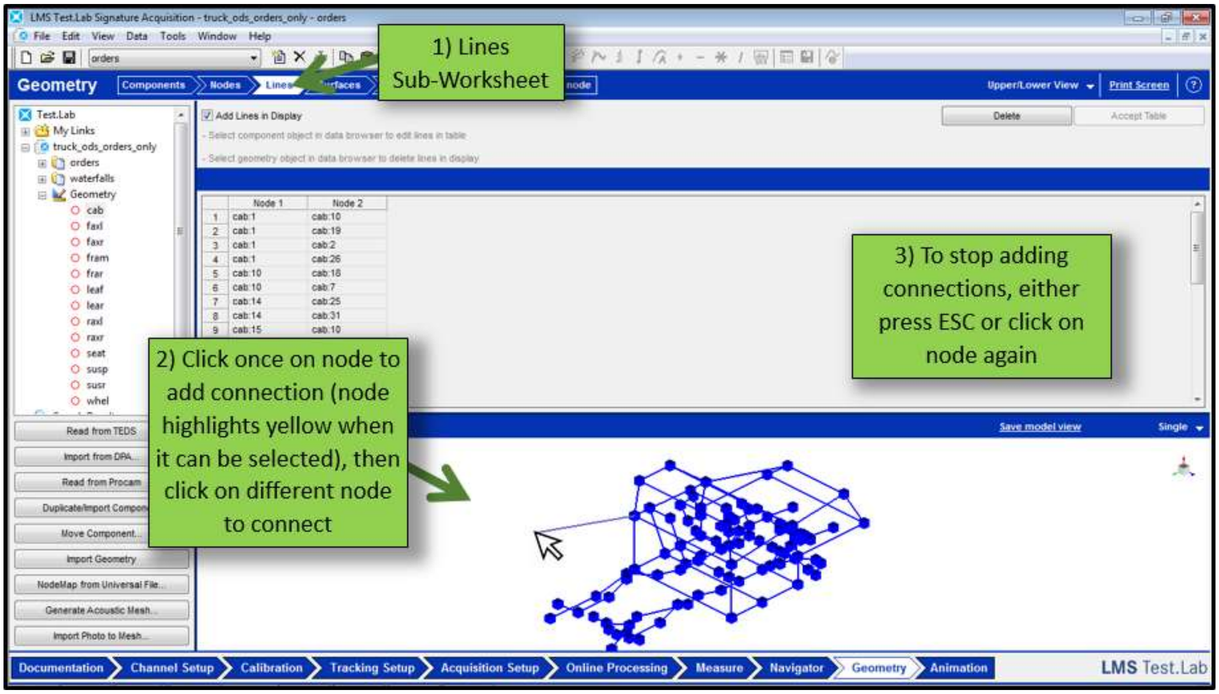


Figure 9: Lines sub-worksheet in Geometry

Lines can be added between points/nodes by:

1. Click on 'Lines' sub-worksheet.
2. In lower geometry display, hover mouse over first point to be connected. When the node 'highlights' click on it. Move the mouse to the node to connect. Click on it when it highlights to complete the connection. Repeat as needed.
3. To stop adding connections, either press 'ESC' key or click on node again.

If desired, the 'Surfaces' sub-worksheet can be used to create surfaces between points. With the geometry complete, now the measurements can be acquired.

Measurement

An important step when performing the measurement is to associate the measurements with the geometry of the test object. The software needs to know which physical measurement location corresponds to each point on the geometry.

This is easily done in the 'Channel Setup' worksheet of Simcenter Testlab Signature, Simcenter Testlab Spectral, and Simcenter Testlab Vibration Control. Select 'Use Geometry' from the pulldown in the upper right as shown in *Figure 10*.

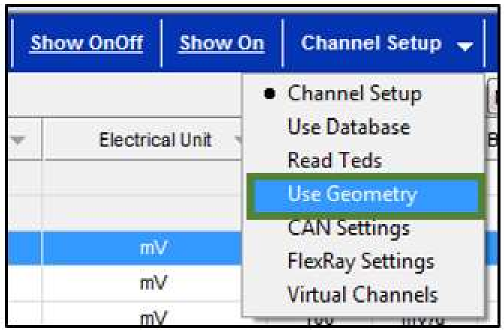


Figure 10: 'Use Geometry' in the upper right of Channel Setup

The geometry node and measurement point identification must be spelled exactly the same (case sensitive) to be associated as shown in *Figure 11*.

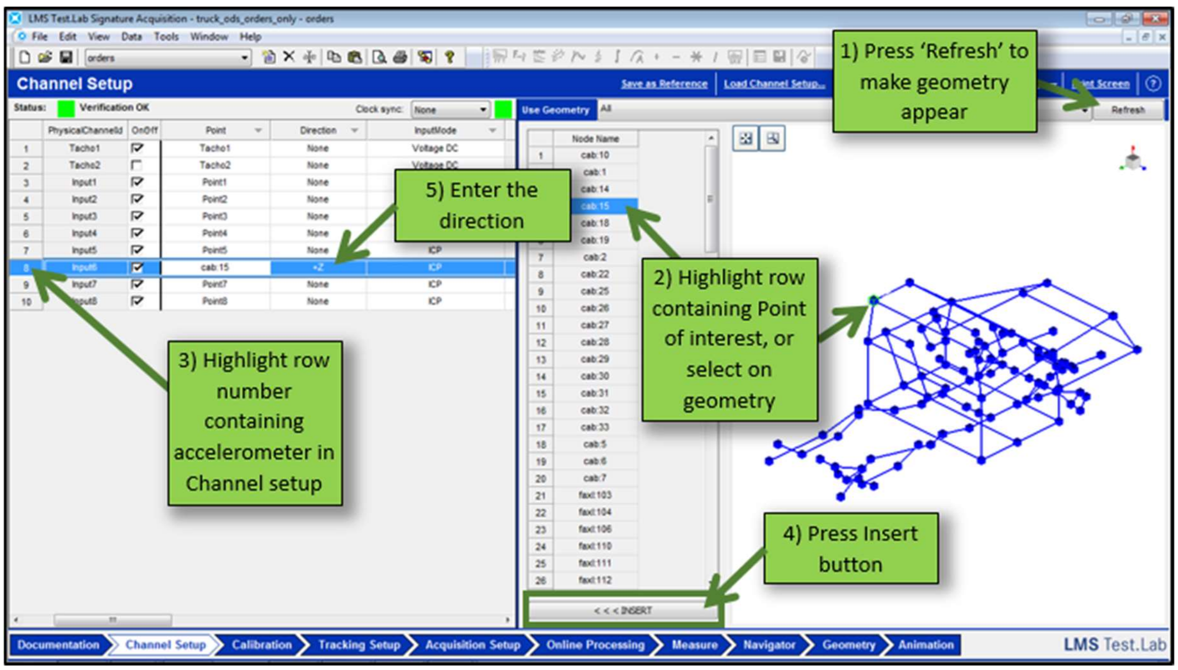


Figure 11: 'Use Geometry' in the upper right of Channel Setup

To create the connection between geometry and the accelerometer measurements properly:

1. After selecting 'Use Geometry', press the 'Refresh' button to view the geometry
2. Select the point to be measured by either highlighting the row with the 'Node Name', or click on the node directly in the geometry. If selecting directly from the geometry, the corresponding row in the node list will be highlighted.
3. In the 'Channel Setup' worksheet, highlight the channel to be associated with the node/point id by clicking on the associated row number.
4. Press the '<<<INSERT' button to copy the node to the channel identification.
5. Be sure to fill in the direction in the Channel identification: +X, -X, +Y, -Y, +Z, or -Z.

Autopower versus Spectrum

Next, the measurement must be setup to *ensure the phase is properly accounted* for between channels. If this is not done, the animation of the operational deflection shape will not be correct.

In the 'Online Processing' worksheet of Simcenter Testlab Signature, the measurement type can be changed from the default 'Autopower Linear' to 'Spectrum' as shown in *Figure 12*:

- The 'Autopower Linear' measurement function does not contain phase
- The 'Spectrum' does contain phase.

It is required to know the phase between measurements so the relative motion between points can be captured. Make sure to switch the measurement function in the 'Vibration' worksheet!

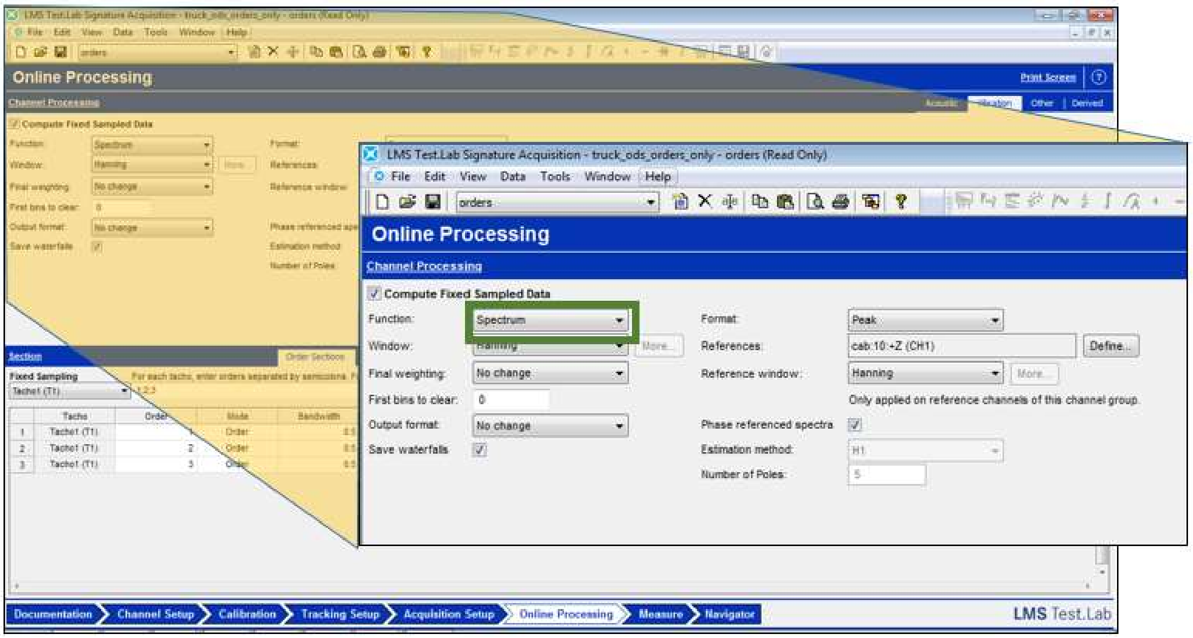


Figure 12: In 'Online Processing' set the function to 'Spectrum' instead of 'Autopower Linear'.

A phase reference measurement channel is required to successfully preserve the phase while roving measurement groups. It is also a good practice when acquiring all channels simultaneously.

Phase Reference Channel

To keep consistent phase between roves, at least one accelerometer should be kept at the same location while the others are moved. This accelerometer will be the phase reference channel that is used to preserve the phase among the different measurement sets.

Turn on the 'Phase referenced spectra' check box as shown in *Figure 13*. Then press the 'Define' button and select a reference channel that is not to be moved during the acquisitions.

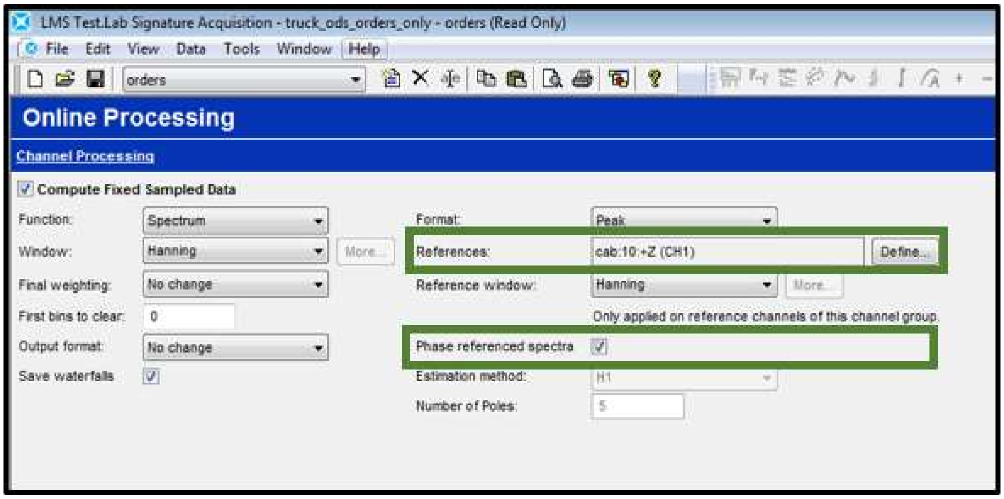


Figure 13: In 'Online Processing' turn on 'Phase referenced spectra' and define a reference channel that is not roved during the acquisition.

The reference channel should be on the test object and be fairly "active", i.e. have vibration that is related to the other channels. For example, if testing a truck, it *would not* make sense to have the reference accelerometer on the floor of the test laboratory, where the floor vibration is not related to the operation of the truck.

The phase of the reference channel is subtracted from both itself and all the other channels.

The reference channel will have a phase value of *zero* at all frequencies after the measurement, but the phase of all other channels will be *correct* relative to the reference channel as shown in *Figure 14*.

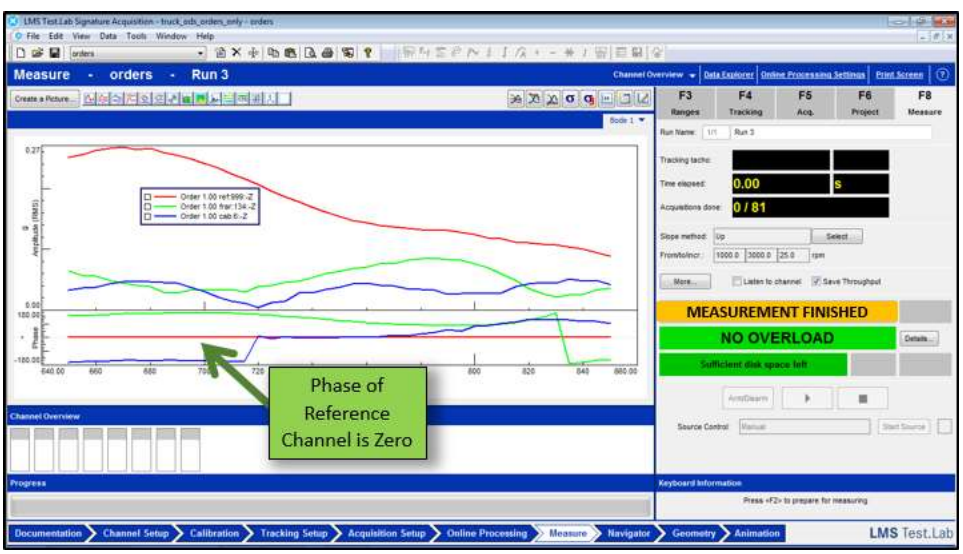


Figure 14: Phase reference channel (red) measurement has zero degrees for phase in lower graph

This phase reference will work with all frequency-based measurements: orders, spectrums, etc. The phase will be correct between the different measurement groups where the common channel was used.

Analysis

With the measurement completed, the analysis can begin. Turn on 'Tools -> Add-ins -> Operational Deflection Shapes & Time Animation' as shown in *Figure 15*.

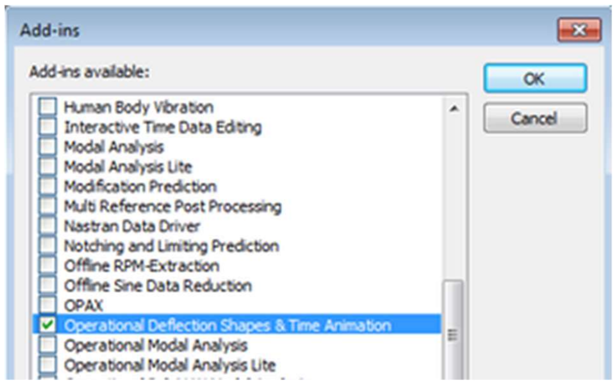


Figure 15: Tools -> Add-ins -> Operational Deflections Shapes

A new worksheet called 'Animation' is created as shown in *Figure 16*.

Figure 16: Animation worksheet

In the 'Animation Worksheet', animate the geometry with the measurement data as shown in *Figure 17*.

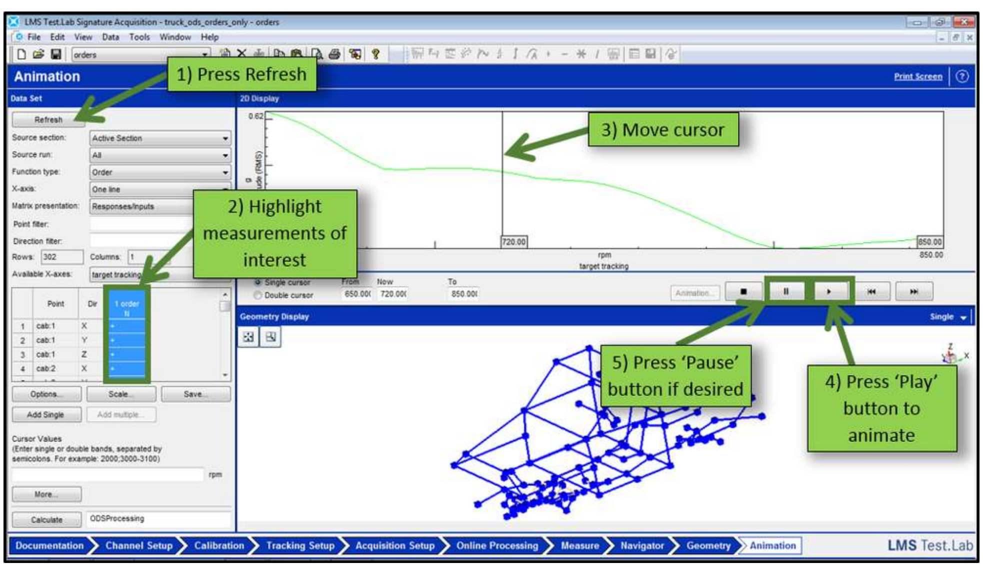


Figure 17: Animating the geometry with operational data

Animations are created by:

1. In upper left, press the 'Refresh' button. All measurements from the active section (even across multiple runs) will be made available for animation
2. Different measurements (example: 1st order, 2nd order, 3rd order) are sorted into different columns. Highlight the column of interest for the animation.
3. Move the cursor to the desired frequency or rpm
4. Press the 'Play' button to animate. The cursor will scroll thru the data along the X-axis.
5. If scrolling is not desired, press the 'Pause' button.

The operational deflection shapes can be saved as shown in *Figure 18*:

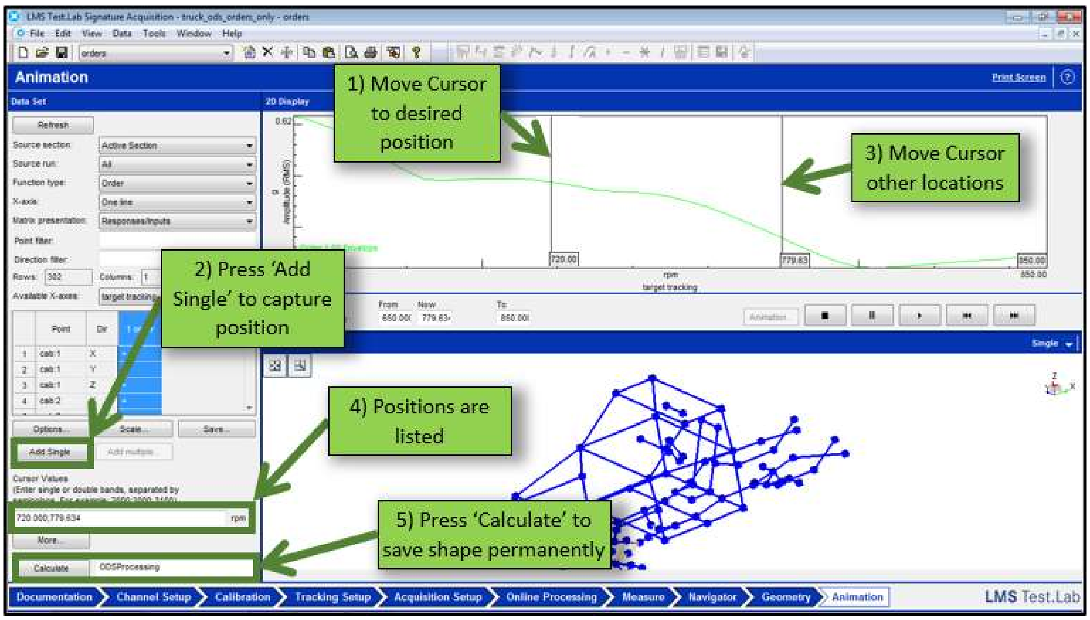


Figure 18: Saving an operational deflection shape

To save the operational deflection shape:

1. Move cursor to desired position. Position can be rpm or frequency.
2. After reaching position, press the 'Add Single' button to record the position.
3. Move to other positions and press 'Add Single' if multiple shapes are of interest.
4. Positions are listed with semi-colons between them.
5. Enter an analysis name, and then press the 'Calculate' button to save the shape.

The analysis is stored in the Simcenter Testlab project. The animation can be retrieved and viewed in the Simcenter Testlab Navigator worksheet. When viewing the previously stored results of the analysis, the 'Operational Deflection Shape and Time Animation' add-on is not required.

There are a few software options that can be helpful when doing Simcenter Testlab Operational Deflection Shapes as shown in *Figure 19*:

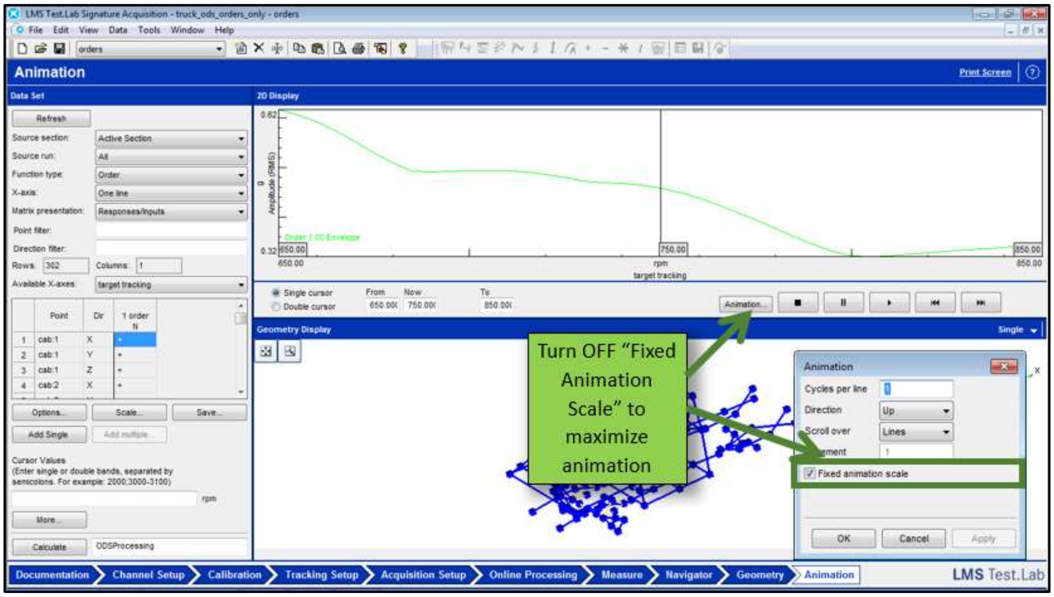


Figure 19: The 'Animation...' button options

Clicking on the "Animation..." button, the 'Fixed Animation Scale' can be turned on and off:

- If turned OFF, the animation will always be at full scale.
- When turned ON, the animation is scaled relative to the maximum vibration level of the entire measurement.

Not Just Vibration

Many other types of measurements can be visualized with operational deflections shapes, not just vibration. For example, acoustic data can also be animated, to visualize an acoustic cavity mode as shown in *Figure 20*:

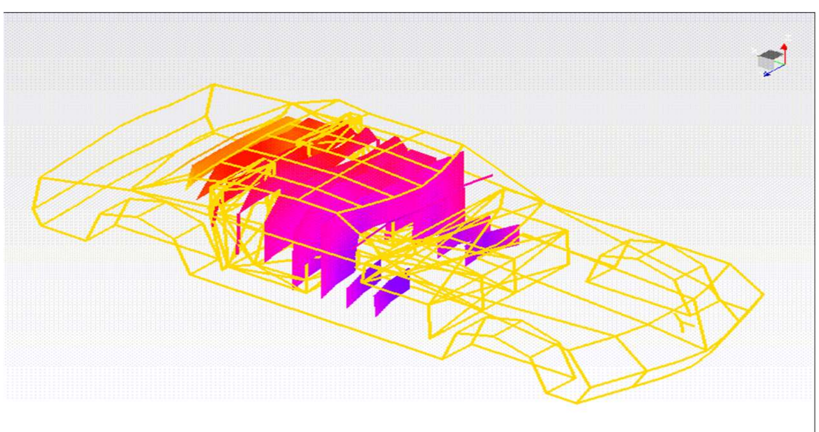


Figure 20: The acoustic shape of an interior cavity of a vehicle

Measuring an acoustic shape is the same process as creating a vibration shape. Only the transducer is changed from an accelerometer to a microphone.

Torsional vibration can also be visualized as shown in Figure 21. An additional visualization component, called a 'rotational pointer' is used to visualize the torsional rpm fluctuations.

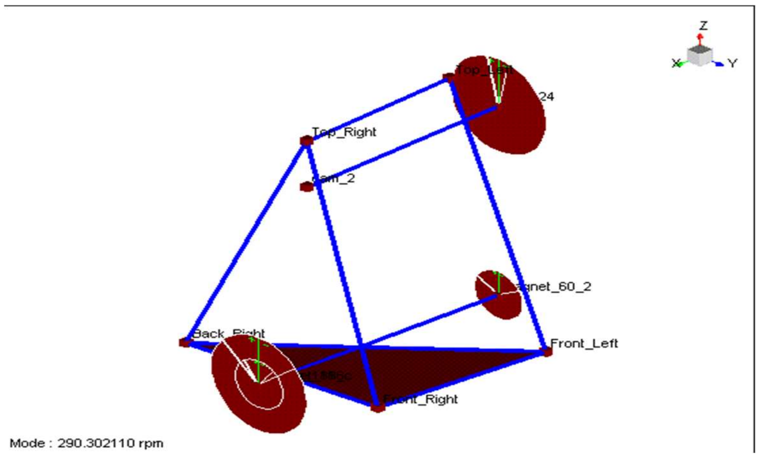


Figure 21: Torsional shape with rotational pointers

Different kinds of data (sound, vibration, torsional ...) can be visualized simultaneously. The Simcenter Testlab software scales each type of data separately to allow the animation to take place.

It is not just products that can be tested. Fixturing used in performing a test, including shaker heads as shown in *Figure 22*, are also useful. Shaker resonances can cause difficulty in performing sine tests due to total harmonic distortion.

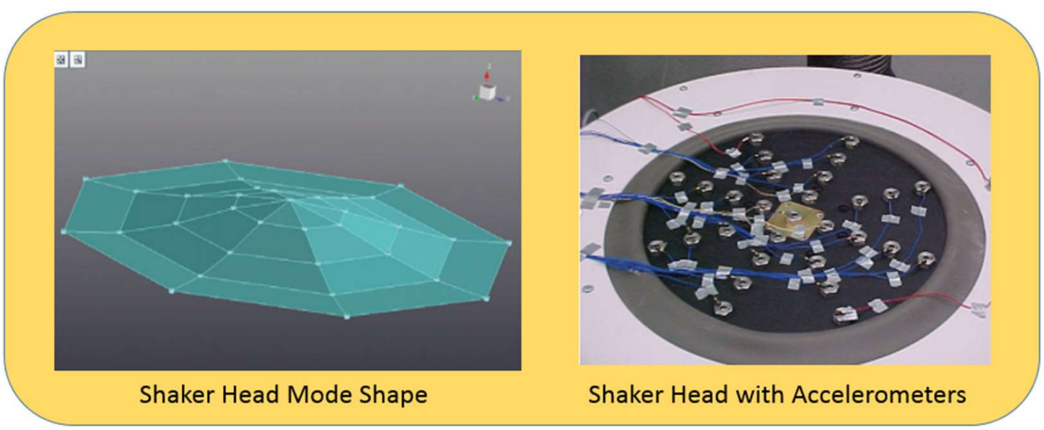


Figure 22: Operational deflection shape of shaker head

Enjoy operational deflection shape analysis!

Simcenter Testlab Modal Analysis: Modification Prediction

Think design modifications can only be made on Finite Element models? Think again...!

After performing an experimental modal analysis and calculating a set of modes, each mode has a mass and stiffness matrix that can be modified.

Simcenter Testlab Modal Analysis Modification Prediction can be used to:

- Add or subtract mass at a node or point
- Increase or decrease stiffness between two nodes or points
- Create tuned absorbers targeted to a frequency

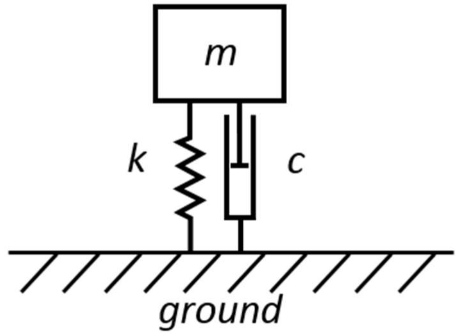
After creating a group of modifications, a new set of mode shapes and modal frequencies is calculated that incorporates the changes.

Spring-Damper and Mass Modification Background

Consider a single degree of freedom system, consisting of:

- Mass (m)
- Stiffness (k)
- Damping (c)

The natural frequency (wn) is equal to the square root of the stiffness over the mass as shown in *Figure 1*.



$$\omega_n = \sqrt{k/m}$$

Figure 1: Single Degree of Freedom Mass-Spring-Damper system

A modal frequency can be increased by:

- decreasing mass
- increasing stiffness

This holds true for all structures, even more complicated ones.

A 'Spring-damper' modification can be used to alter the stiffness between two points as shown in *Figure 2*.

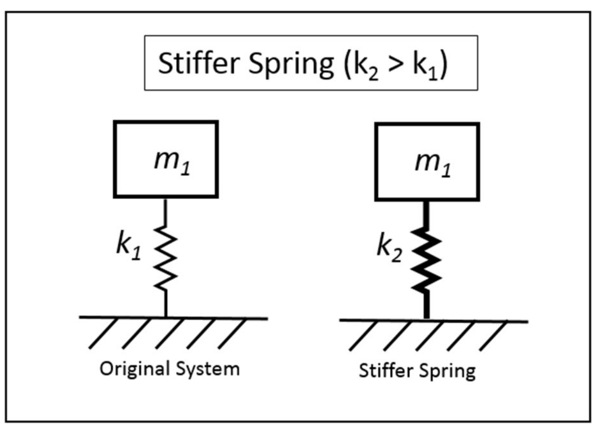


Figure 2: Modified mass-spring system with increased stiffener

By increasing the stiffness of the spring, the modal frequency will shift higher as shown in *Figure 3*.

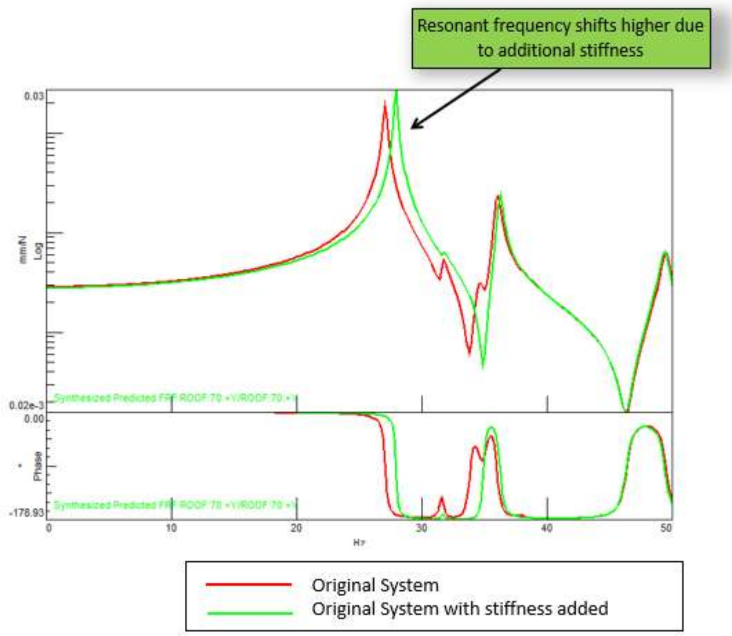


Figure 3: Frequency Response Function of original system and system with increased stiffness

How would increasing stiffness address vibration issues? For example, if vibration from driving on rough road ranged from 1 to 20 Hz, increasing the first body modes beyond 20 Hz would reduce vibration experienced by the driver. In addition to the stiffness, the damping can also be changed.

Tuned Absorber Background

A tuned absorber is a secondary mass-spring system that is added to an existing mode as shown in *Figure 4*.

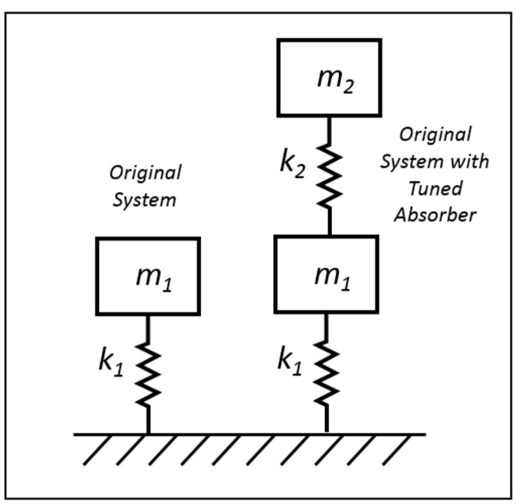


Figure 4: Tuned absorber (m2, k2) applied to mass-spring system (m1, k1)

A tuned absorber takes the original frequency of the original system and divides it into two modes. The frequency of the first mode is lower than the original system. The frequency of the second mode is higher than the original system as shown in *Figure 5*.

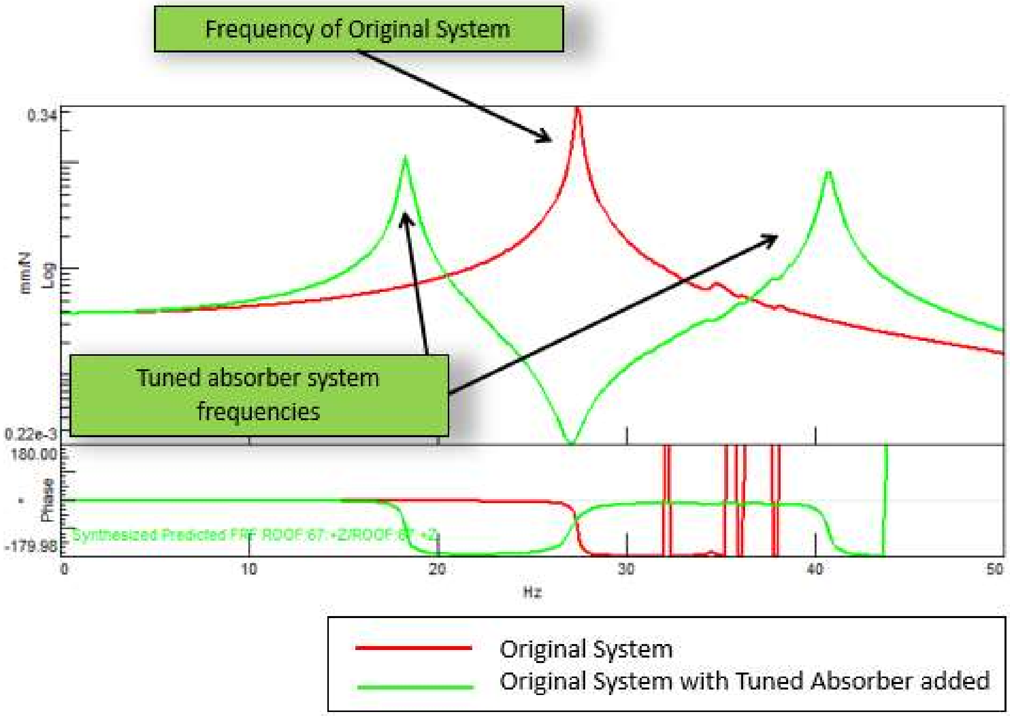


Figure 5: Original System versus Tuned Absorber system

The mode shape of the lower frequency would have both the original system mass (m_1) and the tuned absorber mass (m_2) move back and forth *in phase*. The two masses would move back and forth *out of phase* in the higher frequency mode. This is illustrated in *Figure 6*. If the tuned absorber mass and stiffness is carefully selected, the motion on the original system can be forced to zero by the absorber.

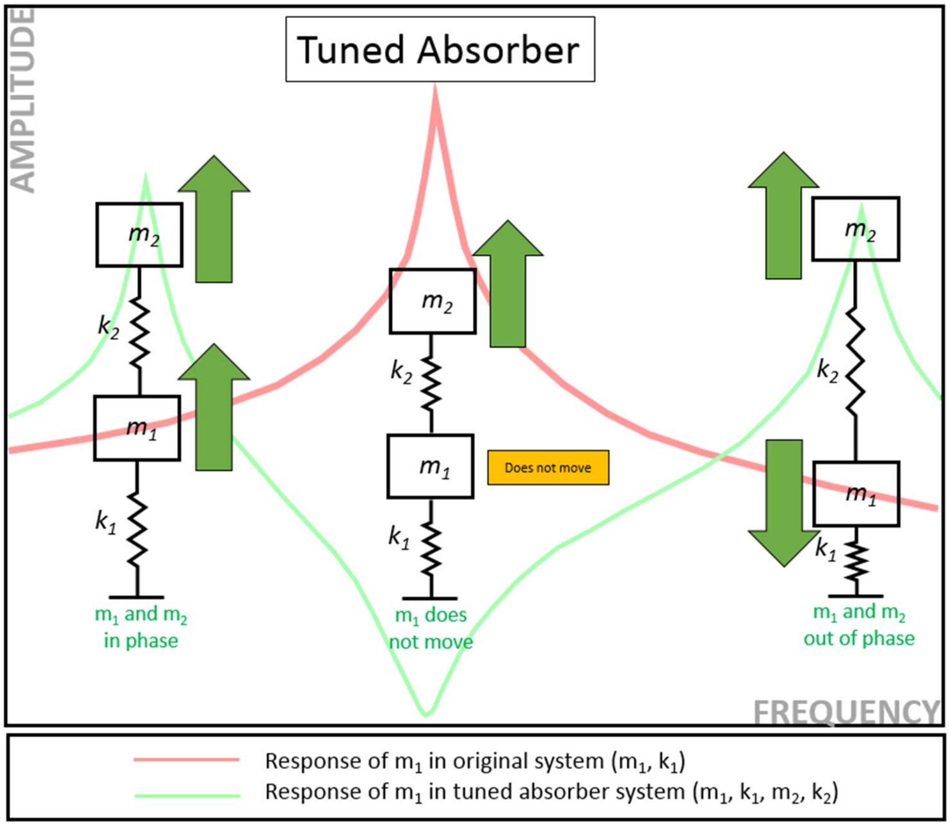


Figure 6: Original System versus Tuned Absorber system

How can a tuned absorber be used to abate a noise and vibration issue? Consider a vehicle where the combustion frequency of engine idle excites a bending mode of the steering wheel column. A tuned absorber could be placed on the end of the steering wheel. This would place one mode of vibration at a frequency lower than the idle which would never be excited. The higher mode could be placed at an engine combustion frequency that the vehicle does not commonly operate at, like 30 mph.

Tuned absorbers are useful when an existing mass can be used. For example, the airbag module at the end of the steering wheel is an existing mass that could be sprung to create a tuned absorber. This modification would not add mass to the overall vehicle (which would have an adverse affect on fuel efficiency). It might also be a cheaper modification than stiffening the steering column to avoid the resonant situation.

Additional Considerations

When using experimental modes for modification prediction, the following should be considered:

1. Calibration – It is important that the proper calibration was used during data acquisition. This includes both the input and response transducers. If the accelerometer calibration was off by a factor of 100, a 2 kilogram modification could act like a 200 kilogram modification.
2. Dimensions - Proper dimensions should be used when creating the geometry. The dimensions can affect the modification prediction results.
3. Out-of-Band Modes – To have accurate modal predictions, it is advisable to have at least one mode below and one mode above the frequency of interest for the modification. It is even

better if several modes above and below the frequency of interest is included. When using a modal model for a limited frequency band it is possible that important structural modifications would generate modes with a natural frequency outside the range of this frequency band. Since the original modal models are not valid at these frequencies, the predicted results will not be very reliable.

4. Proper Frequency Shift - If adding mass, modes should shift down. If they shift up instead, there is a high probability that there is a sign direction convention problem. For example, on the input point, the +Z direction may have been substituted for the –Z direction. Check the directions again.

Getting Started with Simcenter Testlab Modification Prediction

Under “Tools -> Add-in” from the main menu, select “Modification Prediction”. If using Simcenter Testlab tokens, it requires 23 tokens total (*Figure 7*).

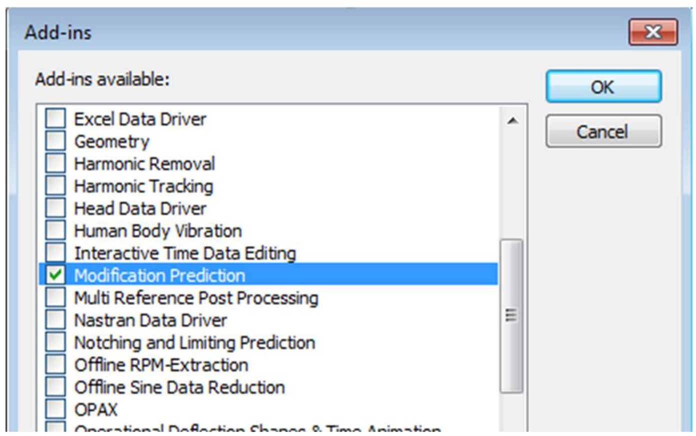


Figure 7: Tools -> Add-ins -> Modification Prediction

A new worksheet called ‘Modification Prediction’ appears at the bottom as shown in *Figure 8*.

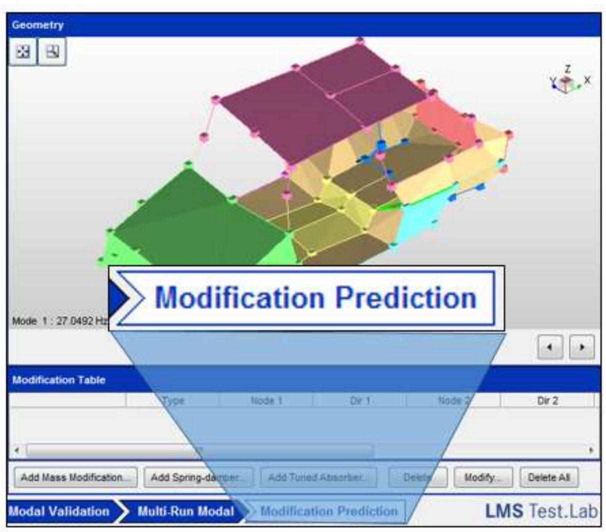


Figure 8: Tools -> Modification Prediction on worksheet

There are two minor worksheets at the top of the 'Modification Prediction' worksheet:

- List Modifications – Create the set of modifications to be applied
- Predict Modes – Calculate and view new mode set with modifications

Select a set of modes for modification in the upper left corner of the 'Modification Prediction' worksheet. Drag and drop modal frequencies over the geometry to view the shape as shown in *Figure 9*.

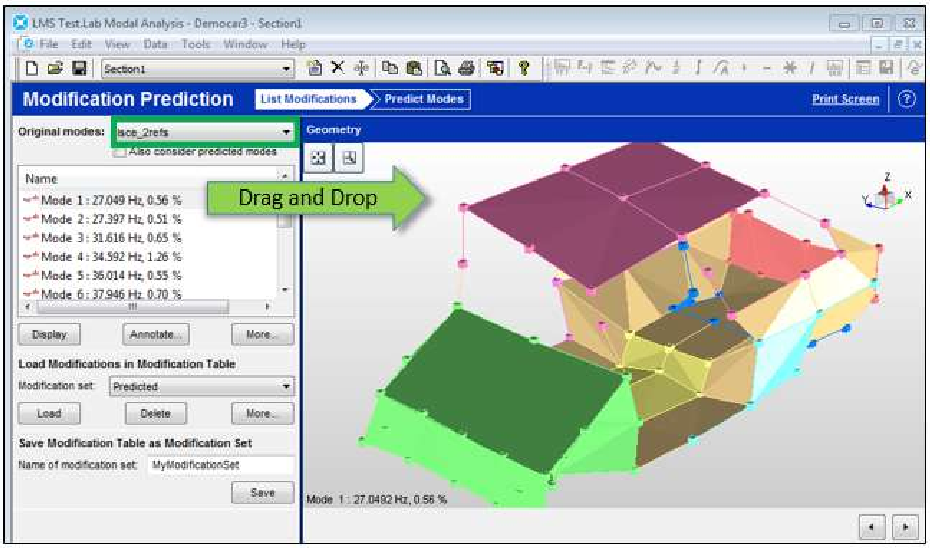


Figure 9: Select mode set in upper right, drag and drop to geometry

Viewing the mode shape aids in deciding appropriate modifications to alter the modal frequency.

Making a Modification: Spring-Damper

Choose a modification of interest. For example, this vehicle body has a torsion mode as shown in *Figure 10*. Adding additional stiffness around the windshield can make the torsion mode frequency higher.

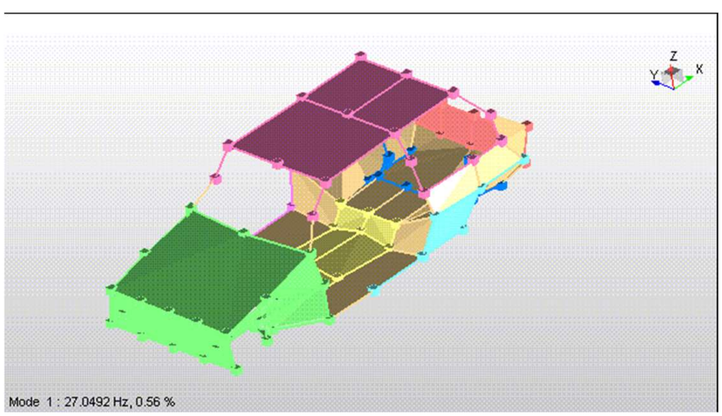


Figure 10: Vehicle body torsion mode

To add a stiffener, select 'Add Spring-Damper' at the bottom of the screen as shown in *Figure 11*.

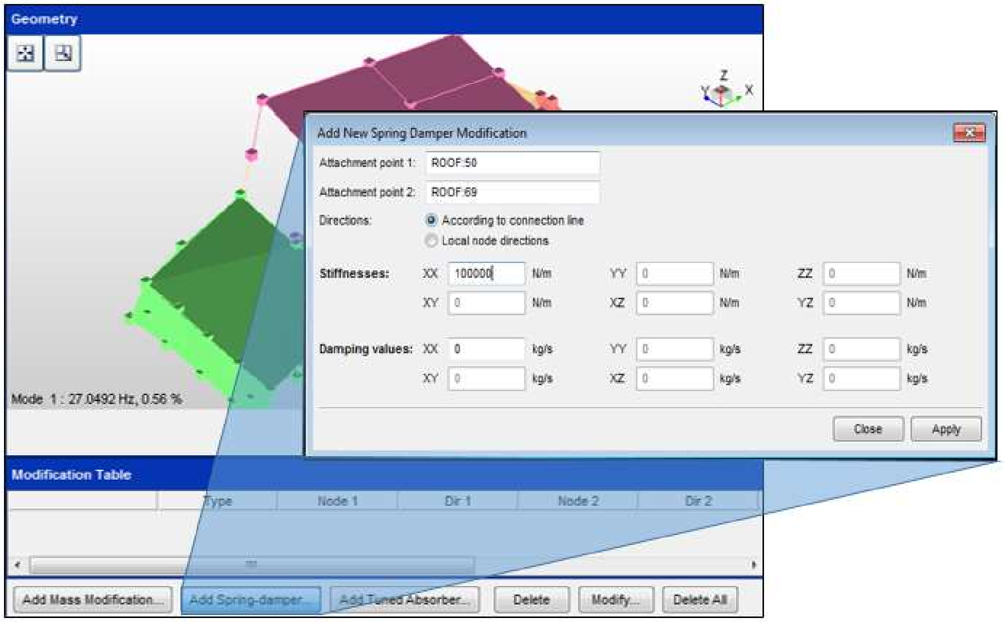


Figure 11: 'Add Spring-Damper' button

In the 'Add Spring Damper' menu, define two connection points and a spring constant value. By default, with the "According to connection line" setting, the XX stiffness is added axially between the two points as shown in *Figure 12*. Damping can also be added.

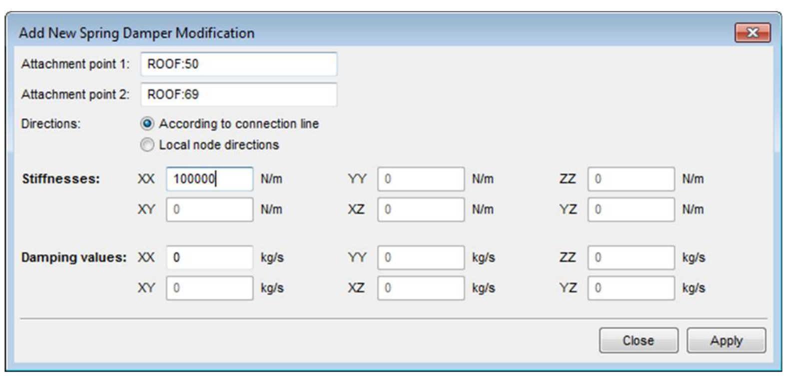


Figure 12: Select mode set in upper right

It is not necessary to type the node names into the 'Attachment point' menu fields. You can simply click on the node in the geometry and it will fill into the 'Attachment point' automatically, as shown in *Figure 13*. Press the 'Apply' button when finished.

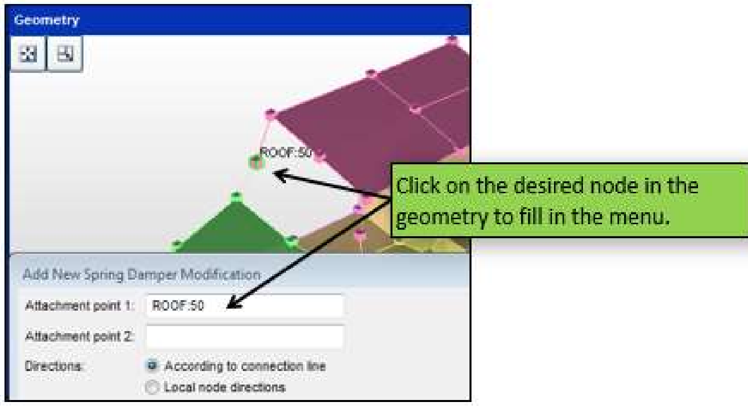


Figure 13: Select mode set in upper right

Multiple modifications can be made. In this case, two 'Spring-damper' elements were added to increase the torsion mode of the vehicle as shown in *Figure 14*.

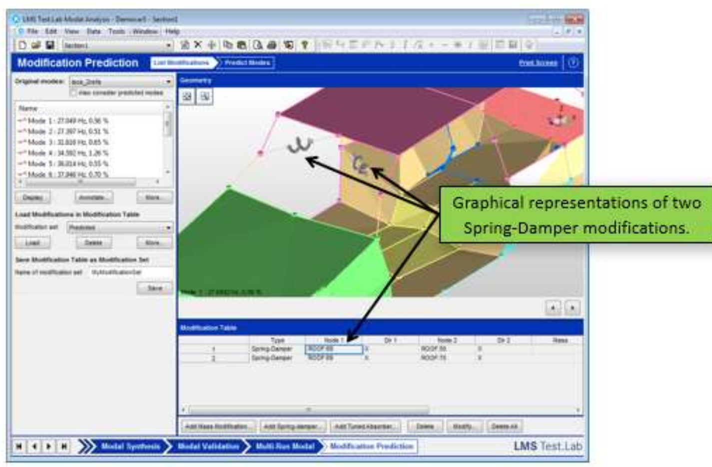


Figure 14: Multiple 'Spring-Damper' modifications

Click on the 'Predict Modes' tab at the top of the 'Modification Prediction' worksheet as shown in *Figure 15*. Press the 'Calculate' button to create a new set of modes, with the modifications applied.

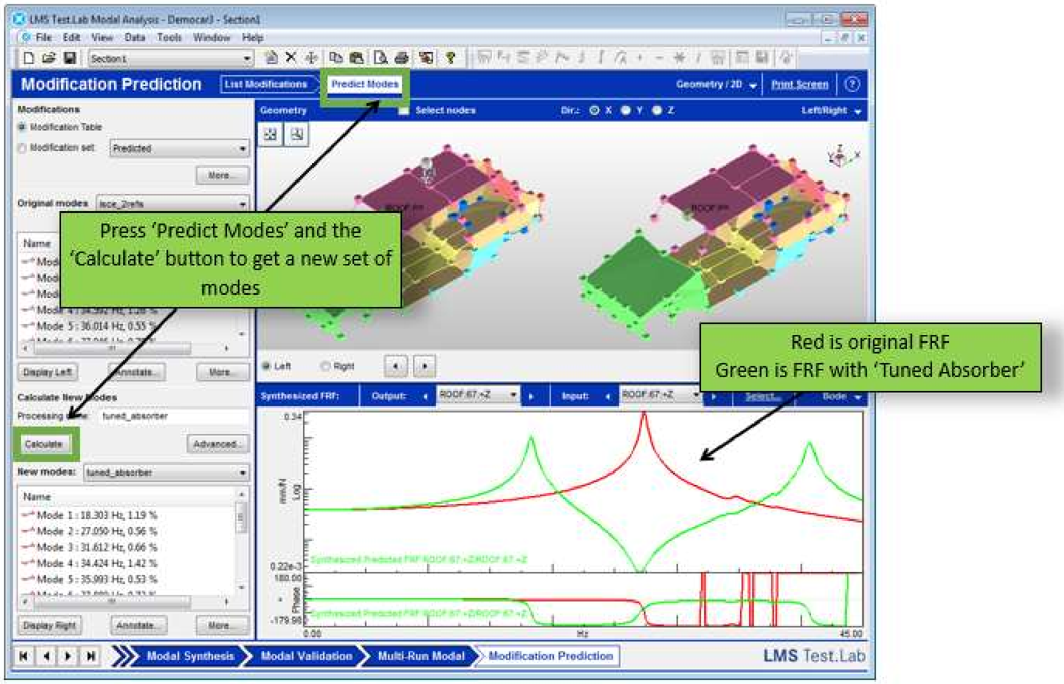


Figure 15: Press 'Predict Modes' button then the 'Calculate' button to calculate a new set of predicted modes

The Frequency Response Functions (FRFs) of on the points of interest are displayed. Green is from the modified mode set, while red is from the original mode set.

Making a Modification: Tuned-Absorber

At 27.39 Hz, there is a roof pumping mode as shown in *Figure 16.* A tuned absorber can be applied to this mode.

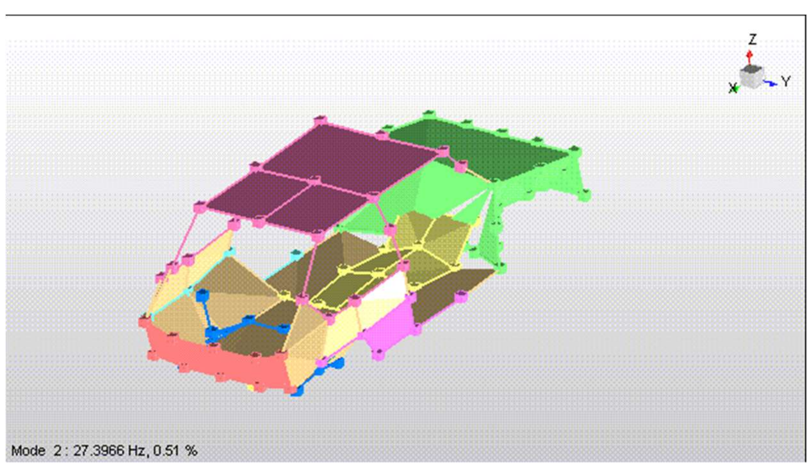


Figure 16: Roof pumping mode

Click on the 'List Modifications' minor worksheet. If desired, eliminate any previous modifications by highlighting the entire row and pressing the 'Delete' button as shown in *Figure 17*.

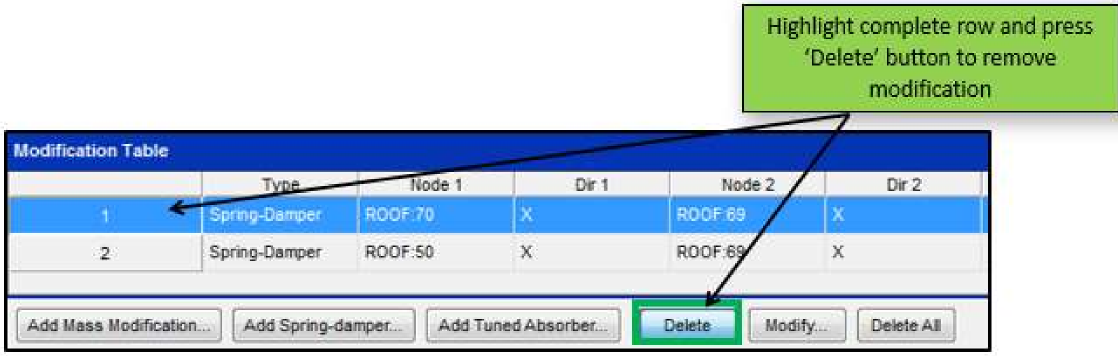


Figure 17: Highlight complete row and press 'Delete' button to remove a modification

Click on the 'Add Tuned Absorber' button.

In the tuned absorber menu (*Figure 18*):

- Select an attachment point (can be done by clicking on node in geometry display)
- Enter a tuned absorber mass
- Enter a frequency to target with the absorber
- Then press 'Tune' button to calculate a stiffness and damping value
- Press 'Apply' when finished

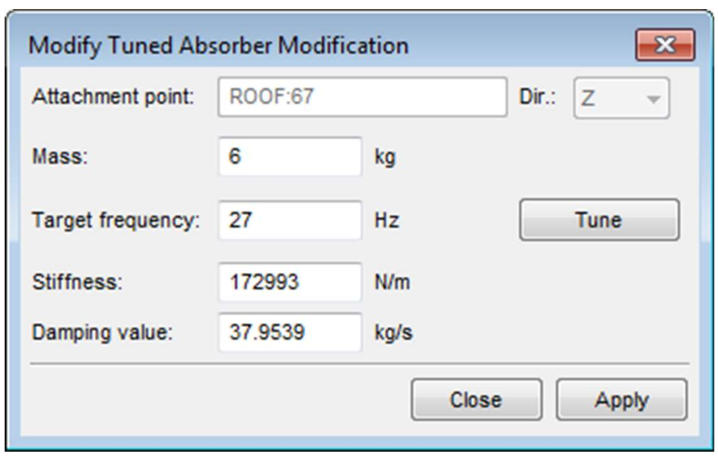


Figure 18: 'Modify Tuned Absorber Modification' menu

Press 'Predict Modes' tab to create a new set of modified modes. Just above the 'Calculate' button, on the middle left, enter text in the 'Processing data' field to name the new data set. Press the 'Calculate' button to apply the tuned absorber as shown in *Figure 19*.

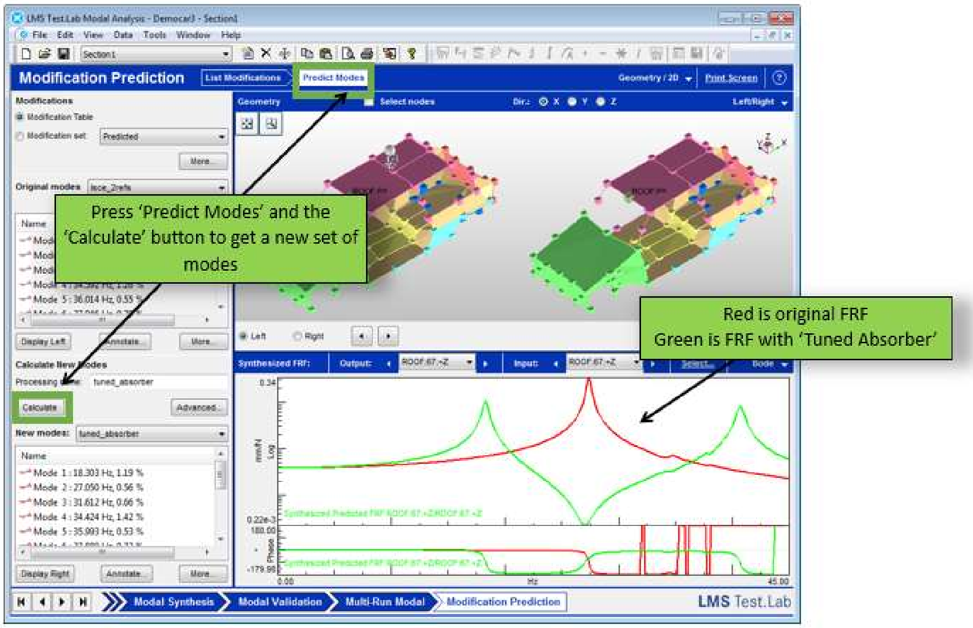


Figure 19: 'Predict modes' and 'Calculate'

The new modes and their frequencies are listed in the lower left of the screen. The FRF with the red line is based on the original set of modes. The FRF with the green line includes the tuned absorber modification.

Tuned Absorber Trivia

From a dynamics point of view, skyscrapers are equivalent to long metal beams coming out of the ground. They have low frequency modes of vibration excited by the wind. The top of a skyscraper can move many feet. Many have tuned absorbers to help reduce the movement/vibration.

The Taipei 101 skyscraper contains the world's largest and heaviest tuned mass dampers, at 660 metric tons (730 short tons) as shown in *Figure 20*.

Figure 20: Taipei 101 (left) and tuned absorber (right)

The tuned absorber is viewable by the public on an indoor observation deck at the top of the skyscraper. It cost an estimated $4 million to build.

Conclusion

Use Simcenter Testlab Modal Analysis Modification Prediction to simulate product design modifications on experimental modal analysis results. The following modifications can be done:

- Add or subtract mass at a node or point
- Increase or decrease stiffness between two nodes or points
- Create tuned absorbers targeted to a frequency

After creating a new mode set, it is interesting to use the Modal Assurance Criterion (MAC) to compare the original set of modes to the modified modes as shown in *Figure 21*.

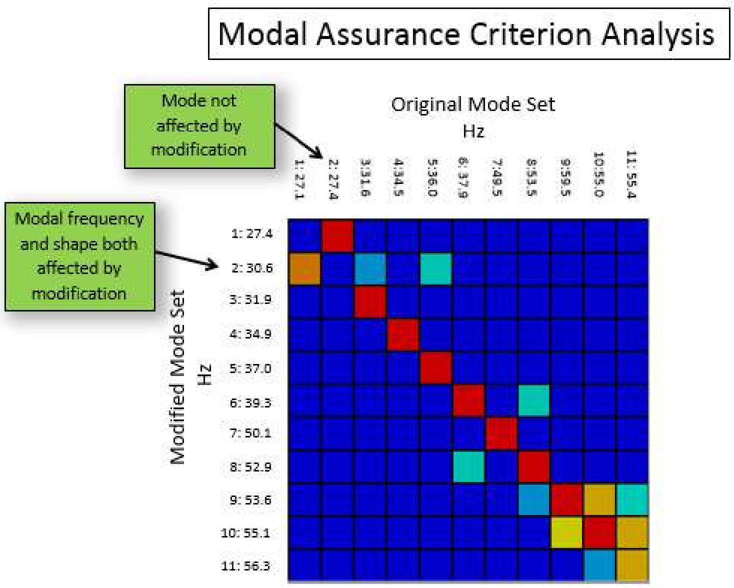


Figure 21: Modal assurance criterion analysis of original versus modified mode set

Using the MAC analysis, it is possible to see the effects of modifications on modal frequencies and mode shapes. In the MAC table of *Figure 21*, red indicates modes that are 100% alike.

- For these modes with a MAC of 100%, the frequencies were changed by the modifications, but the mode shape was not.
- The orange color indicates modes that are 80% alike. In this case, we can see that the second mode of the original structure at 27.4 Hz was shifted upward to 30.6 Hz, and the shape was significantly changed.

Maximum Likelihood Estimation of a Modal Model (MLMM)

Spending a lot of time finding the best poles in your stabilization diagram?

Stop wasting your time! Put the computer to work to find the optimal solution using the new Maximum Likelihood estimation of a Modal Model (MLMM) feature in Simcenter Testlab.

Released with Simcenter Testlab 17 (now called Simcenter Testlab), the new MLMM Modal Parameter estimator automatically iterates on the parameters of the initial modal model to optimize the fit between the identified model and the measured Frequency Response Function (FRF) data, resulting in the best fitting modal model. Depending on the quality of the initial fit, the results can be significantly improved, as seen in *Figure 1*.

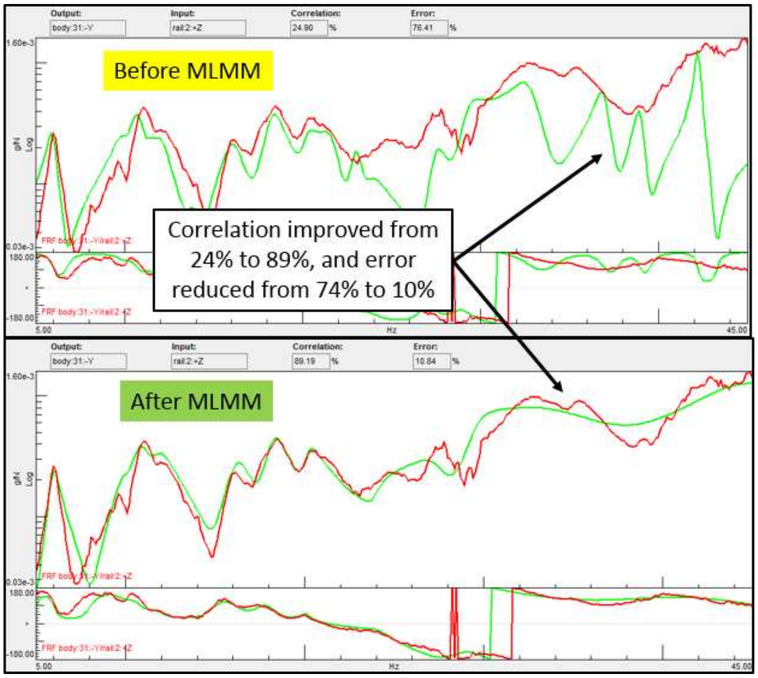


Figure 1: MLMM automatically improved this curve fit on a heavily damped structure, reducing error from 74% to 10% for this FRF synthesis

The initial modal model can be calculated either using Polymax or Time MDOF curve fitting routines, as is standard practice today. But instead of manually iterating one mode at a time between the stabilization diagram and the modal synthesis, the MLMM takes over! It automatically adjusts the frequency, damping, and modal participation values to reduce the difference between the modally calculated FRF and actual FRF for all identified modes.

For heavily damped structures and complex systems such as acoustic cavities or trimmed-body analysis, MLMM has significantly higher quality results. However, for other structures and curvefits, where the initial fit was very accurate, there is not significant improvements using MLMM.

Using MLMM

The Maximum Likelihood Modal Model is available for 26 tokens in Simcenter Testlab. From the main Simcenter Testlab menu, select "Tools -> Add-ins" to turn on "MLMM" as shown in *Figure 2*.

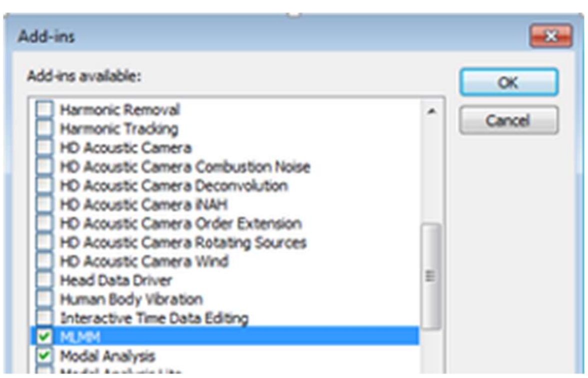
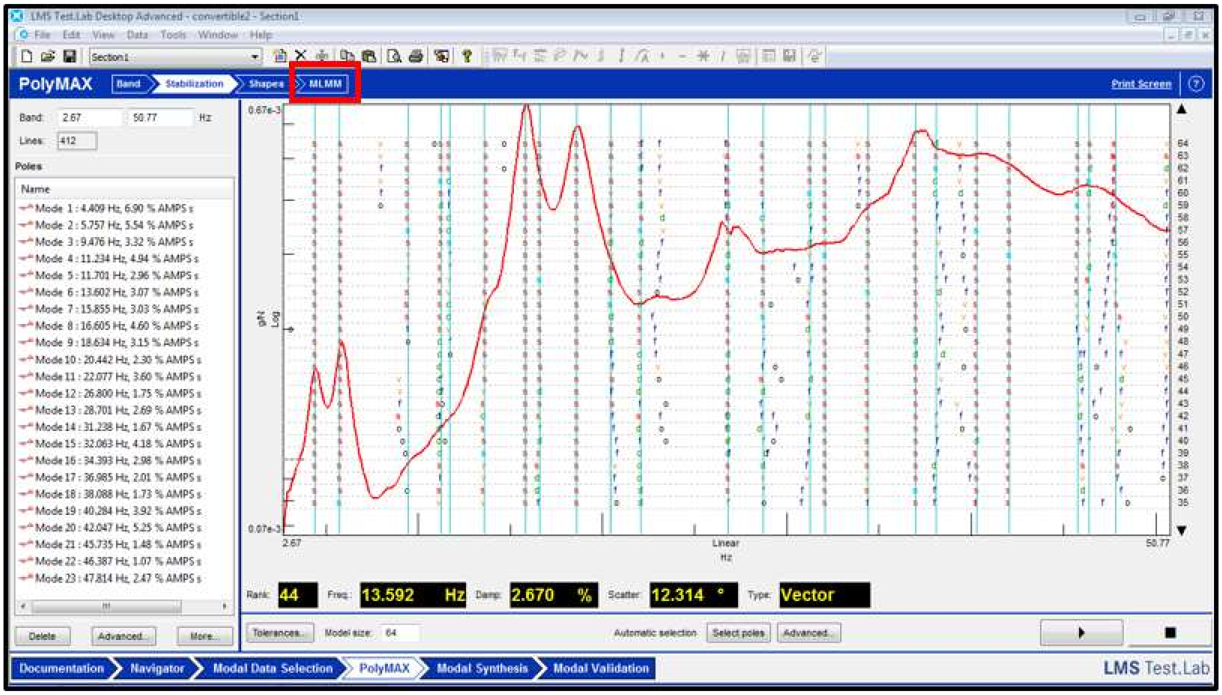


Figure 2: Tools -> Add-ins -> MLMM

At the top of the Polymax worksheet, an additional minor worksheet or tab is added to the Modal curve fitter as shown in *Figure 3*.



Select poles (the red "s") from the stabilization diagram as normal. After creating the list of poles on the left side, press the "MLMM" button on top of the screen.

Select poles from the stabilization diagram as normal. After creating the list of poles on the left side, press the "MLMM" button on top of the screen.

In the MLMM worksheet, set the "Maximum number of iterations". This is the number of times that the frequency, damping, and modal participation factors will be adjusted. Press the "Calculate" button on the left side of the screen as shown in *Figure 4*.

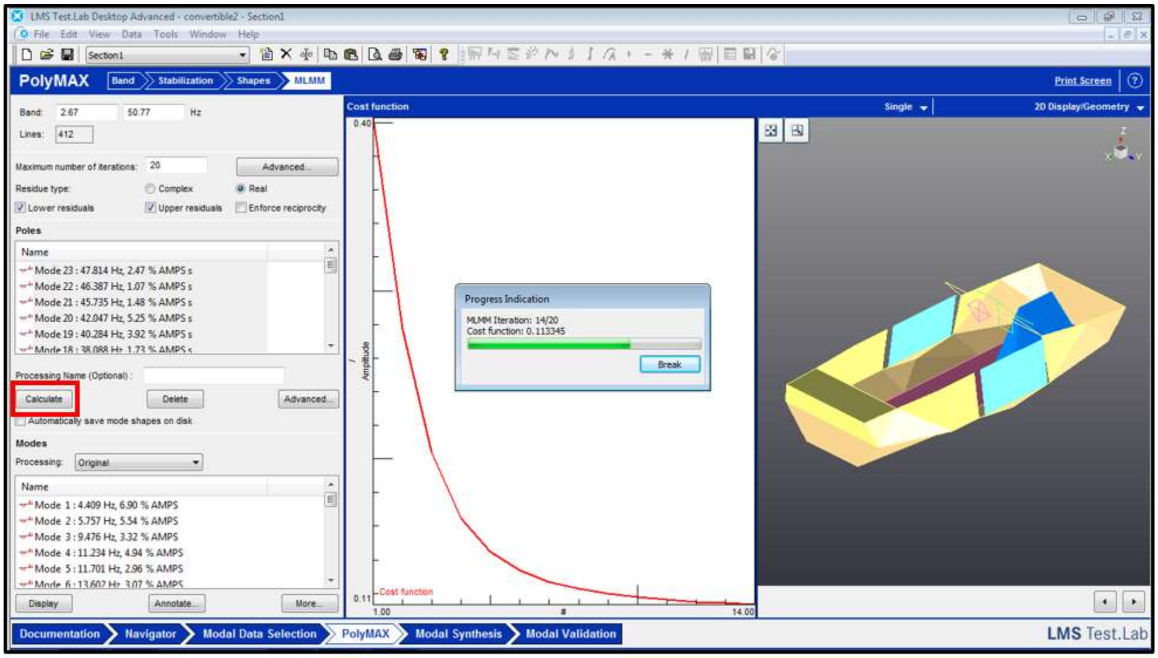


Figure 4: With each iteration the error between the experimental FRFs and synthesized FRFs becomes smaller

In the middle of the screen the error between the synthesized FRFs of the model and the actual measured FRFs is shown. With each iteration, it should get progressively smaller. Eventually the changes become smaller and smaller. Once the iterations are complete, the modal synthesis can be checked. Click on the 'Modal Synthesis' worksheet as shown in *Figure 5*.

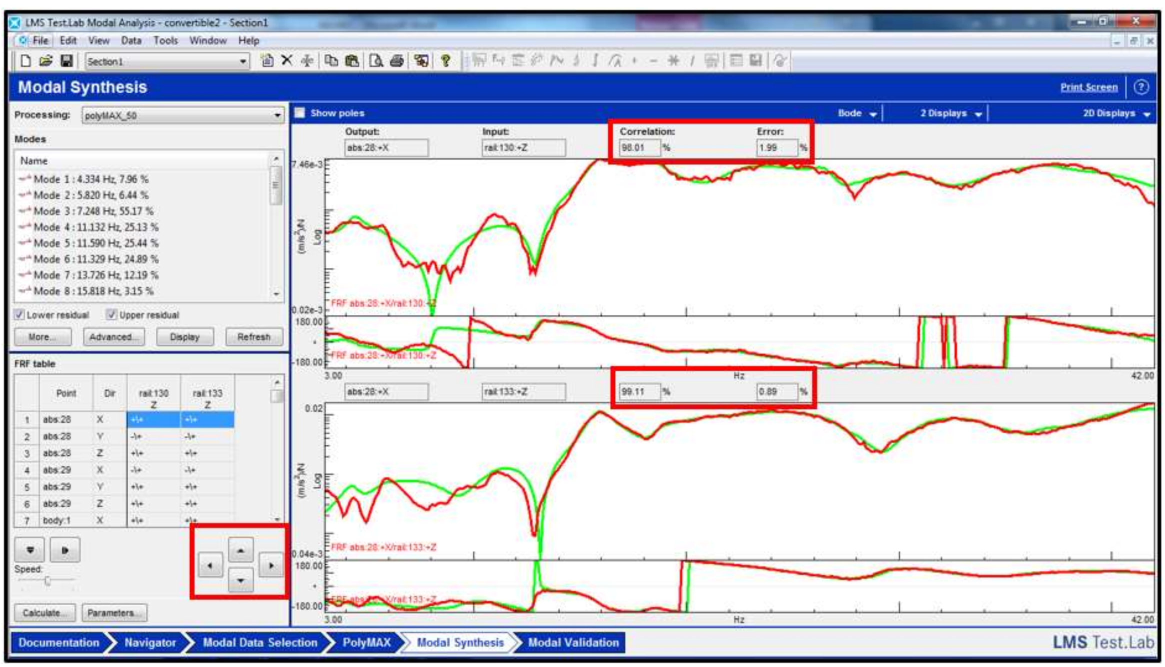


Figure 5: Modal synthesis after MLMM

In the Modal Synthesis worksheet, there is a correlation and error percentage shown above the FRFs:

- The correlation indicates how well the shape of the synthesized FRF follows the shape of the actual FRF. Ideally this should be 100%
- The error indicates the difference between the amplitude of the calculated FRF and synthesized FRF. Ideally this should be 0%.

The arrow buttons, located in the lower left of the screen, can be used to view FRFs from different measurement locations.

Under the "Advanced..." button of the MLMM worksheet, some constraints can be set on the frequency and damping values as shown in *Figure 6*.

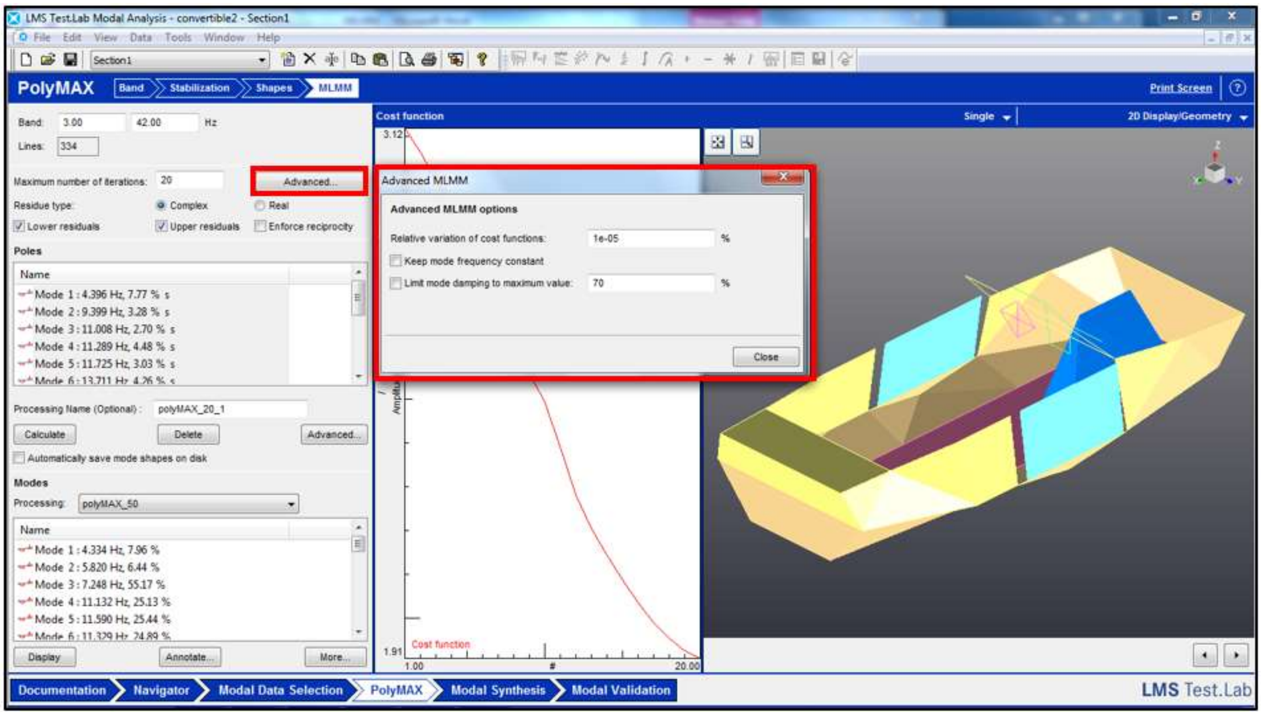


Figure 6: Known frequencies and maximum damping values can bet set as constraints in MLMM

The following constraints can be set:

1. Maximum percent variation between iterations
2. The "Keep mode frequency constant" can be checked to prevent frequencies from being adjusted
3. Limit modal damping to a maximum different than the default 70%. After iterating, any modes that MLMM did not think helpful for describing the FRFs get set with a large damping value. By default this value is 70%.

Enjoy using MLMM to speed up the curvefitting process.

Ground Vibration Testing and Flutter

Two closely related dynamic tests performed on aircraft are Ground Vibration Tests (GVT) and Flutter Tests. These tests are used to help define the safe flight “envelope” for a given aircraft configuration:

- The GVT is a modal test of the entire airframe, which determines the resonant frequencies, mode shapes, and damping. This is performed on the aircraft before first flight while it is still on the “ground”. GVT modal results are compared with the structural Finite Element Model used for flutter predictions. If the models and test results agree, the aircraft is cleared for the next round of tests, which usually includes ground runs and a flight flutter investigation.

- Flutter is an in-flight phenomenon which causes the wings and other key components to vibrate uncontrollably, due to the convergence of multiple modes as the resonance and damping shift due to aero-elastic loading, making flight impossible. Before flying an actual aircraft, flutter performance is predicted, so the flight envelope (altitude and speed) of the aircraft is safely bounded. This is later verified by performing a flutter test with the actual aircraft.

The analytical flutter predictions bring together a structural model and an aerodynamic model. The aerodynamic model is verified by scale model testing in a wind tunnel, while the structural model is verified by GVT testing.

The Importance of Flutter

Flutter is a dynamic instability of an elastic structure in a fluid flow, caused by positive feedback between the body's deflection and the force exerted by the fluid flow. The interaction of these different forces are depicted in the Collar diagram in *Figure 1*.

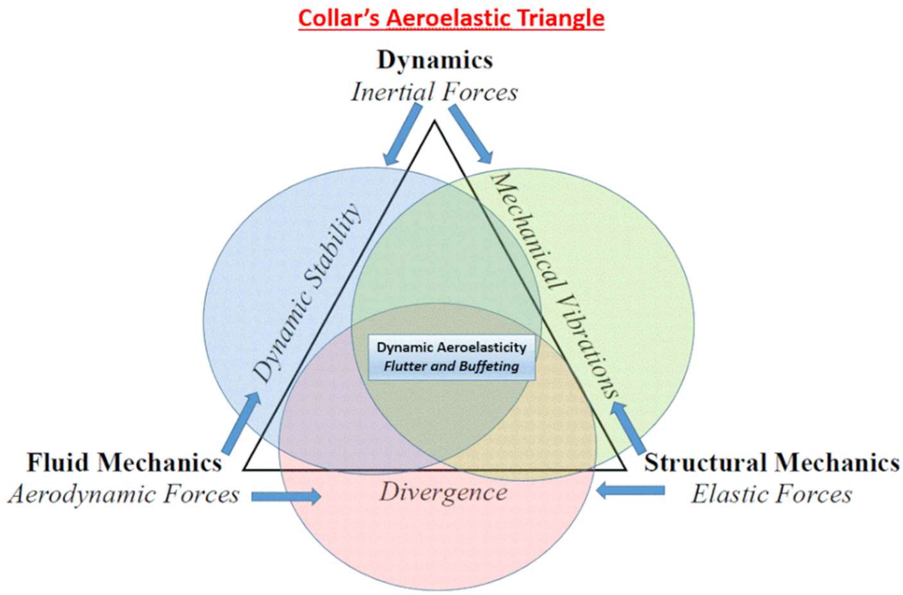


Figure 1: The interaction of the different forces that are in play for an airplane are depicted in the Collar Diagram. The coalescence of these can result in an unstable condition called flutter.

Flutter needs to be avoided since this can be a catastrophic event, and total loss of the airframe is possible. Model predictions need to be supplemented with ground testing to build up absolute confidence in the predictions.

After verification with analysis, an actual flutter flight test methodically and carefully expands the flight envelope (*Figure 2*) based on altitude and flight speed, where the airplane is designed to fly.

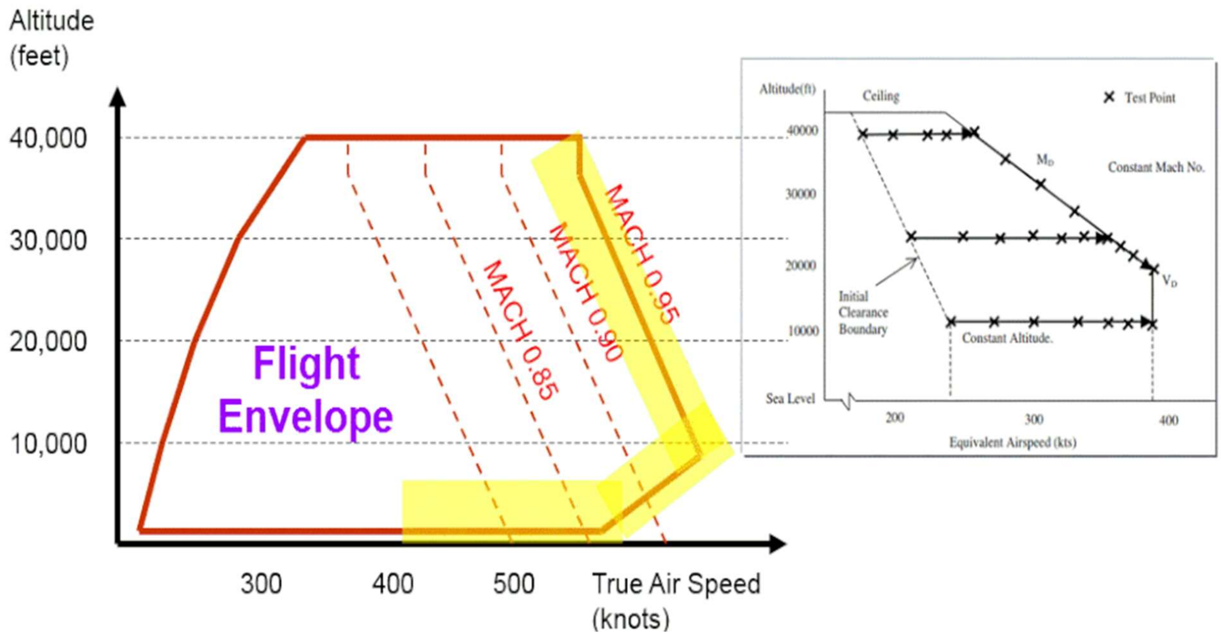


Figure 2: Expanding the flight envelope is done slowly and methodically at specific "flight data points" during flutter flight testing. Test points are shown in the right-side plot.

The FAA will type certify an airframe for its specific flight regime only after the successful GVT and flutter flight test is completed for each structural derivative.

Performing a GVT Test

The duration of the GVT campaign can consume from a few days to a couple of weeks, depending on the size of the airframe, the number of channels, the number of different payload or mission configurations, etc.

Pretest planning to ensure testing efficiency is very important so that the down time of the new aircraft is minimized. If the preliminary finite element model is available, it can be used to help in the most efficient configuration of the test, including shaker and accelerometer number and location, and boundary condition verification.

Building up to a full aircraft GVT is given in the product development timeline in *Figure 3* as depicted with the famous "V" diagram.

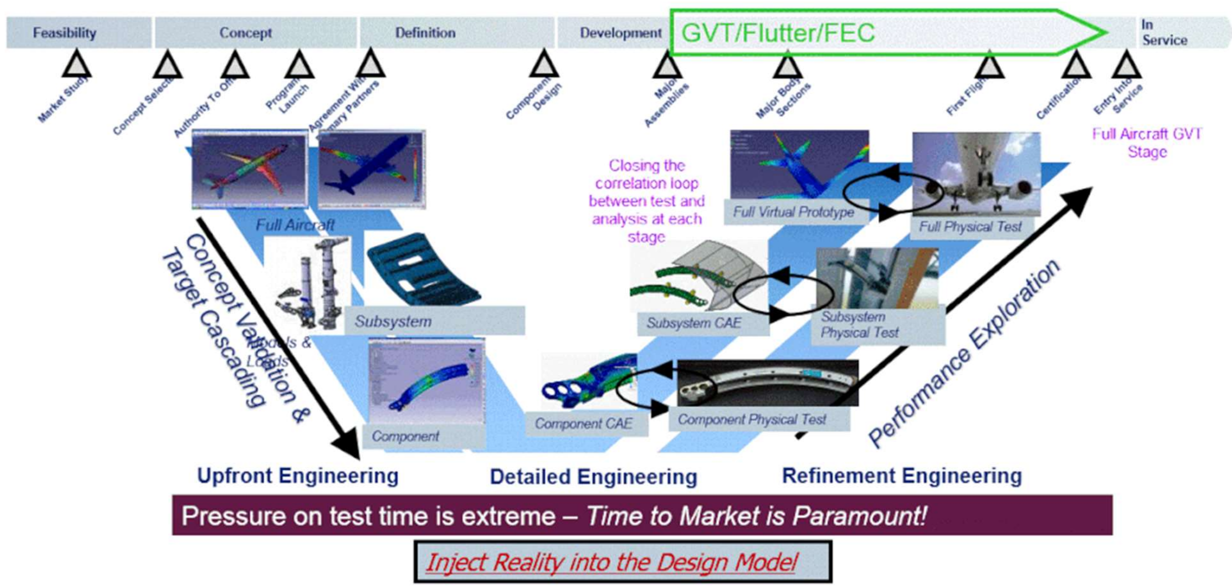


Figure 3: Building up to the GVT is depicted here in the Software Development Life Cycle (SDLC) "V" shaped diagram.

As time moves from left to right, the individual components and subsystems are modeled, and verified with test. Eventually the full aircraft GVT is performed, clearing the way for a safe "Flutter Envelope Clearance" (FEC) through flutter flight testing, and finally FAA type certification allowing the aircraft entry into commercial service.

If it is determined that structural responses can change significantly due to additional payload and fuel configurations, (*Figure 4*), then a GVT will be performed on that configuration. It is important to test any airplane configuration which can have an effect on flutter performance.

Figure 4: Depending on an aircraft, many different store and fuel configurations may be tested with GVT.

Variants can add significant time to the GVT test program, risking the planned first flight schedule.

GVT Data Acquisition: Excitation and Instrumentation

It is not uncommon for multiple exciters and hundreds of accelerometers to be used in a single GVT test.

Prior to testing, careful consideration of shaker and accelerometer placement is important. Some airplane bodies can be quite large and compliant, and require multiple exciters to obtain high quality modal data. Other aircraft bodies are smaller, but stiffer, so less exciters are required. Two exciters placed laterally and vertically on an aircraft engine are shown in *Figure 5*.

Figure 5: Lateral exciter (on left side) and vertical exciter (on bottom) for an aircraft engine.

For all cases it is important to provide sufficient excitation energy throughout the entire structure to get good signal-to-noise response. The various excitation signals available for GVT are given in *Figure 6*.

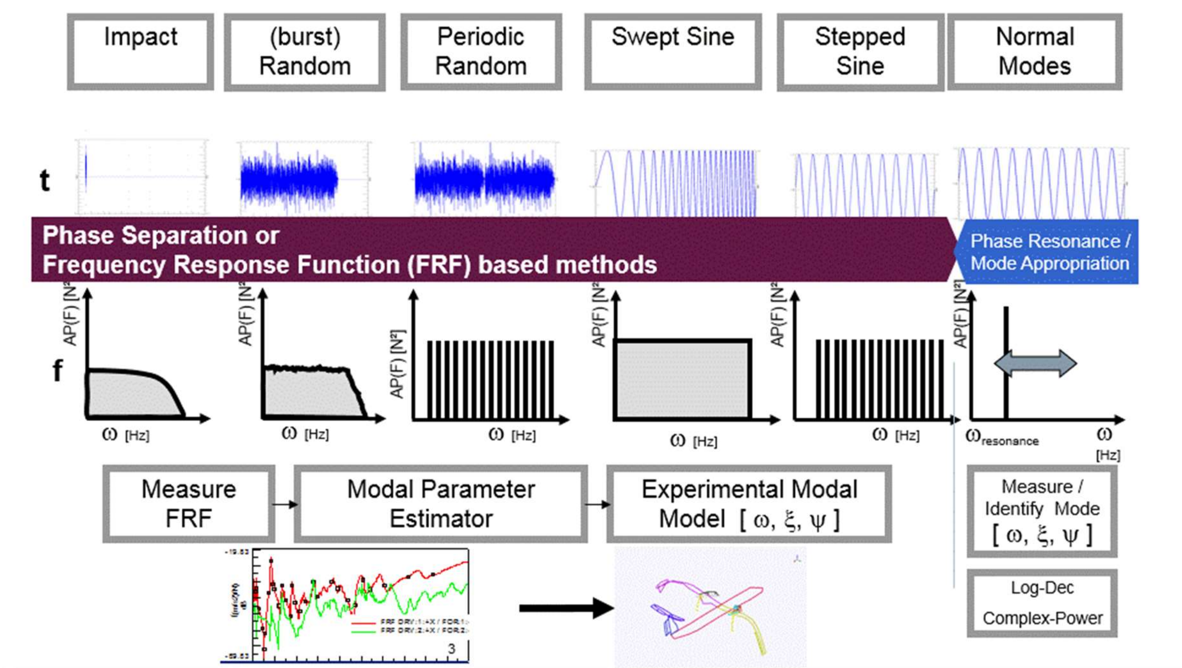


Figure 6: Different excitation signals are available for GVT. Some are sinusoidal and some are random in nature.

Typical excitation signals are burst random or swept sine, but for the critical modes important for the flutter predictions, other signals which are sinusoidal in nature are used.

Sinusoidal Methods: Stepped Sine and Normal Modes

Special cases of these sinusoidal signals are Stepped Sine and Normal Modes:

- Stepped Sine consists of a short dwell at each spectral line, stepping through the entire frequency range several times, based on the number of shakers.
- Normal Mode testing strives to tune a particular mode and isolate it as a single-degree-of-freedom response by managing the amplitude and phase of all exciter forces with respect to the acceleration responses at each driving point or response control point.

The Lissajous display is used to plot the Force vs Acceleration at a driving point location to verify the phases have a 90deg separation, which is an indication of a resonant condition. A driving point location is where both a force is applied, and a corresponding acceleration response is measured. In the case of a resonance, the lissajous graph will form a circle, as shown in *Figure 7*.

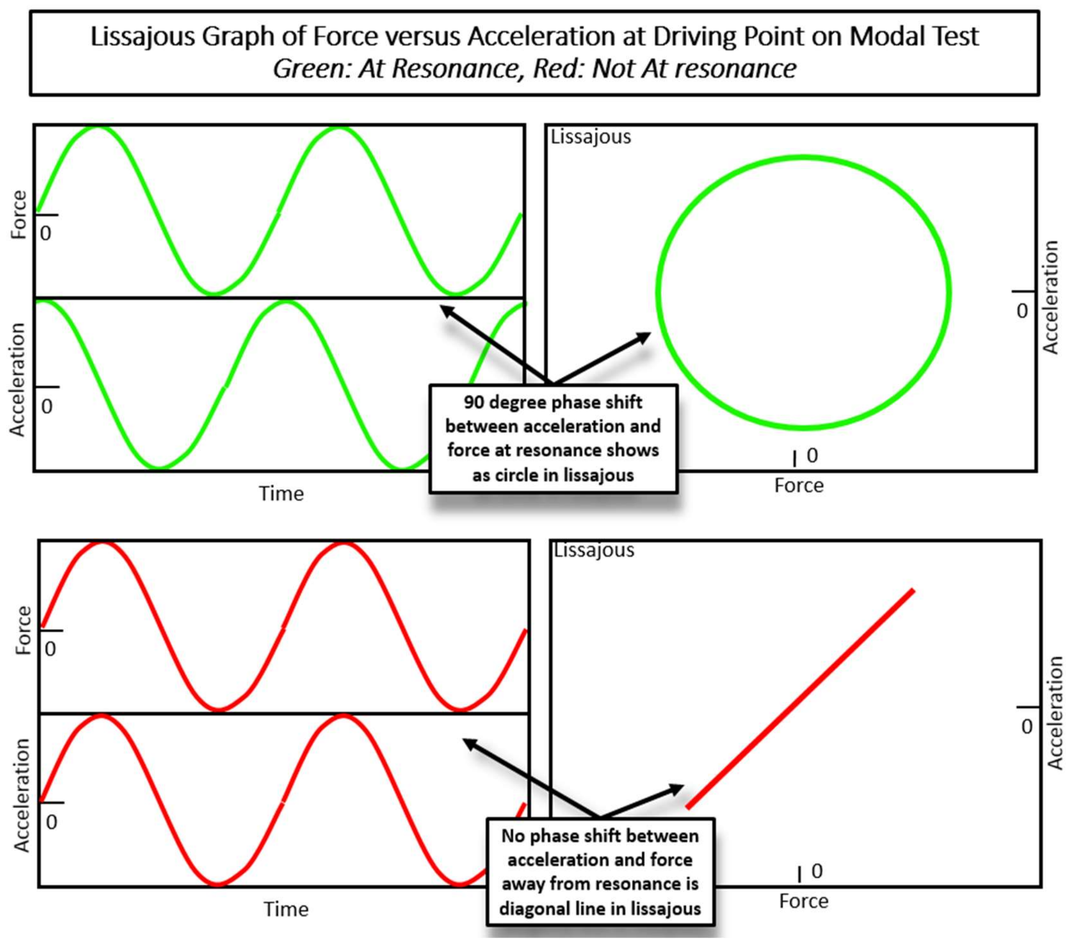


Figure 7: A Lissajous graph forms a circle at resonance (top, green), and is a diagonal line when not at resonance (bottom, red).

There are usually multiple exciters used, and the phase of the responses for each must be taken into account simultaneously. Since there are coupled responses between the exciters and responses, a matrix evaluation of the excitation phasing is required in Normal Modes.

In the "old days", this was done at a console by a highly skilled test operator who would look at multiple Lissajous displays, but nowadays the software programs automatically perform the tuning. An example of an online display for Normal Modes is shown in *Figure 8*. This can provide a lot of information simultaneously to verify the resonant conditions.

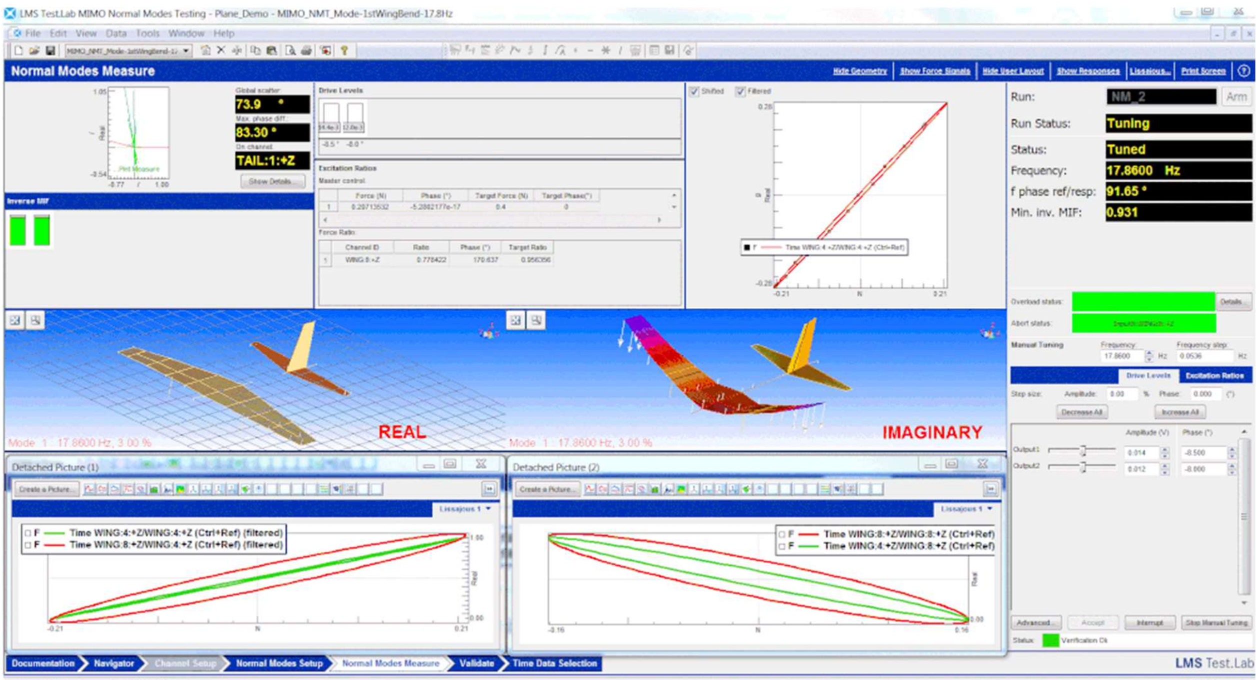


Figure 8: Normal Modes Screenshot with Lissajous displays (bottom)

Due to the time-consuming nature of these tests, normally only 2 or 3 modes may be chosen which are critical for flutter analysis. Varying the amplitudes of excitation allows an assessment of the linear range. Correctly identifying the modal parameters in the linear range, especially the damping, is essential for an accurate flutter prediction.

Normal Mode Testing and Damping

Note that with Normal Mode tuning, FRFs are not acquired. Instead mode shapes are calculated directly from the responses, and modal parameter estimation is not necessary. The modal damping is not contained in the Normal Mode results and must be obtained with an extra damping test as shown in *Figure 9*.

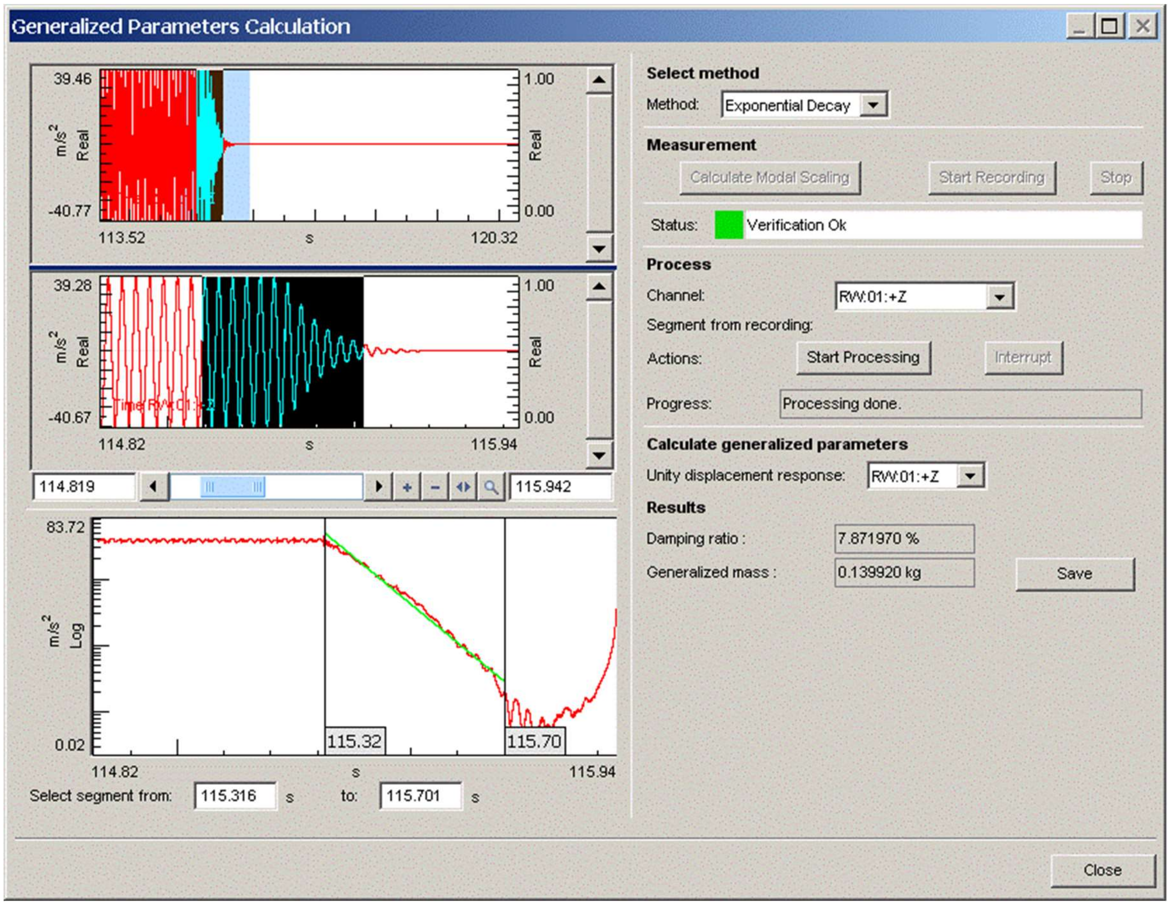


Figure 9: Damping calculation from Exponential decay

There are several techniques available in Simcenter Testlab to determine the modal damping, including:

- Log-decrement from an exponential decay using a Hilbert Transform
- Complex Power
- Force Quadrature
- SDOF Circle Fit

Pseudo-Random and Multi-Sines

A compromise between the accuracy obtained with sinusoidal tuning and dwelling, and the speed of the random type of signals is a hybrid approach in the class of "Multi-sines". A technique that is getting more popular is the Pseudo-random, but with a couple of twists to the classical version. With classical pseudo-random the randomized phases can have a constructive and destructive effect on the amplitudes, possibly causing overloads. If the phases are managed better with the so-called Schroeder sines, then the sinusoidal interactions can produce the best SNR like the sinusoidal signals, and still be comparable to the speed of the random testing.

GVT Data Processing: Modal Analysis

Using FRFs collected by either broadband random, sine approaches, or Stepped Sine FRF data, the general modal model is calculated using a "curve-fitting" parameter estimator. A "curve-fitter" estimates resonant frequencies, modes, and damping from the FRF data. An example of an extracted mode shape is shown in *Figure 10*.

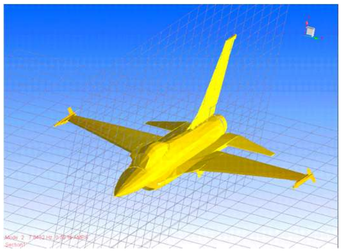


Figure 10: Mode shape of an F-16 calculated from FRFs collected during GVT test

As mentioned, the modal data for the Normal Modes method is a direct outcome from the mode tuning.

The modal parameters from the test modes are first evaluated at the test site by comparing with the expected results from computer based Finite Element Analysis (FEA) predictions. This post-test assessment is done to make sure there are not any blatant differences before the test is disassembled. FRF's and mode shapes are compared between FEA and Test. These can be directly overlaid in displays by importing the NASTRAN or ANSYS FE Model results directly into Testlab. For more information on this please see the community article Viewing Nastran results in Testlab.

In addition, a Modal Assurance Criteria (MAC) Matrix can be calculated to quantify the amount of correlation between the set of test and analysis mode shapes. An example of a MAC Matrix is shown in *Figure 11*. Note the important first several modes show good correlation. A value of 1 along the diagonal means there is perfect correlation of the mode shapes being compared.

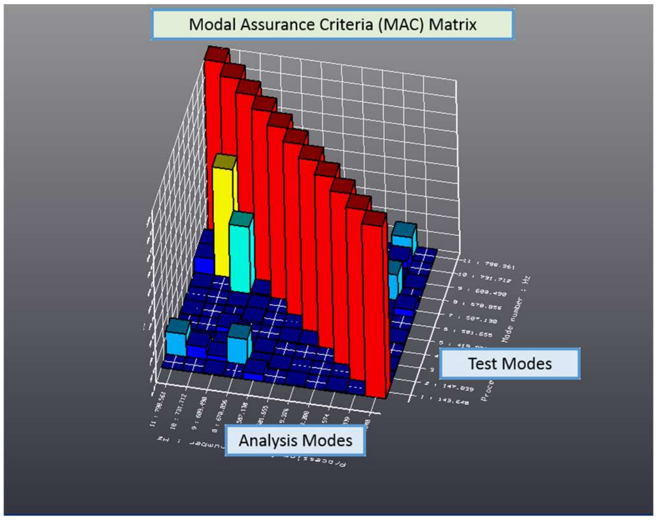


Figure 11: MAC Matrix which is used to visualize the amount of correlation between two sets of mode shapes. In this case it is the Test modes versus the Analytical modes.

As mentioned, the purpose of the Modal test is to obtain a modal model that can be compared with the FE model and update the FE model appropriately. Ideally the test model should be a “Real” mode, meaning the modal model represents a structure with proportional damping throughout, which is a compromise with the FE model since damping can only be estimated. To impose this constraint on the test modal model, an extension to the Simcenter Testlab modal analysis is available called MLMM. This method will iterate on the calculated modal parameters to optimize the modal model with the intent towards the Real modes and driving towards a better fit of the synthesized FRF’s.

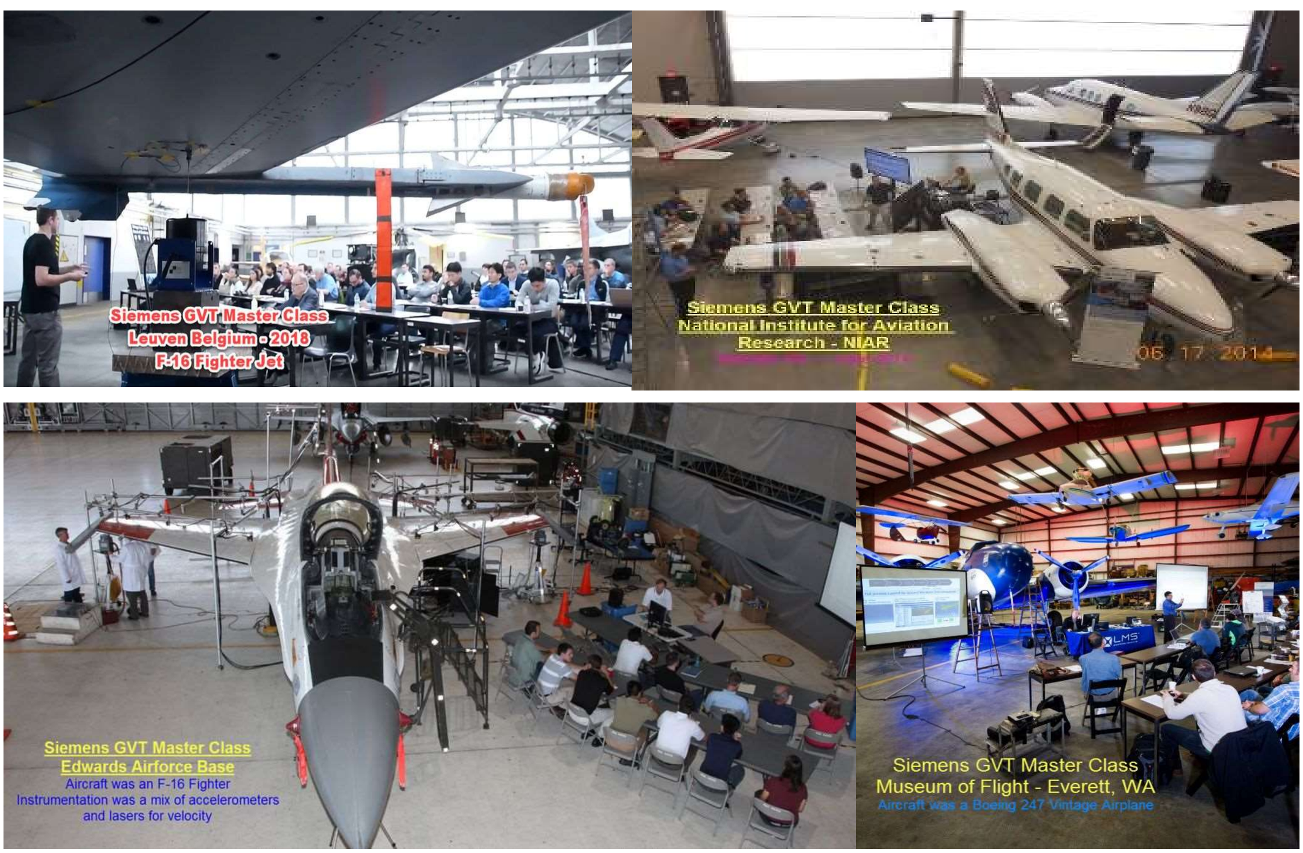
If there are discrepancies in the results from test compared with analysis, then the FE model goes through an iteration process to update the model and bring it in line with the test results. Of course, the test must be a good representation of the configuration of the analysis model in terms of mass properties and boundary conditions. The rigid body modes should not interfere with the flexible structural modes, and this is usually checked as part of the test procedure. Once a validated FE model is obtained then it can be used for analysis which continues with flutter predictions.

A nice way to view the mode shapes is with the CAD model to expand the mode shapes for a better understanding of the modes through Mode Shape Expansion.

Conclusion

Together, Ground Vibration Tests and Wind Tunnel Tests are used for verification of the structural and aerodynamic models, which when combined will result in more accurate flutter predictions. The flutter predictions are still only computer model simulations, while flight flutter testing is the final verification that the flight envelope is clear of unwanted flutter dynamics.

If you have any questions about this please contact our GTAC Support , or post a comment to this article.



Multi Input Multi Output MIMO Testing

Multi Input Multi Output MIMO Testing

When we speak of MIMO Testing, this can have different meaning to different people. In the general sense it means the vibration, or noise, input from multiple drive signals to an exciter system in an MDOF configuration, with multiple measured output responses from a fixture, test item, or acoustic volume in an MDOF configuration. The general concept is depicted in Figure #1.

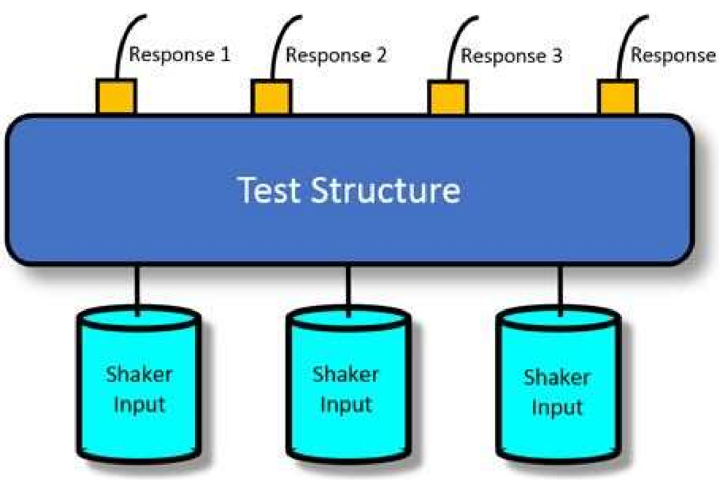


Figure 1: A MIMO system with 3 inputs and 4 outputs. Test results will have 12 transfer functions.

The resulting success criteria can be defined 2 ways, in terms of time, or frequency responses. Where the time domain version is called Time Waveform Replication, (MIMO TWR), for this article we will focus on the frequency domain version, of which there are various applications based on the sinusoidal or random nature of the input and corresponding output signals.

The applicable test scenarios regarding MIMO testing are the following:

1. Structural Testing for FRF Measurements in Open-Loop Mode
2. Environmental Testing with Closed-Loop Control
3. Acoustic Testing with Closed-Loop Control

We will address the first 2 structural scenarios in this article, with a separate article discussing the acoustic aspect of MIMO testing.

When it comes to MIMO vibration testing, the proper excitation of the structure is very important. For structural Frequency Response Function FRF measurements, one wants to make sure that all modes of interest are excited, and the FRF estimation is able to properly separate and assess the signal coming from multiple exciters, to form the matrix of FRF's. This will allow the determination of modes when there's a small separation in frequency, called "closely spaced modes".

For the case of Environmental testing, there are standards that prescribe the response levels that are required. The total response at each control point can be due to the multiple excitation sources, each with an independent contribution to the response. The amount of excitation energy from each source must be extracted from the response signal at each control point by the control system, so each source can be independently updated by the controller to maintain the proper response at the control points.

Structural Testing

For the Structural FRF Test scenario, this has to do with the objective of transfer function measurements, typically in the context of a modal test.

Some common modes of excitation for transfer function measurements are depicted in Figure #2, and listed here:

- Burst Random Testing
- Pseudo-random Testing
- Swept and Stepped Sine Testing

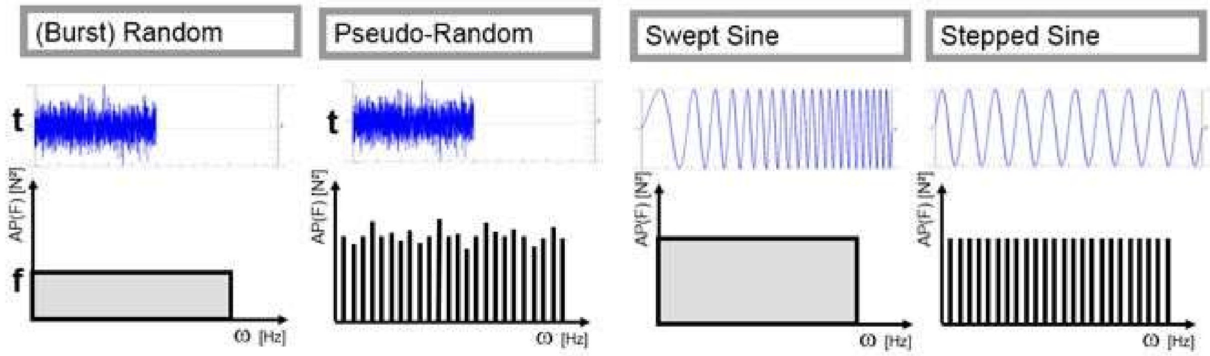


Figure 2: Time and frequency domain representations of different MIMO excitation signals used in FRF Measurements.

These types of tests are open-loop tests, where the output levels are established ahead of the test and are not controlled as feedback from the responses to the controller system.

There are however limit levels that are continuously checked in real-time as the test progresses to ensure these are not exceeded. So, the control may not necessarily be a closed-loop test, but the safety checks are.

Another aspect of exciting a structure with Multiple-Inputs concerns the extraction of the modal parameters (the ultimate goal for such a test) from the matrix of measured FRF's. To calculate the FRF's, the independent excitation energy from each source, is extracted from each response. For extraction of each source signal from the overall response to be possible, the excitation signals must be decorrelated. This is not trivial, since the shakers are connected to the same structure which creates a certain degree of correlation. There are functions available in Simcenter Testlab to assess this decorrelation in the form of Principle Component Analysis (PCA), these functions can be displayed during the test to verify that the number of independent sources detected in the responses, match the physical number of independent sources. If there is sufficient decorrelation, then the quality of the FRF data improves, which eventually results in a higher quality modal parameter estimation. It is imperative to assess the quality of the FRF as it is being acquired, and viewing different functions such as the PCA can help in the assessment.

Environmental Testing

For the Environmental Testing scenario, the responses from the test article must adhere to a level which is typically established from published standards, or can be measured data from the field in operational conditions, and the frequency spectrum representation of this data must be replicated in the lab. This scenario requires a closed-loop controller, which can imply either a single shaker or multi-shaker case. The MIMO Random version of this type of test is shown with a 3DOF shaker system in Figure #3.

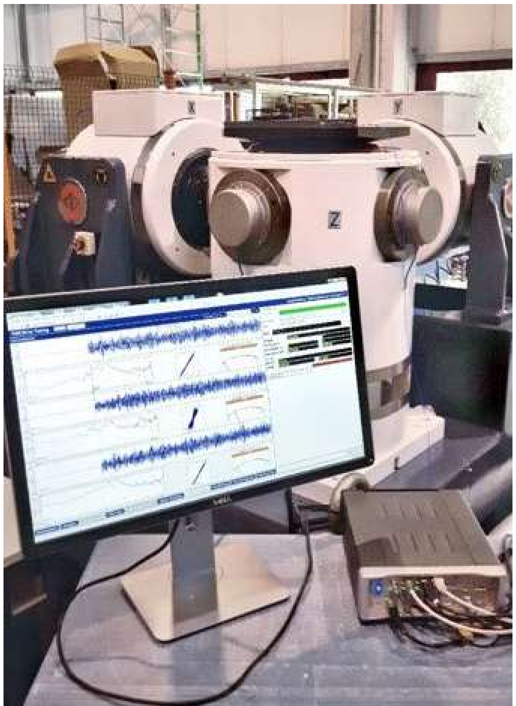


Figure 3: A MIMO closed loop vibration control test using a 3DOF shaker and the Simcenter Testlab MIMO Random Controller.

There can be many different varieties of environmental testing, and can be categorized as either Single-exciter or Multi-exciter tests. For the single input version please see the related article, “What is Vibration Control Testing?” For this current article we will focus on the Multi-exciter case, for which there can also be several different versions to the strategy used. Some of the most common terms related to environmental testing regarding this are summarized here.

- MIMO Multi Input (>1 Drive) Multi Output (>1 Control point)
- SIMO Single Input (=1 Drive) Multi Output (>1 Control point)
- MESA (Multi Exciter (>1 Shaker) Single Axis)
- MEMA (Multi Exciter (>1 Shaker) Multi Axis)

There can be many different reasons and objectives for choosing one of these versus another, or sometimes a combination of these. For this article we will discuss the MEMA case, which is the most general form. A separate article will discuss the MESA case.

MEMA - Multi Exciters combined with multiple control points, in Multiple Axes

The multi-exciter multi-axis case is the most general form of a MIMO environmental test, and the most complex and difficult in terms of test setup and control. The upside though is this case probably is the best method for achieving the realistic responses throughout the DUT due to the operating conditions, which is the overarching goal of environmental testing. This is becoming more recognized in the environmental testing community, and at the same time the MIMO testing technology continues to evolve to support this direction.

There are a couple of different forms of MEMA testing, and we will discuss first the 3DOF shaker case, and then the more general form of MEMA testing with multiple shakers in any axis.

MEMA Test using 3DOF Shaker - For the case where simultaneous excitation of all 3 axes are required, the 3-Degree-of-Freedom (3DOF) shaker can be used, as shown in Figure #4. These are 3 single-axis shakers in a single system. Inside the center casing is an elaborate mechanical linkage whose goal is to allow the uncoupled response in the 3 orthogonal directions, so that if energy in any one single direction is provided, then the response in the other 2 directions is null or as close to zero as possible. To achieve this feat is extremely difficult, if not impossible to achieve, which means a “multi-axis” control strategy must be used in this case.

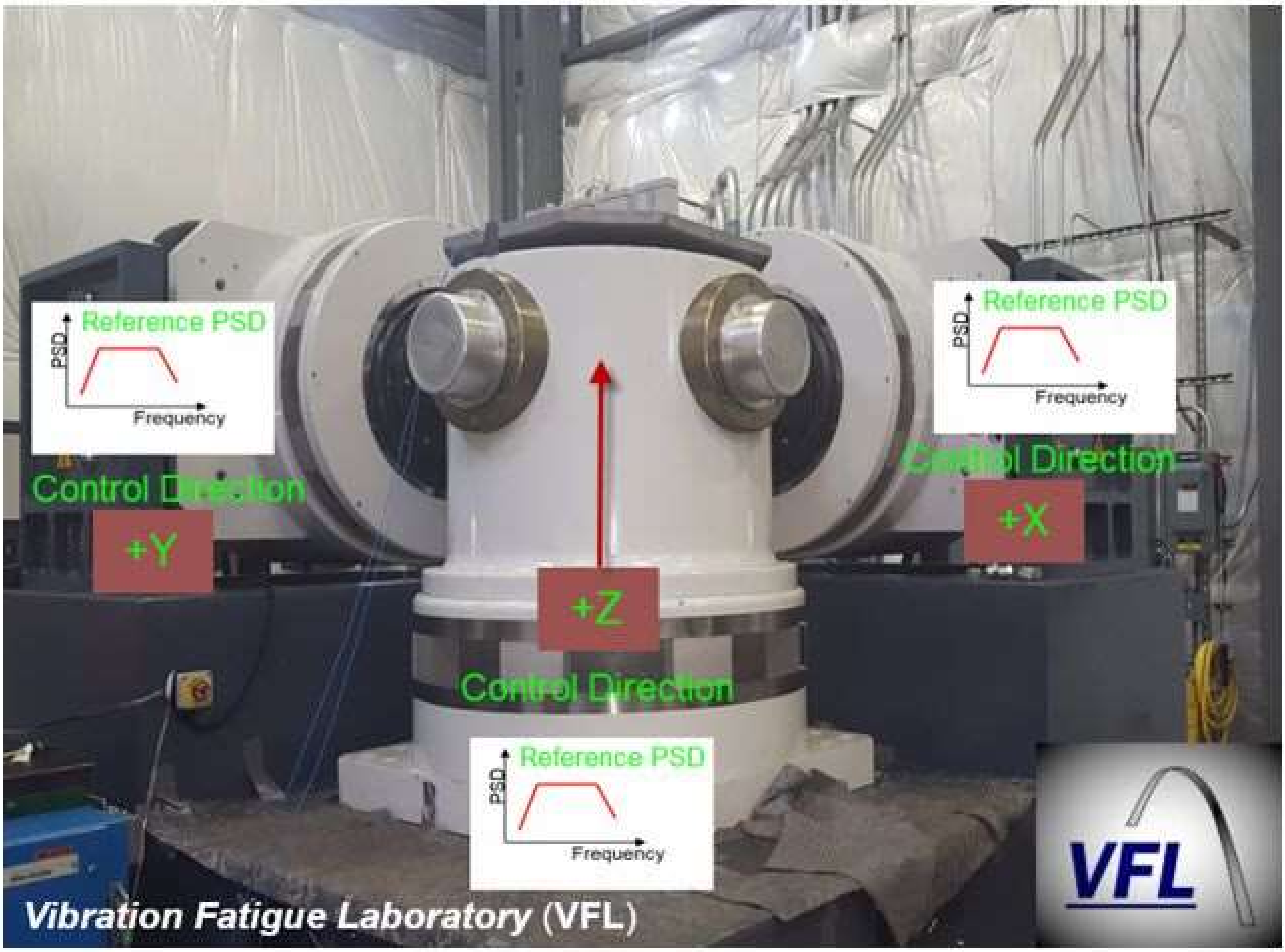


Figure 4: With a 3DOF Shaker, the 3 orthogonal axes can have the same or independent control spectrums. A Multi-axis controller is required due to the mechanical cross-coupling of the 3 shakers.

For the 3DOF shaker scenario, how the test specifications are determined is still a current hot topic of debate in the scientific community; however the easy solution is to use the reference spectrums that would normally be used in 3 sequential single-axis tests, and use these as the control targets for the 3DOF shaker test. The execution of this test can produce profoundly different results in terms of time and spectral response of the DUT at response points around and inside the structure, so the goal of getting the same results as the 3 sequential test scenario in reality cannot easily be achieved. However in the end, again the goal of representing real-life responses in operational conditions can be argued are better achieved with a 3DOF test vs the 3 sequential tests. This can be quantified by evaluating the durability results from 3 sequential qualification tests versus a single 3DOF test. There have been many studies to show this, and some results are discussed further in this article.

There is also a business case to support the use of simultaneous excitation using the 3DOF shaker, which combines 3 sequential tests into a single test, as depicted in Figure #5. However this must also be weighed with the much higher cost of such a system. An extension of the 3DOF test would also include the rotational degrees of freedom with a 6DOF shaker system, again expanding on the complexity of the shaker system and controller.

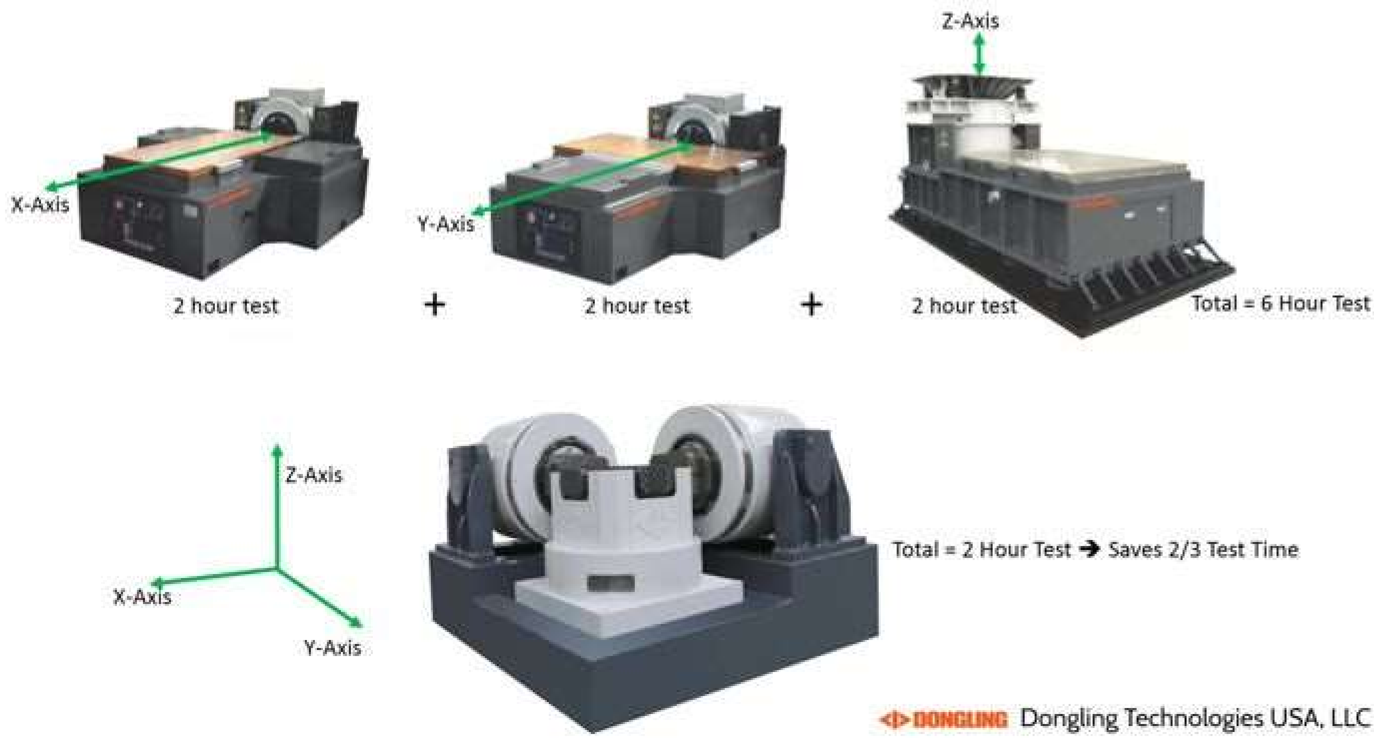


Figure 5: Combining 3 sequential tests into a single 3-axis test can provide justification for the cost of a 3DOF Controller and Shaker.

MEMA Test using Multiple Shakers in any axis - This is the most general form of MIMO closed-loop control testing. An extreme case of this is give in the Figure #6.

Figure 6: Multiple Exciters applying excitation energy in several different axes at many points on the structure allow a better distribution of energy to achieve better results in terms of responses around the structure. This case was to simulate the excitation forces due to aerodynamic loading.

The case studied here was to reproduce the responses around the scaled missile, by replicating the aerodynamic excitation that it would see in real operating conditions. This particular case studied was also comparing the results of a MEMA test with what is typically achieved with 3 sequential single-axis (SIMO) tests, and to show how different they can be. The small white disks are piezo-exciters. The interaction (coupling) of these among each other was characterized through some measurements, and the information was used in the closed-loop control. In the extreme case where there is a controlled excitation source for each of the multiple control responses, this would be the best in terms of response replication. However there are of course practical limitations to this, but since there would be some coupling between the excitation sources, this can be used to an advantage, Musella: Optimizing the Drives in a MIMO Control Test . By utilizing these "coherent sources", it can be easier to achieve the desired responses. This concept is very well explained in the corresponding technical paper, (Enhanced ground-based vibration testing for aerodynamic environments – P.M. Daborn, P.R. Ind, D.J. Ewins) , and will be expanded upon in the next section.

MIMO closed-loop Testing Strategy

Boundary Conditions

As has been already mentioned in this article, and a fundamental aspect of dynamic environmental testing, is that the support and suspension and the corresponding boundary conditions in the lab must replicate, or be as close as possible, to what is given in the real scenario from which the responses were derived, or at least be free enough to allow the replication of the responses from the exciters. This is a key factor in determining if a successful test is even possible. Another way to put it is that the responses that need to be reproduced in the lab with artificial excitation must be "Physically Realizable", given the test setup.

Excitation Sources

Another consideration are the exciters themselves. The means by which energy enters the system or Device Under Test (DUT) in real life conditions also should be replicated as close as possible in the lab. The artificial excitation energy is normally input to the structure through electro-dynamic, or piezo-electric shakers. Understanding the modal properties of the structure being tested can be key to determining the optimal exciter input locations. The input energy from multiple excitation sources can have an influence on each other, and this interaction is what is called **coupling,** (**Figure #7**).

Figure 7: A structural test on aircraft engine with two modal shaker exciters. During the test, these exciters will probably exhibit coupling.

The interaction of the excitation sources also has influence on the responses at points around the structure. The amplitude of the responses at the control points can be due to the interaction of multiple sources and the amplitudes of the excitation source as a function of frequency. This is a complex interaction, meaning it is not only the amplitude of the source interactions, but the phasing of the sources and how this influences the response amplitudes.

For example, if the amplitudes from two sources are in phase with each other at a point on the structure, or in an acoustic volume for the acoustic case, then there is "Constructive Interference", and the amplitudes can add with each other. If they are partially or completely out of phase with each other, then they will have "Destructive Interference" with each other. (Figure 8) This complex interaction is frequency dependent, and can be characterized through the Cross-Spectrum function, and is measured during the System ID phase of a test.

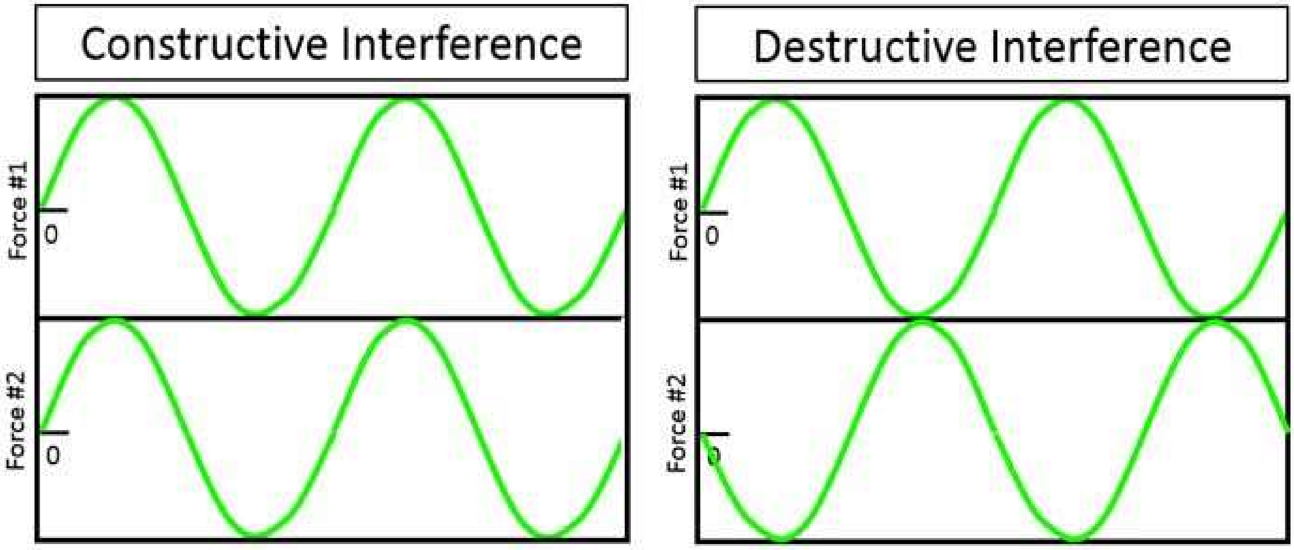


Figure 8: Constructive versus Destructive Interference of the Exciter Forces

Adding more exciters to the system adds to the complexity of this interaction, and this interaction can be quantified by the Coherence function. So when the term Coherent Sources are used, that simply means there is some level of complex interaction of two sources with each other at some point where the interaction occurs. This other point being referred to could be either a control point, or a response point. If there are multiple coherent sources acting at a control point, then the amount of energy which is output from a single source can be reduced to obtain the desired response. The interaction of the sources could be "Incoherent", meaning completely out of phase, (Coh = 0), "Partially Coherent", (coherence 0<Coh<1, or "Fully Coherent", (Coh=1). This interaction is what can amplify or attenuate the response at the points on the test structure, and can be taken advantage of with the controller. On the other hand, if there are coherent sources acting at a point which is not controlled or monitored, then the response can be greater than what is desired.

Getting back to the boundary conditions, the locations of the source points, and the coherence between the sources, can all have profound effects on the responses that need to be produced at the control points around the structure. You can also gain an understanding on how these can have an effect on the "Physical Realizability" and hence the success of the test. The best-case scenario for the lab setup of a MIMO Control test is where the boundary conditions are the same, or as close as possible, as in real conditions which occurred during the data measurement. This also increases the chances of a successful test.

For some environmental tests the control point can be located at the base of the rigid support structure where the shaker table attaches to the structure. This can be used as the target response point and using the spectrum as control in each direction of a 3DOF shaker test, as shown in Figure #9.

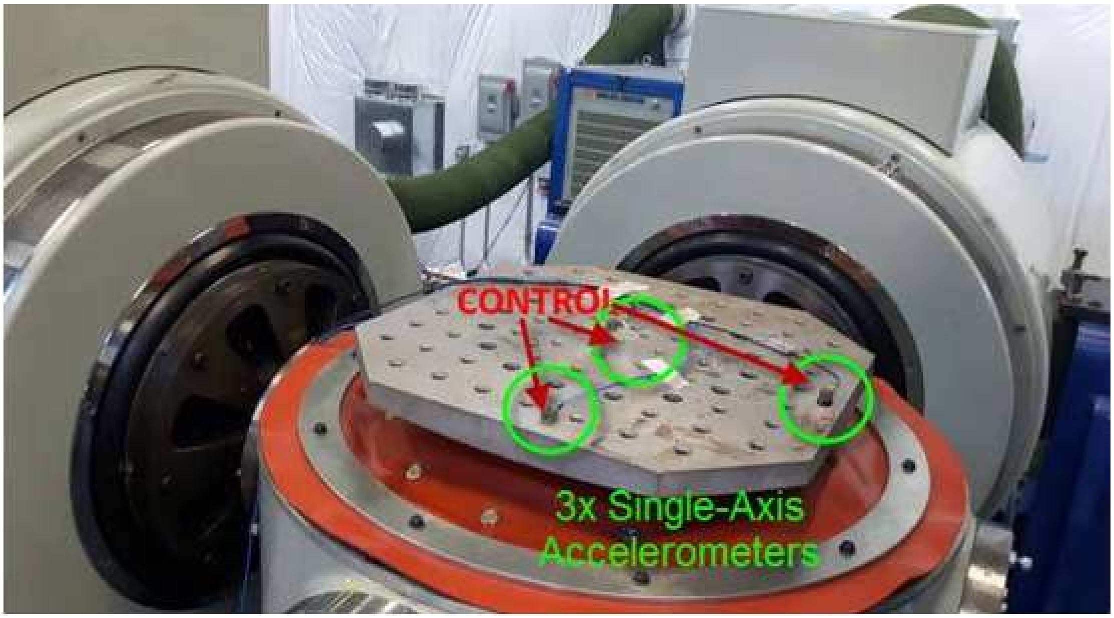


Figure 9: For the 3DOF setup, a single-axis accelerometer can be used for each axis, or a Triaxial accel can be positioned at the center of the shaker head expander. A MIMO Controller is required when cross-coupling of the exciter forces occur.

MIMO Control Targets

Measuring the transfer function matrix of the different drive voltage sources relative to the response points is done prior to the test and is called the “System ID” phase of the test. During this step, the controller ‘gets to know’ the dynamics of the system (amps+shakers+DUT) and measures (and saves) the System Transfer Function matrix necessary to compute the drive signals to the different shakers. For this reason it is important that the FRF matrix which is obtained is as decorrelated as possible (in mathematical terms this makes the FRF invertible).

Regarding the online control of the test, just like in a Single-axis test, the success criteria of a MEMA test is based on the feasibility of the response targets; however, unlike a single axis-test, the target level is no longer a single Autopower PSD spectrum vector but a whole Spectral Density Matrix (SDM) which includes the Cross-Spectral-Density (CSDs) between each control channel, (Figure #10). These CSDs capture the complex interaction of the different control points vs frequency. During such a MEMA test, it is therefore crucial to set CSD targets which are physically achievable. To respond to this need, the CSD’s are measured and saved prior to the MIMO test during the System ID phase described above.

$ASD_{xx}(f)$	$CSD^*_{xy}(f)$	$CSD^*_{xz}(f)$
$CSD_{xy}(f)$	$ASD_{yy}(f)$	$CSD^*_{yz}(f)$
$CSD_{xz}(f)$	$CSD_{yz}(f)$	$ASD_{zz}(f)$

3x3 Reference Matrix

Figure 10: The target Spectral Density Matrix (SDM) is defined as the target Autopowers at the control points and corresponding Crosspowers between each control point. The SDM components are measured prior to the start of the control test during the SysID phase and are also used as target spectra.

The Diagonal AP spectrum (PSDs) are the targets that are defined by the requirement spec, as would be done for a single-axis control test. The off-diagonal terms are the Crosspowers as measured during the System ID phase (and which therefore capture the dynamics of the system being tested). These combined AP and XP functions are related and also define the spectral Coherence (Figure #11) between the different control points through the equation:

$$\gamma^2{}_{0ij}(k) = \frac{\left|\overline{S}_{ij}(k)\right|^2}{\overline{S}_{ii}(k) \times \overline{S}_{jj}(k)}$$

Figure 11: The Coherence Function. The control of the target points considers the AP's and also the XP's of each control point with every other control point. These spectrums are what make up the Spectral Density Matrix, and are related to the coherence.

During a control test, the response APs and XPs are measured and updated through averaging. Using the system transfer function from the SysID phase, the controller will quantify the errors of the target and measured SDM, and update the drives as necessary. Since the targets for the test are all considered equally in the SDM matrix, there is equal weighting of these target functions and the drives are updated to reduce the errors in a least-squares sense between the target SDM and measured SDM.

Why MIMO Testing?

There are several reasons why multi-exciters would be needed:

- Insufficient Force from Single Shaker - When the force from a single shaker is not sufficient
- Multiple Point Control Profile - When there are multiple control points with its own reference profile
- Multi-Direction – When needing to excite simultaneously in multiple directions.

The underlying objective is to replicate the real-world responses in a structure given differing boundary conditions in a laboratory setting.
Single axis environmental testing has evolved in its technology and use since World War II when the military started to mandate suppliers provide these tests to qualify the products they were providing. The short-comings of this type of test continues to be recognized for not providing realistic test conditions compared to the real conditions a part or system would see in real life operation. Single axis testing, in many cases simply does not replicate the forces that parts would experience in real-world conditions. It has been shown that multi-axis testing can bring about failures in a shorter amount of time due to the interaction and cross-axis coupling of the excitation forces.

There are several references given below on the studies done to show this. For example, an automotive engine, like many industrial parts, experiences in-use forces from multiple directions simultaneously. Modeling under such complex conditions would predict fatigue and failure modes that are impossible to simulate with only one vibration direction or, for that matter, by sequential testing along different axes of the part. Testing with 3DOF, or even 6DOF, provides the best approach to mimic reality, by producing the off-axis contributions that may expose the greatest weakness in the part.
With 3DOF testing, this can also provide the additional benefit of efficiency. With the multi-inputs and phases, the overall testing time can usually be reduced by at least a factor of three, compared with the sequential testing in three directions, and also often leading to a decrease in power input required to accomplish the task.

Some companies are starting to grasp these concepts, and this type of testing is proving beneficial. Until recently, the problem has been these test rigs have not been easily created. With several companies now creating these rigs, the testing in MIMO (3 DOF) will accelerate. Hopefully the sharing of knowledge, and benefits of this testing will soon be passed on and accepted by the testing community.

Conclusion

The ultimate goal for any environmental qualification test is to reproduce the realistic responses that are seen in the real operational conditions. In a lot of cases this can only be achieved with MIMO testing, and there is definitely a growing trend and interest in the test community in this direction. When there is a significant amount of coupling between the inputs to the system, then a MIMO Control strategy is required.
There is also a lot of on-going work in the area of defining the target responses required for MIMO Control testing. The goals of the testing must be well defined to determine what is feasible for the test setup and what the success criteria should be. Lots of care is required in defining the test configuration and the physical setup of a MIMO test in order to ensure the responses are "Physically Realizable" given the test setup.

The vibration testing community is nowadays living a technology shift: providers of excitation systems (shaker, loudspeakers) are providing more advanced systems for MIMO excitation, and in parallel, the vibration controllers are evolving to supply test engineers with the tools to carry out such tests. And most importantly, standards are being updated to reflect these technological and scientific advances. Of course, each aspect of a MIMO test has its own challenge, including the test engineers themselves: MIMO tests are a lot more complex (physically and mathematically) to comprehend and execute than the ubiquitous single-axis vibration tests. But it is clear that MIMO tests are the future of dynamic environmental testing.

The standards that have been written to support the MIMO Testing are:

- MIL-STD-810G METHOD 527 - Multi-Exciter Testing
- IEST-RP-DTE022: MULTI-SHAKER TEST AND CONTROL (Recommended Practice)

All Articles - Siemens PLM Community Testing Knowledge Base

Siemens PLM Community

Digital Signal Processing

- What is Fourier Transform?
- Digital Signal Processing: Sampling Rates, Bandwidth, Spectral Lines, and more...
- Gain, Range, Quantization
- Aliasing
- Overloads
- Averaging Types: What's the difference?
- Kurtosis
- Time-Frequency Analysis: Wavelets
- Spectrum versus Autopower
- Autopower Function...Demystified!
- Power Spectral Density
- Shock Response Spectrum (SRS)
- Windows and Leakage
- Window Types
- Window correction factors
- Exponential Window Correction Factors
- RMS Calculations
- The Gibbs Phenomenon
- Introduction to Filters: FIR and IIR

Modal Data Acquisition

- Simcenter Testlab Impact Testing
- Modal Impact Testing: User Defined Impact Sequence
- What modal impact hammer tip should I use?
- Modal Tips: Roving hammer versus roving accelerometer
- Modal Testing: Driving Point Survey – What's at stake?
- Attach Modal Shaker at Stiff or Flexible Area of Stucture?
- Using Pseudo-Random for high quality FRF measurements
- Multi Input Multi Output MIMO Testing
- Ground Vibration Testing and Flutter

Modal Analysis and Operational Deflection Shapes

- Natural Frequency and Resonance
- What is Frequency Response Function (FRF)?
- The FRF and it Many Forms!
- How to calculate damping from a FRF?
- Getting Started with Modal Curvefitting
- Modal Assurance Criterion
- Simcenter Testlab Modal Analysis: Modification Prediction
- Import CAD into Simcenter Testlab
- Animate CAD Geometry
- Alias Table: Mapping Test Data to Geometry
- Geometry in Simcenter Testlab
- Maximum Likelihood estimation of a Modal Model (MLMM)
- Difference Between Residues and Residuals?
- What is an Operational Deflection Shape (ODS)?
- Making a Video of a Time Animation

Acoustics

- History of Acoustics
- Sound Pressure
- What is a decibel?
- What is A-weighting?
- Octaves and Human Hearing
- What is Sound Power?
- Decibel Math
- What is the Acoustic Quantity called Q?
- Sound Absorption
- Sound Transmission Loss
- Noise level certification, how to select the right standard?
- Sound pressure, sound power, and sound intensity: What is the difference?
- Sound fields: Free and diffuse field, near and far field

Sound Quality

- History of Acoustics
- Sound Pressure
- What is a decibel?
- What is A-weighting?
- Octaves and Human Hearing
- Loudness and Sones
- Calculating a Specific Loudness Spectrum in Simcenter Testlab
- How to Calculate N10 Time Varying Loudness in Simcenter Testlab?
- Articulation Index
- Auditory Masking

- Tone-to-Noise Ratio and Prominence Ratio
- Fluctuation Strength and Roughness
- Critical Bands in Human Hearing
- Kurtosis
- Time-Frequency Analysis: Wavelets
- Sound Quality Jury Testing
- Alter Speed of Rotating Sound for Listening
- Simcenter Testlab Sound Diagnosis

Durability

- History of Fatigue
- Stress and Strain
- Calculating Damage with Miner's Rule
- What is a SN-Curve?
- Rainflow Counting
- Difference between 'Range-Mean' and 'From-To' Counting
- Power Spectral Density
- Shock Response Spectrum (SRS)
- How to Calculate a Shock Response Spectrum with Testlab?
- Some Thoughts on Accelerated Durability Testing
- Goodman-Haigh Diagram for Infinite Life
- Measuring strain gauges in Simcenter Testlab
- Rosette Strain Gauges
- Calculating Damage in Simcenter Tecware Process Builder
- Strain Gauges: Selecting an Excitation Voltage
- Simcenter Testlab SCADAS and Long Strain Cables

Rotating Machinery

- What is an order?
- Torsional Vibration: What is it?
- Zebra Tape Butt Joint Correction for Torsional Vibrations
- Balancing: Static, Coupled, and Dynamic
- Removing Spikes from RPM Signals
- Using the Smoothing function to remove rpm spikes
- RPM Calculation problems and Laser Tachometers
- Simcenter Testlab Signature
- Fixed Sampling versus Synchronous Sampling
- Tips and tricks for acquiring torsional orders
- Harmonic Removal
- Interpreting Colormaps
- Cycle to Cycle Averaging in Simcenter Testlab
- Speed Sweep Data Processing: RPM Increment, Framesize, Sweep Rate, and Overlap
- Order Cuts: How to Get the Correct Amplitude?

Vibration Control

- What is Vibration Control Testing?
- Vibration Control FAQs
- Power Spectral Density
- Shock Response Spectrum (SRS)
- Sine Control: Estimation Methods
- Sine Control: Closed Loop Control Parameters
- Sine Control: Notching
- Vibration Control: Understanding Selfcheck
- Direct Field Acoustic Noise Testing
- What is Total Harmonic Distortion (THD)?
- Kurtosis
- "Excessive DC Level" message
- ITF Warning in Random Vibration Control

Simcenter Testlab General

- Simcenter Testlab Overview
- Simcenter Testlab Active Picture
- Simcenter Testlab Active Picture Plug-in Download
- Simcenter Testlab Token Licensing: What is it? How does it work?
- Simcenter Testlab: Import/Export to Excel
- Simcenter Testlab: Documentation Tab
- Simcenter Testlab: Templates
- Simcenter Testlab: Keyboard and Mouse Shortcuts
- Simcenter Testlab: CTRL-PageUp and CTRL-PageDown to change worksheets
- Simcenter Testlab: How to create a custom workflow
- How to lockdown a test in Simcenter Testlab
- Simcenter Testlab: Format Based Printing with Powerpoint
- Simcenter Testlab: Replace Data Origin...
- Simcenter Testlab: Batch Reporting
- Nastran and Simcenter Testlab: Viewing Mode Shapes and FRFs
- Import ANSYS Geometry and Mode Shapes in Simcenter Testlab

Simcenter Testlab Display tips:

- Simcenter Testlab: Display Layouts
- Simcenter Testlab: Customized Legend Text
- Simcenter Testlab: Curve Linestyles
- Simcenter Testlab: Cursor Tips and Tricks
- Simcenter Testlab: Coupled Cursors and Coupled Limits
- Simcenter Testlab: Interactive order cursor!
- Simcenter Testlab: Order Cursor vs Time in Colormap
- Simcenter Testlab: The "Double X cursor"
- Simcenter Testlab: Automatic Peak-Valley Cursor

- Simcenter Testlab: Harmonic Cursors
- Simcenter Testlab: XY Graph
- Simcenter Testlab: Keyboard and Mouse Shortcuts
- Simcenter Testlab: Matrix-Heatmap Display
- Simcenter Testlab: Replace a curve in a display directly?
- Simcenter Testlab: Overlay Mode Shapes
- Simcenter Testlab: Compare data to target quickly with preview mode

Simcenter SCADAS and other Acquisition Hardware

- Simcenter SCADAS Mobile and Recorder Hardware
- Simcenter SCADAS XS: Everything you need to know!
- Simcenter SCADAS Cable Guide
- Simcenter Sound Camera
- Simcenter Soundbrush
- Qsources: Acoustic and Structural Exciters
- Simcenter Testlab Smart App Favorites!
- Single Ended versus Differential Inputs
- AC versus DC Coupling
- How to use SCADAS T8 Thermocouple card?
- ICP versus IEPE versus Charge Accelerometers
- Long Cable Lengths and ICP Transducers
- Long cable lengths and strain gauges

Simcenter Testlab Acquisition Tips

- Simcenter Testlab Signature
- Hands-free acquisition (almost)
- Capturing Transient Events in Simcenter Testlab
- Cool Channel Setup tricks for Triaxial Accelerometers
- Copy Channel Setup from One Project to Another?
- View Channel Setup from Desktop Navigator!
- Data Acquisition Based on GPS Location
- How to calculate a VECTOR SUM
- Tips and tricks for acquiring torsional orders
- CAN Bus Measurements
- No more tach! Use OBDII to get engine and vehicle speed…
- Record Raw CANBUS Data and Decode Offline!
- Single Plane Balancing in Simcenter Testlab
- Measuring strain gauges in Simcenter Testlab
- Correcting a bad calibration factor
- Simcenter Testlab Calibration

Simcenter Testlab Time Signal Calculator

- Simcenter Testlab: Time Signal Calculator Tips

- Simcenter Testlab: Calculating Statistics from Time Histories
- Simcenter Testlab: Appending Time Histories
- Simcenter Testlab: Create a new recording from a segment cut
- Simcenter Testlab: Applying an arbitrary filter shape to a time history
- Simcenter Testlab: Integrating a Time History and Avoiding Offsets
- Analyzing PWM Signals with Time Signal Calculator
- Introduction to Filters: FIR vs IIR
- Applying a Custom Filter Shape: FILTER_FRF command
- Filter an Order and Listen to It
- Generate Custom Signals in Time Signal Calculator
- Removing Spikes from RPM Signals
- Harmonic Removal
- Rosette Strain Gauges

Simcenter Testlab Processing Tips

- Simcenter Testlab Throughput Processing Tips
- RPM Extraction in Simcenter Testlab
- Run Comparison in Simcenter Testlab
- Order Sections: Setting Parameters
- Time Signal Calculator Tips
- Simcenter Testlab: Calculating Statistics from Time Histories
- Simcenter Testlab: Appending Time Histories
- Simcenter Testlab: Create a new recording from a segment cut
- Simcenter Testlab Sound Diagnosis
- Can I view data in Google Earth?
- Analyzing PWM Signals in Simcenter Testlab

Simcenter Tecware

- Running Simcenter Tecware with Simcenter Testlab tokens
- Simcenter Tecware: Open Time Files fast!
- Simcenter Tecware: Overlay multiple signals in X and Y display
- Simcenter Tecware: Access to different file formats
- Simcenter Tecware: Spike Filter
- Simcenter Tecware: Removing Multiple Segments
- Simcenter Tecware: Calculate Infinite Life with the Goodman Diagram
- Calculating damage with Simcenter Tecware Processbuilder
- Butterworth filter "regular" and "zero phase"