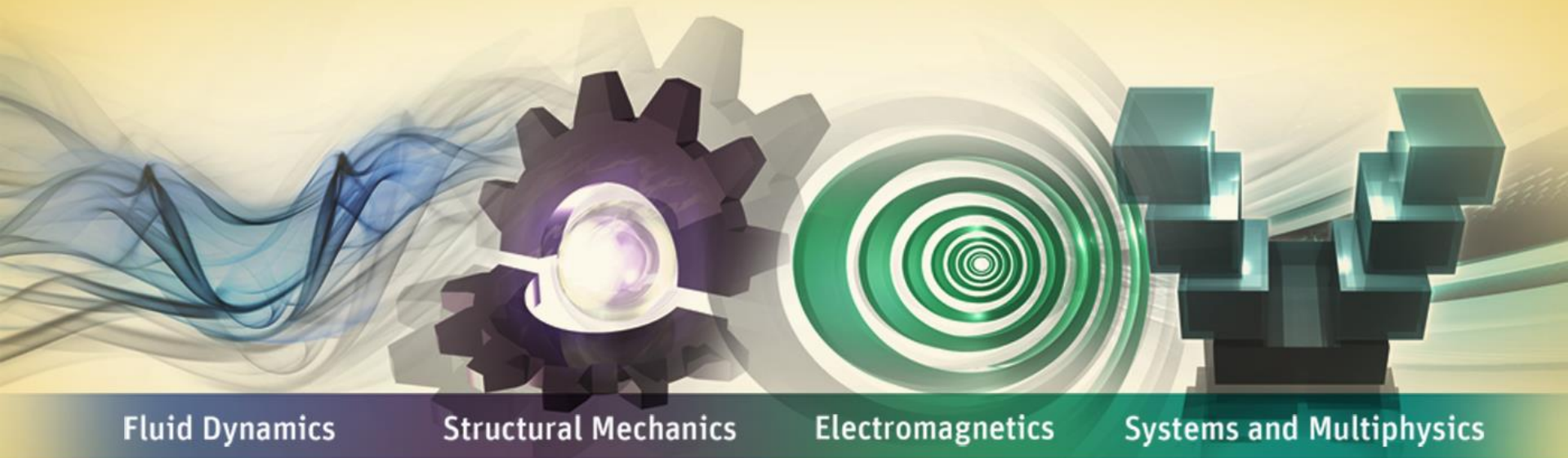


Shock & Vibration using ANSYS Mechanical



Kelly Morgan
ANSYS Inc.

1. Introduction to Dynamic Analysis

2. Types of Dynamic Analysis in ANSYS

- **Modal**
- **Harmonic**
- **Transient**
- **Spectrum**
- **Random vibrations [PSD]**

3. Case Study: Half-Sine Shock on a PCB model

Definition & Purpose

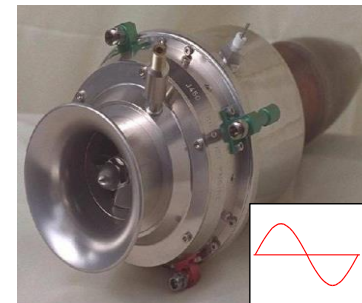
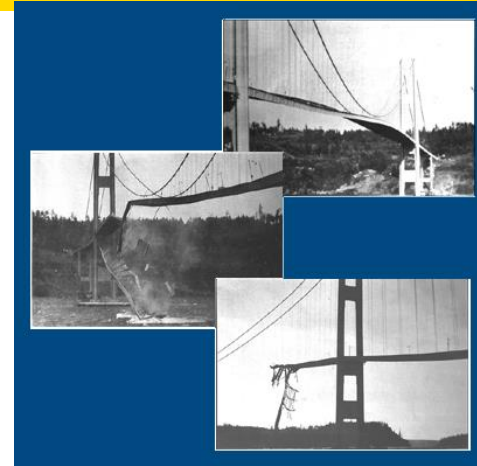
Determine the dynamic behavior (inertia and possibly damping) of a structure or component.

Why not just Static? It is quick!

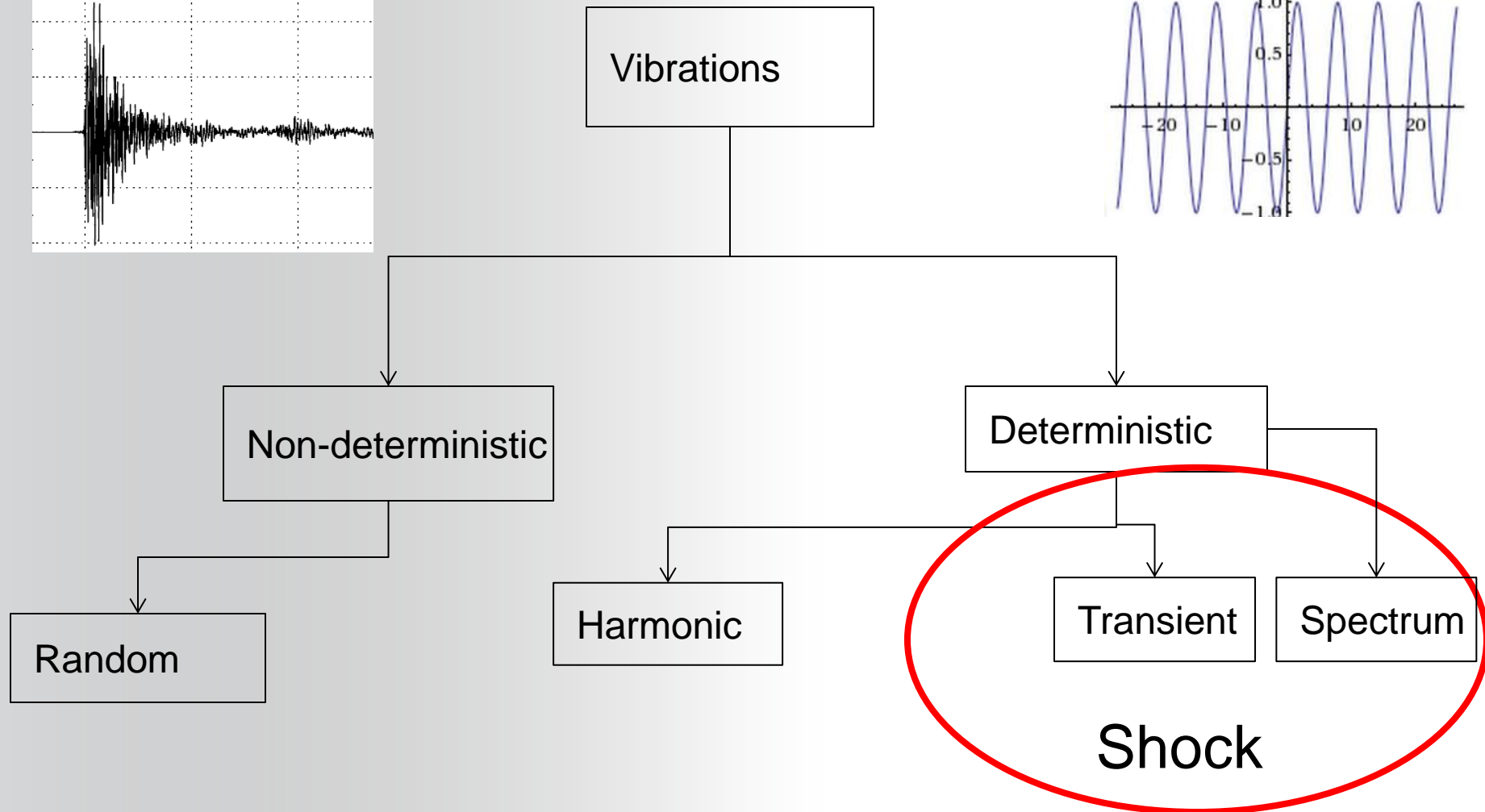
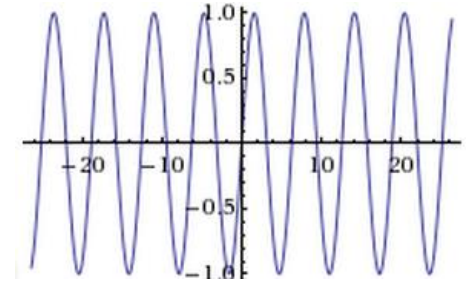
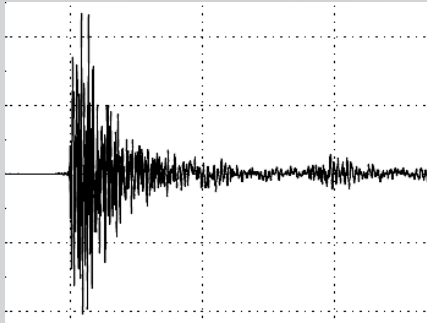


What is “Dynamic behavior”?

- **Vibration characteristics**
 - how the structure vibrates and at what frequencies
- **Effect of harmonic loads.**
- **Effect of seismic or shock loads.**
- **Effect of random loads.**
- **Effect of time-varying loads.**



Types of Vibrations



Types of Dynamic Analysis

Type	Input	Output
Modal	<ul style="list-style-type: none">• Prescribed BCs/none	<ul style="list-style-type: none">• natural frequencies and corresponding mode shapes• stress/strain profile
Harmonic	<ul style="list-style-type: none">• sinusoidally-varying excitations across a range of frequencies	<ul style="list-style-type: none">• sinusoidally-varying response at each frequency• min/max response over frequency range
Spectrum	<ul style="list-style-type: none">• spectrum representing the response to a specific time history	<ul style="list-style-type: none">• maximum response if the model were subjected to the time history
Random	<ul style="list-style-type: none">• spectrum representing probability distribution of excitation	<ul style="list-style-type: none">• response within specified range of probabilities
Transient	<ul style="list-style-type: none">• time-varying loads	<ul style="list-style-type: none">• time-varying response

Equation of Motion

The linear general equation of motion, which will be referred to throughout this course, is as follows (matrix form):

$$[M]\{\ddot{u}\} + [C]\{\dot{u}\} + [K]\{u\} = \{F\}$$

$[M]$ = structural mass matrix $\{\ddot{u}\}$ = nodal acceleration vector

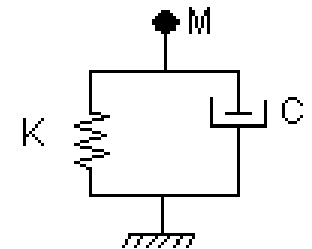
$[C]$ = structural damping matrix $\{\dot{u}\}$ = nodal velocity vector

$[K]$ = structural stiffness matrix $\{u\}$ = nodal displacement vector

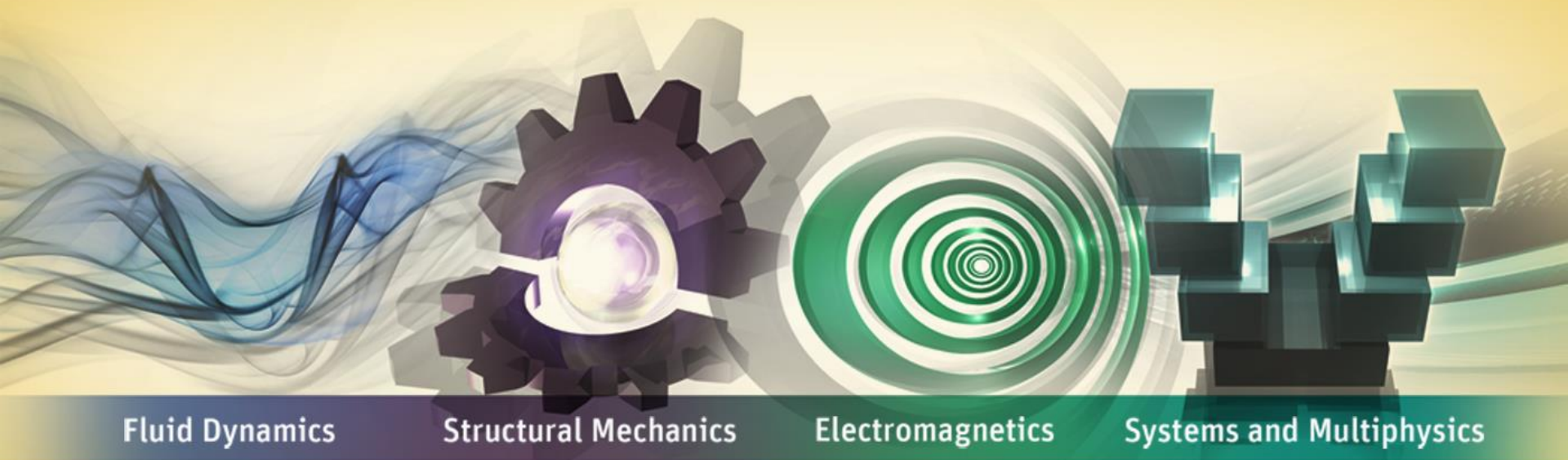
$\{F\}$ = applied load vector

Note that this is simply a force balance:

$$\overbrace{[M]\{\ddot{u}\}}^{F_{\text{inertial}}} + \overbrace{[C]\{\dot{u}\}}^{F_{\text{damping}}} + \overbrace{[K]\{u\}}^{F_{\text{stiffness}}} = \overbrace{\{F\}}^{F_{\text{applied}}}$$



Modal Analysis



Description & Purpose

A modal analysis is a technique used to determine the vibration characteristics of structures:

- **natural frequencies**
 - **at what frequencies the structure would tend to naturally vibrate**
- **mode shapes**
 - **in what shape the structure would tend to vibrate at each frequency**
- **mode participation factors**
 - **the amount of mass that participates in a given direction for each mode**

Most fundamental of all the dynamic analysis types.

Description & Purpose

Benefits of modal analysis

Allows the design to avoid resonant vibrations or to vibrate at a specified frequency (speaker box, for example).

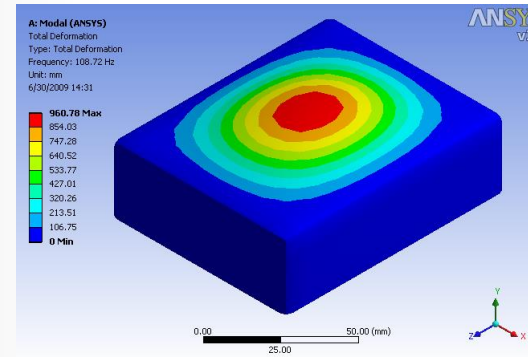
Gives engineers an idea of how the design will respond to different types of dynamic loads.

Helps in calculating solution controls (time steps, etc.) for other dynamic analyses.

Recommendation: Because a structure's vibration characteristics determine how it responds to any type of dynamic load, it is generally recommended to perform a modal analysis first *before* trying any other dynamic analysis.

Eigenvalues & Eigenvectors

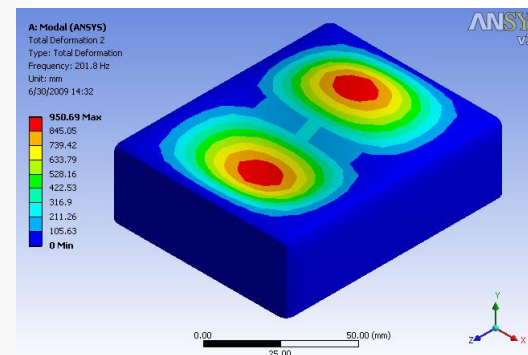
- The square roots of the eigenvalues are ω_i , the structure's natural circular frequencies (rad/s).
- The eigenvectors $\{\phi\}_i$ represent the mode shapes, *i.e.* the shape assumed by the structure when vibrating at frequency f_i .



mode 1

$\leftarrow \{\phi\}_1$

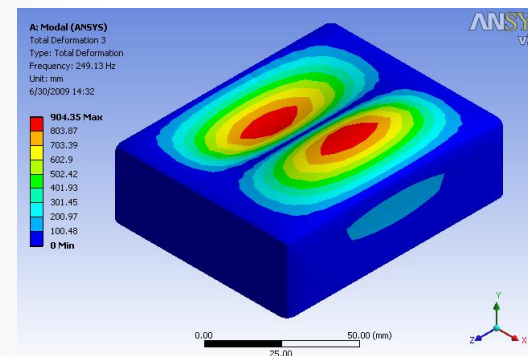
$f_1 = 109 \text{ Hz}$



mode 2

$\leftarrow \{\phi\}_2$

$f_2 = 202 \text{ Hz}$



mode 3

$\leftarrow \{\phi\}_3$

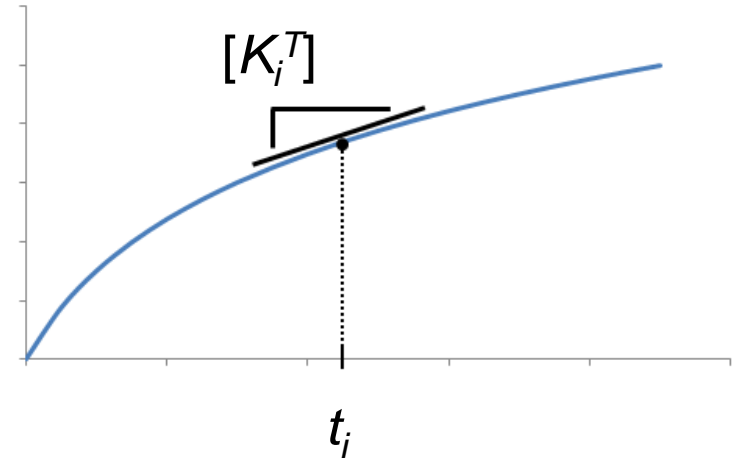
$f_3 = 249 \text{ Hz}$

Tangent Stiffness matrix

$$K_i^T = K_i^{Mat} + K_i^{Contact} + K_i^{LoadStiffness} + K_i^{SpinSoftening} + K_i^{StressStiffness}$$

Material Stiffness $[K_i^{Mat}]$

- For nonlinear materials, only the linear portion is used
- For hyperelastic materials, the tangent material properties at the point of restart are used



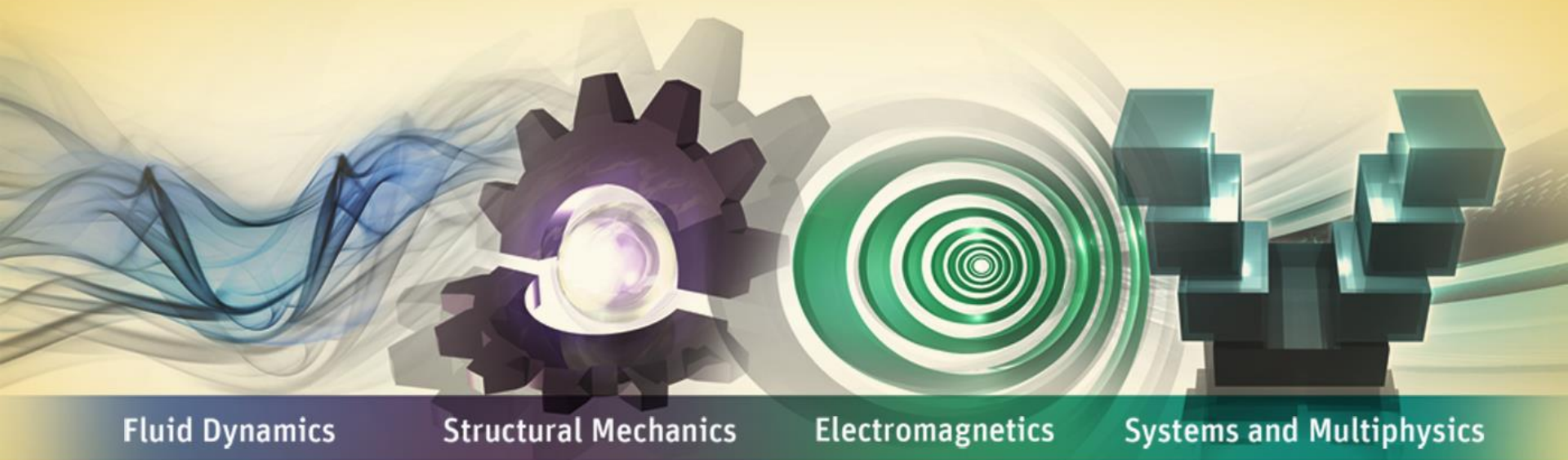
Stress Stiffening $[K_i^{StressStiffness}]$ / Spin softening $[K_i^{SpinSoftening}]$

- Effects included automatically in large deflection analyses

Contact stiffness $[K_i^{Contact}]$

- Contact behavior can be changed prior to the modal analysis

Harmonic Analysis



Definition & Purpose

What is harmonic analysis?

A technique to determine the steady state response of a structure to sinusoidal (harmonic) loads of known frequency.

Input:

- **Harmonic loads (forces, pressures, and imposed displacements) of known magnitude and frequency.**
- **May be multiple loads all at the same frequency. Forces and displacements can be in-phase or out-of phase. Body loads can only be specified with a phase angle of zero.**

Output:

- **Harmonic displacements at each DOF, usually out of phase with the applied loads.**
- **Other derived quantities, such as stresses and strains.**

... Definition & Purpose

Harmonic analysis is used in the design of:

Supports, fixtures, and components of rotating equipment such as compressors, engines, pumps, and turbomachinery.

Structures subjected to vortex shedding (swirling motion of fluids) such as turbine blades, airplane wings, bridges, and towers.

Why should you do a harmonic analysis?

To make sure that a given design can withstand sinusoidal loads at different frequencies (e.g, an engine running at different speeds).

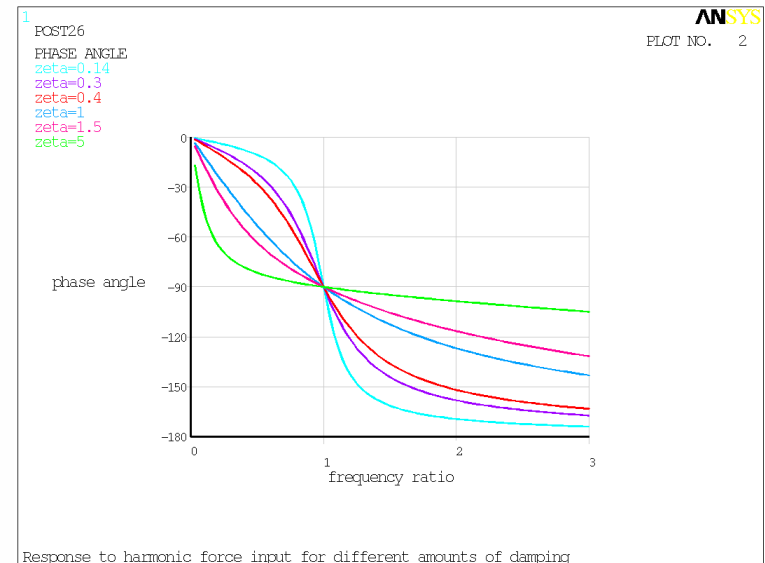
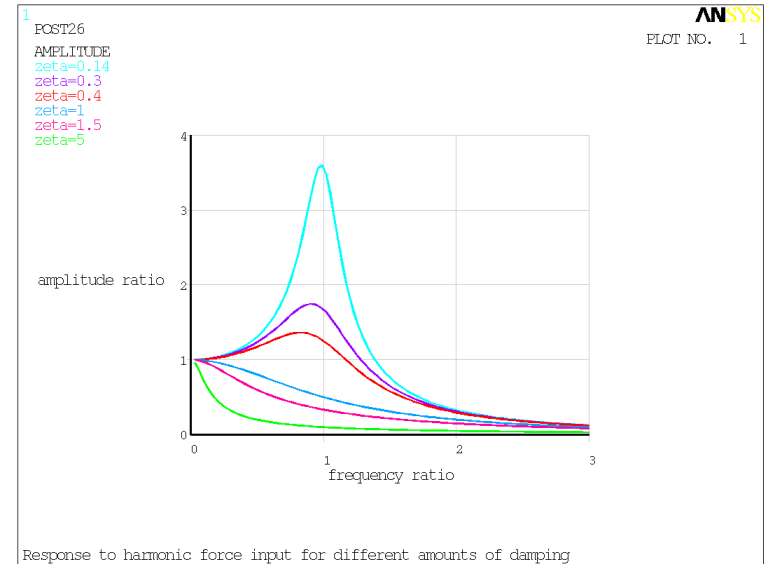
To detect resonant response and avoid it if necessary (by using dampers, for example).

When the imposed frequency approaches a natural frequency in the direction of excitation, a phenomenon known as resonance occurs.

- This can be seen in the figures on the right for a 1-DOF system subjected to a harmonic force for various amounts of damping.

The following will be observed:

- an increase in damping decreases the amplitude of the response for all imposed frequencies,
- a small change in damping has a large effect on the response near resonance, and
- the phase angle always passes through $\pm 90^\circ$ at resonance for any amount of damping.



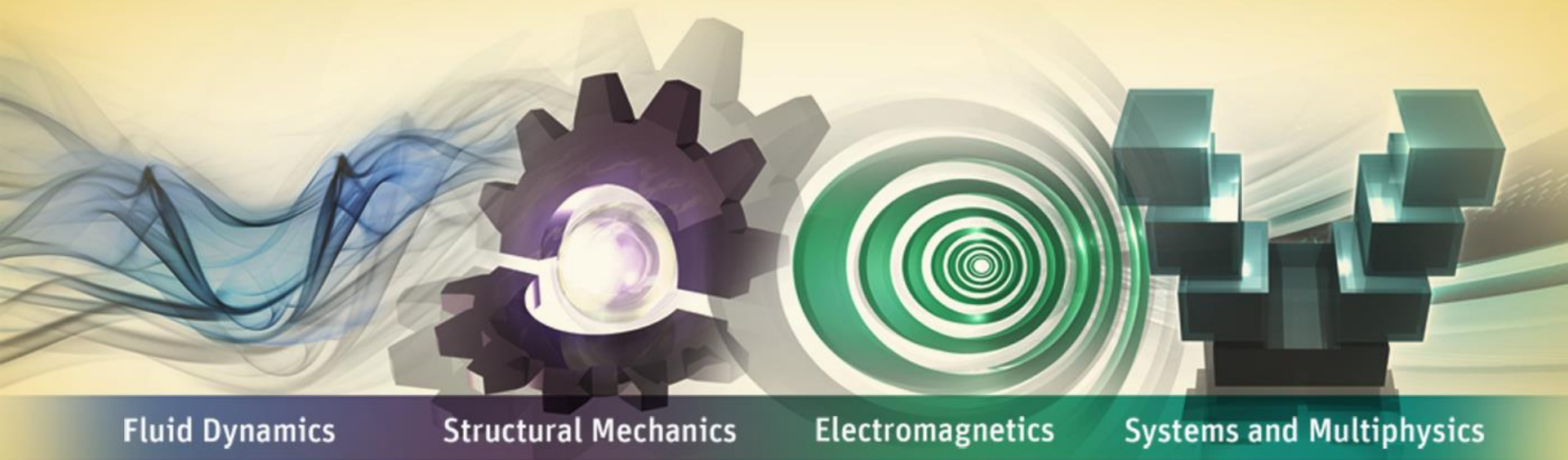
FULL

- Exact solution.
- Generally slower than MSUP.
- Supports all types of loads and boundary conditions.
- Solution points must be equally distributed across the frequency domain.
- Solves the full system of simultaneous equations using the Sparse matrix solver for complex arithmetic.
- Prestressing is available for Harmonic Response in Workbench.

MSUP

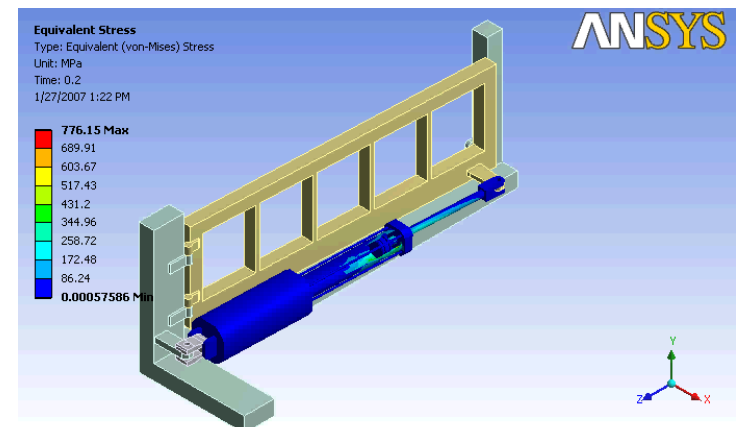
- Approximate solution; accuracy depends in part on whether an adequate number of modes have been extracted to represent the harmonic response.
- Generally faster than FULL.
- Support nonzero imposed harmonic displacements through Enforced Motion
- Solution points may be either equally distributed across the frequency domain or clustered about the natural frequencies of the structure.
- Solves an uncoupled system of equations by performing a linear combination of orthogonal vectors (mode shapes).

Transient Analysis



Transient structural analyses are needed to evaluate the response of deformable bodies when inertial effects become significant.


- If inertial and damping effects can be ignored, consider performing a *linear or nonlinear static analysis* instead
- If the loading is purely sinusoidal and the response is linear, a *harmonic response analysis* is more efficient
- If the bodies can be assumed to be rigid and the kinematics of the system is of interest, *rigid dynamic analysis* is more cost-effective
- **In all other cases, *transient structural analyses* should be used, as it is the most general type of dynamic analysis**

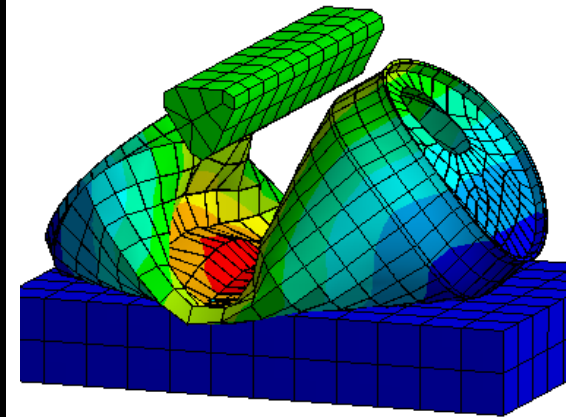


Assembly shown here is from an Autodesk Inventor sample model

Implicit vs Explicit Dynamics?

“Implicit” and “Explicit” refer to two types of time integration methods used to perform dynamic simulations

Solution	Impact Velocity (m/s)	Strain Rate (/s)	Effect
Implicit 		$< 10^{-5}$	Static / Creep
	< 50	$10^{-5} - 10^{-1}$	Elastic
	50 - 1000	$10^{-1} - 10^1$	Elastic-Plastic (material strength significant)
	1000 - 3000	$10^5 - 10^6$	Primarily Plastic (pressure equals or exceeds material strength)
	3000 - 12000	$10^6 - 10^8$	Hydrodynamic (pressure many times material strength)
Explicit	> 12000	$> 10^8$	Vaporization of colliding solids



Implicit vs Explicit Dynamics

Contact

- Implicit dynamics
 - All contacts must be defined prior to solve
- Explicit dynamics
 - Non-linear contacts do not need to be defined prior to solve

Materials

- Explicit dynamics generally supports more material failure models than implicit dynamics.

Preliminary Modal Analysis

Use automatic time-stepping, proper selection of the initial, minimum, and maximum time steps is important to represent the dynamic response accurately:

- Unlike rigid dynamic analyses which use explicit time integration, transient structural analyses use *implicit time integration*. Hence, the time steps are usually **larger** for transient structural analyses
- The dynamic response can be thought of as various *mode shapes* of the structure being excited by a loading.
- It is recommended to use automatic time-stepping (default):
 - The maximum time step can be chosen based on accuracy concerns. This value can be defined as the same or slightly larger than the initial time step
 - The minimum time step can be input to prevent Workbench Mechanical from solving indefinitely (1/100 or 1/1000 of the initial time step)

... Preliminary Modal Analysis

A general suggestion for selection of the initial time step is to use the following equation:

$$\Delta t_{initial} = \frac{1}{20 f_{response}}$$

where $f_{response}$ is the frequency of the highest mode of interest

In order to determine the highest mode of interest, a preliminary modal analysis should be performed prior to the transient structural analysis

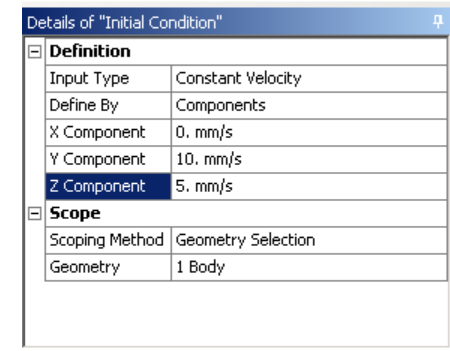
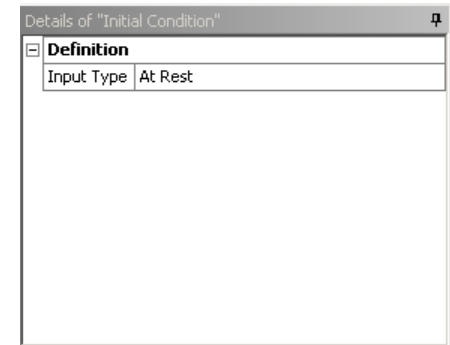
- mode shapes of the structure are known
- value of $f_{response}$ value determine
- It is a good idea to examine the various mode shapes to determine which frequency may be the highest mode of interest contributing to the response of the structure.

For a transient structural analysis, initial displacement and initial velocity is required:

- User can define initial conditions via “Initial Condition” branch or by using multiple Steps

Defining initial displacement & velocity with the “Initial Condition” object:

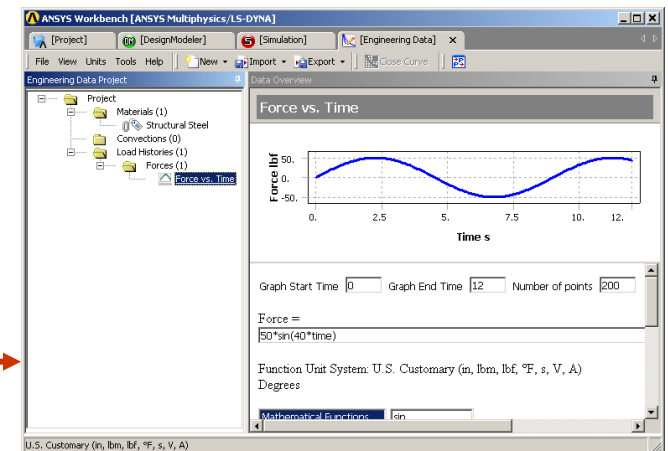
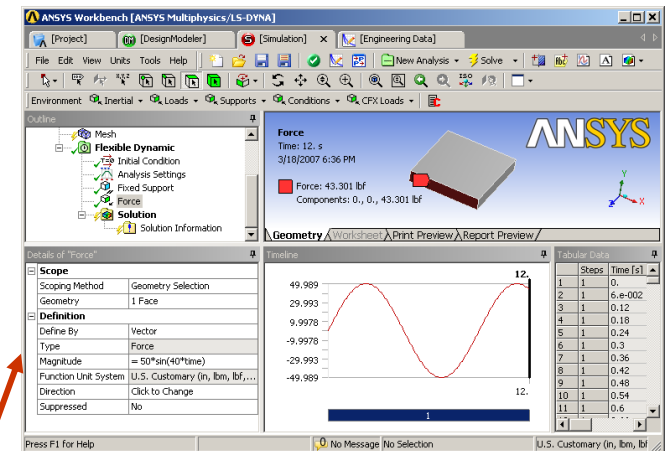
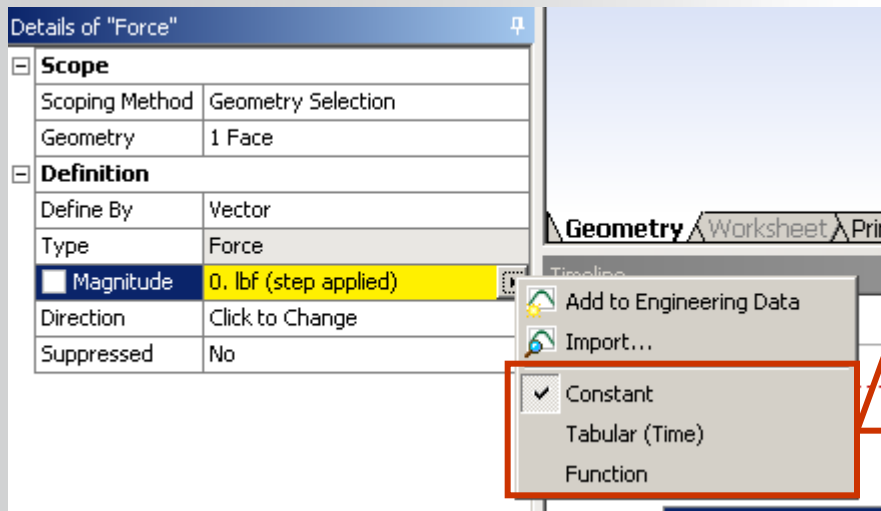
- Default condition is that all bodies are at rest
- If some bodies have zero initial displacement but non-zero constant initial velocity, this can be input
 - Only bodies can be specified
 - Enter constant initial velocity (Cannot specify more than one constant velocity value with this method)



Time-Varying Loads

Structural loads and joint conditions can be input as time-dependent *load histories*

- When adding a Load or Joint Condition, the magnitude can be defined as a *constant*, *tabular* value, or *function*.
- The values can be entered directly in the Workbench Mechanical GUI



Response Spectrum Analysis



Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

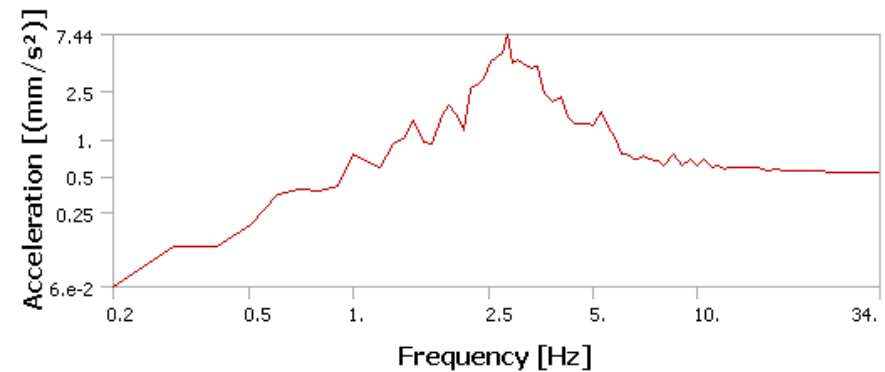
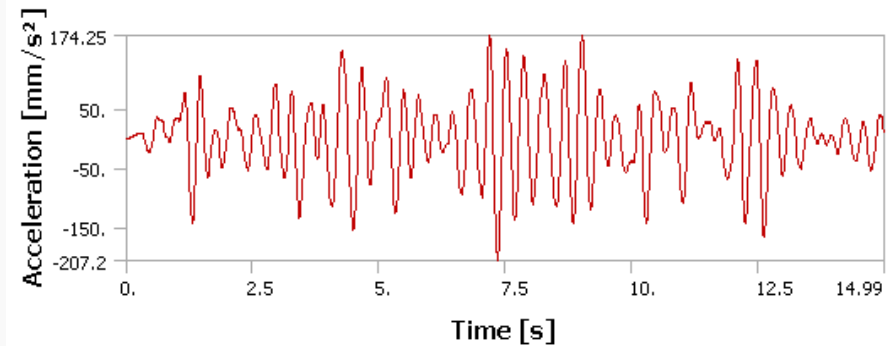
Description & Purpose

A response-spectrum analysis calculates the **maximum response** of a structure to a transient loading.

It is performed as a fast alternative of approximating a full transient solution.

The maximum response is computed as scale factor times the mode shape.

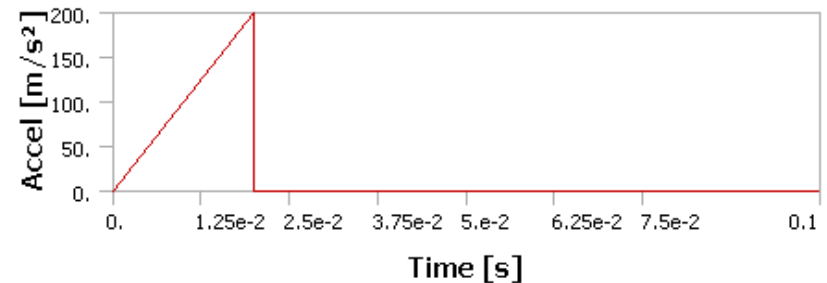
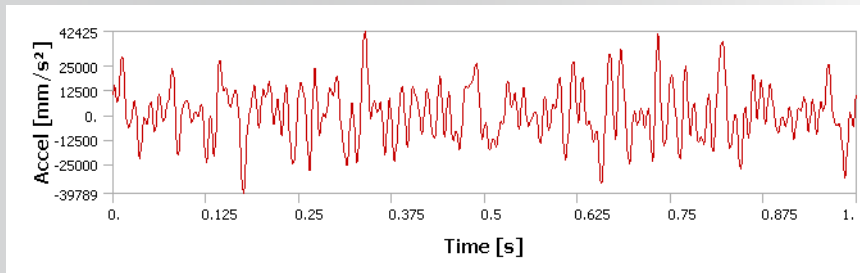
These maximum responses are then combined to give a total response of the structure.



Description & Purpose

It is common to have a large models excited by transient loading.

- e.g., building subjected to an earthquake
- e.g., electronic component subjected to shock loading



The most accurate solution is to run a long transient analysis.

- “Large” means many DOF. “Long” means many time points.
- In many cases, this would take too much time and compute resources.

Instead of solving the (1) large model and (2) long transient together, it can be desirable to approximate the maximum response quickly.

Common Uses

Commonly used in the analysis of:

- **Electronic equipment for shock loading**
- **Nuclear power plant buildings and components, for seismic loading**
- **Commercial buildings in earthquake zones**

Types of Response Spectrum analysis:

Single-point response spectrum

- **A single response spectrum excites all specified points in the model.**

Multi-point response spectrum

- **Different response spectra excite different points in the model.**



These two frequencies can often be identified on a response spectrum

- This divides the spectrum into three regions

1. Low frequency (below f_{SP})

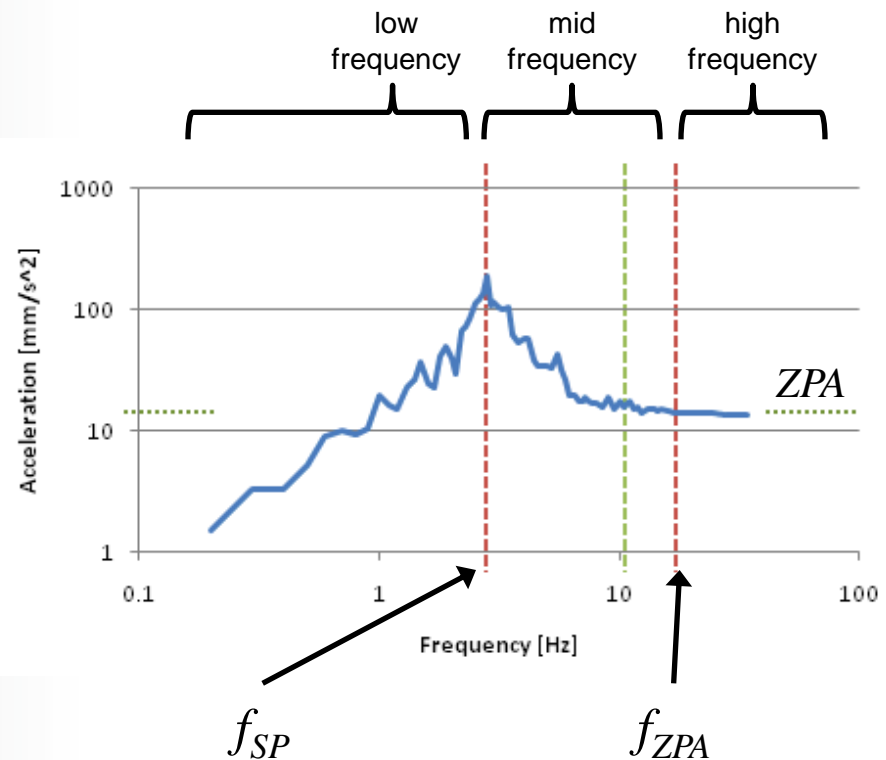
- periodic region
- modes are periodic & generally uncorrelated unless closely spaced

2. Mid frequency (between f_{SP} and f_{ZPA})

- transition from periodic to rigid
- modes have periodic component and rigid component

3. High Frequency (above f_{ZPA})

- rigid region
- modes correlated with input frequency and, therefore, also with themselves



frequency at peak response
(spectral peak)

frequency at rigid response
(zero period acceleration)

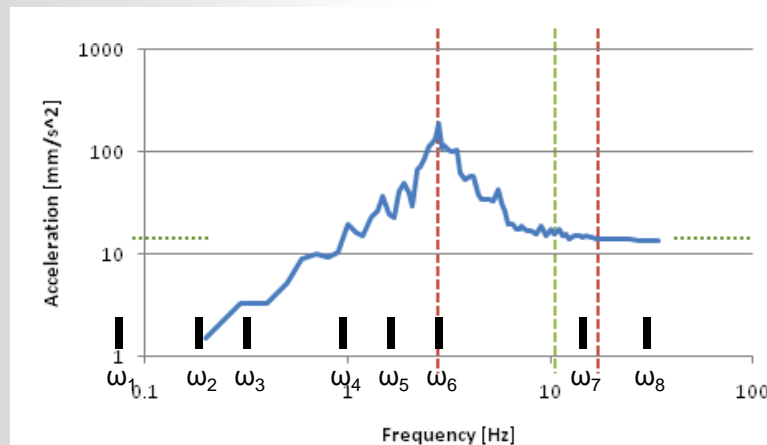
Recommended Solution Procedure

The recommended solution method is generally specified by your design code.

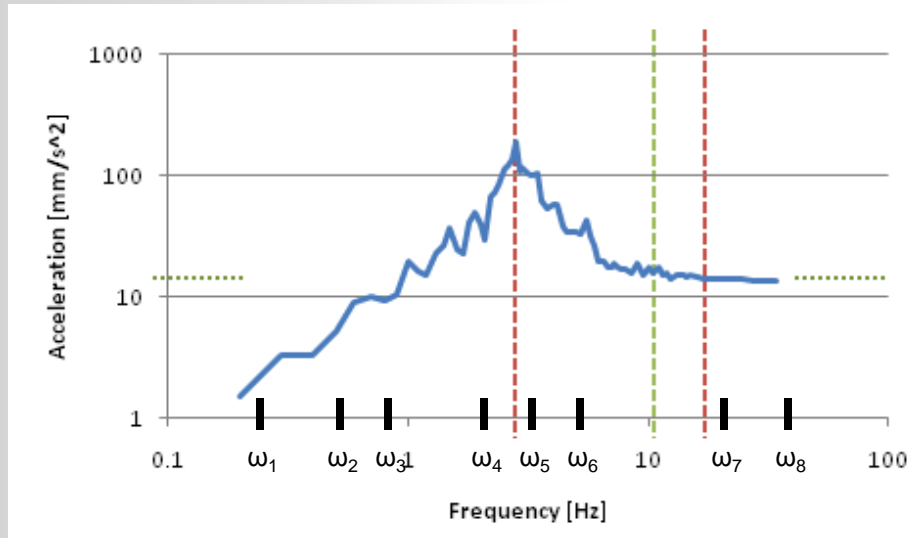
- combination method
- rigid response method
- missing mass effects

Alternatively, the best solution method can be determined by

- extracting the modes to be used for combination and
- comparing them to the response spectrum



Recommended Solution Procedure



Modes only in low-frequency region

- SRSS (or CQC/ROSE for closely spaced modes).
- No rigid response effects. No missing mass effects.

Modes only in mid- to high-frequency region

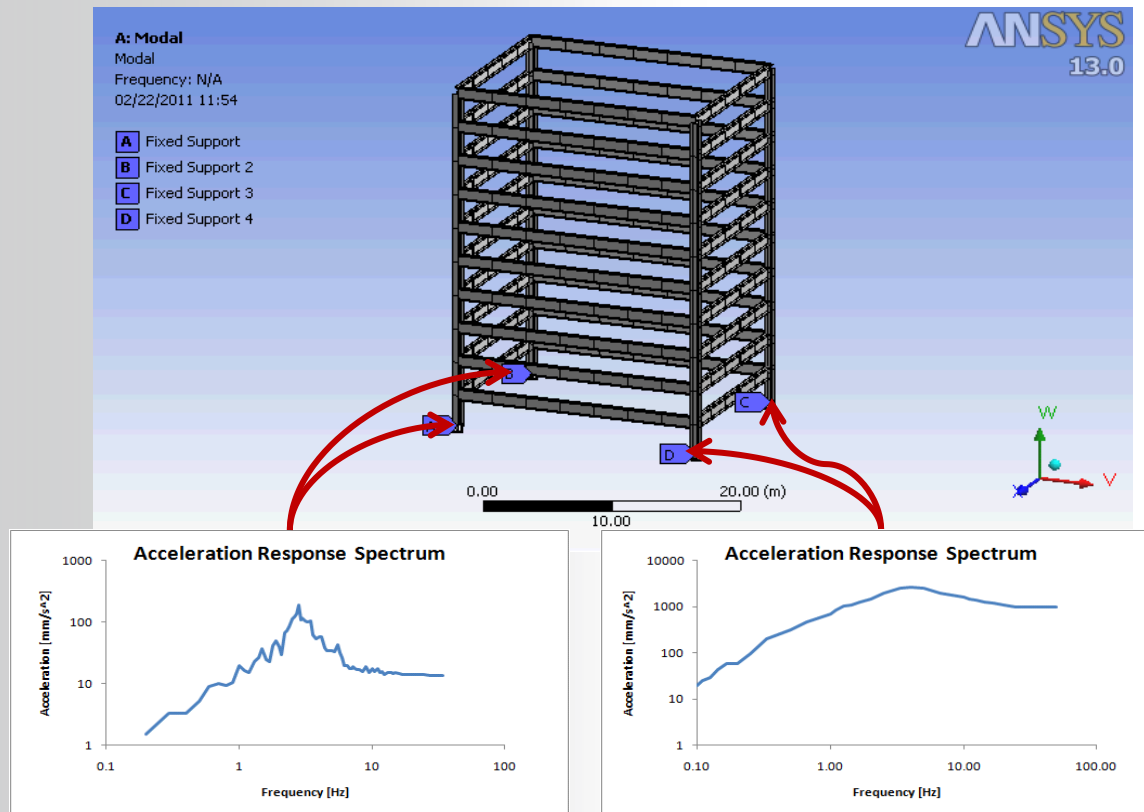
- SRSS (or CQC/ROSE for closely spaced modes).
- Rigid response by Lindley or Gupta method. Missing mass on.

Modes in all frequency regions

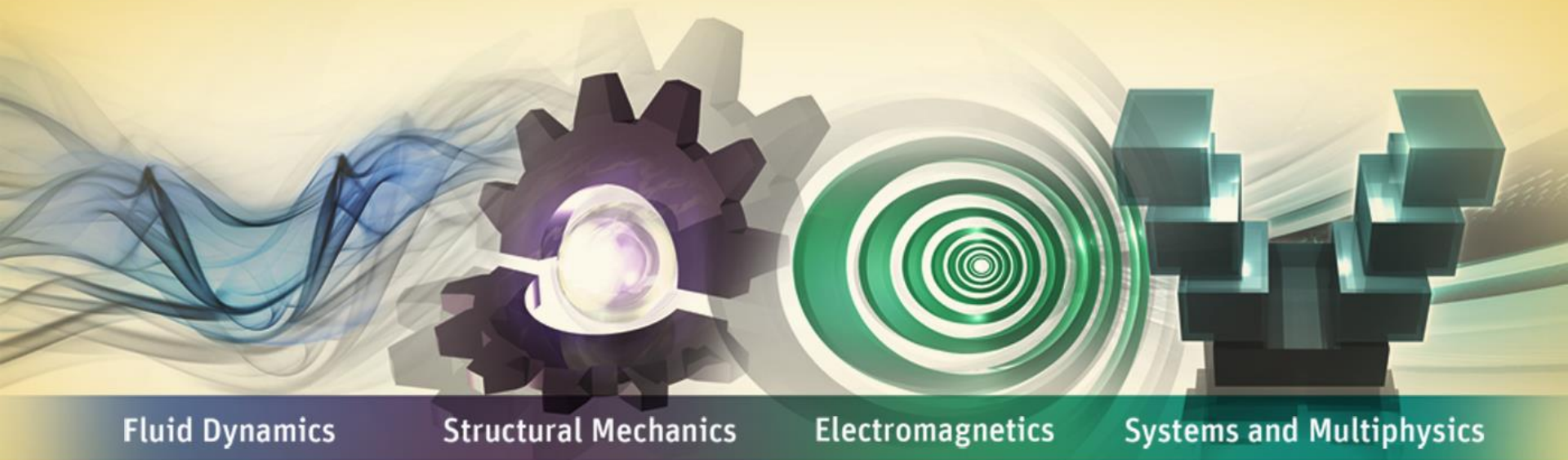
- SRSS (or CQC/ROSE for closely spaced modes).
- **Rigid response by Gupta method. Missing mass on.**

Multi-Point Response Spectrum

In multi-point response spectrum (MPRS), different constrained points can be subjected to different spectra (up to 100 different excitations).



Random Vibration



Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

Definition and Purpose

Many common processes result in random vibration

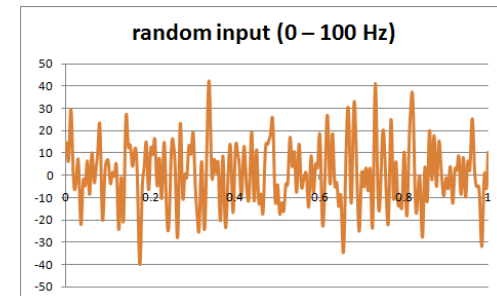
- Parts on a manufacturing line
- Vehicles travelling on a roadway
- Airplanes flying or taxiing
- Spacecraft during launch



Courtesy: NASA

The amplitudes at these frequencies vary randomly with time.

- We need some way of describing and quantifying this excitation.



Assumptions & Restrictions

The structure has

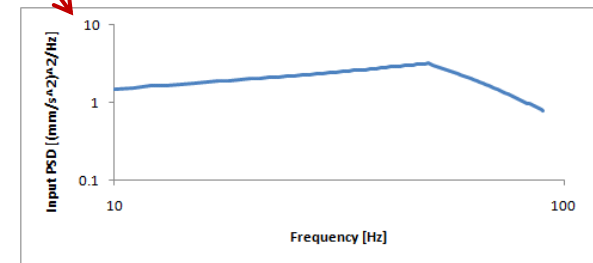
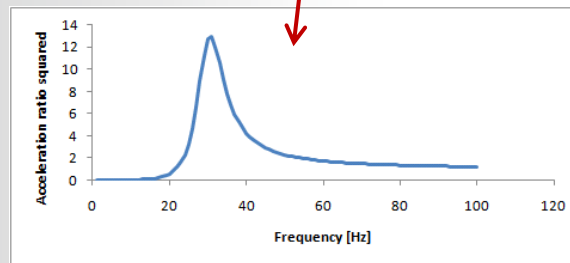
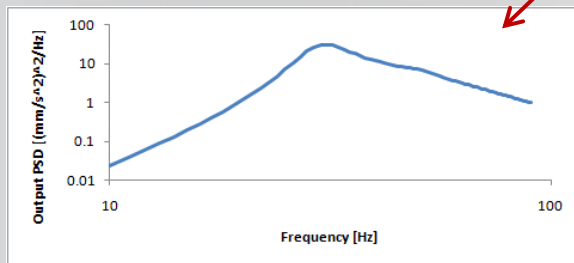
- no random properties
- no time varying stiffness, damping, or mass
- no time varying forces, displacement, pressures, temperatures, etc applied
- light damping
 - damping forces are much smaller than inertial and elastic forces

The random process is

- stationary (does not change with time)
 - the response will also be a stationary random process
- ergodic (one sample tells us everything about the random process)

To calculate the response PSD (RPSD), multiply the input PSD by the response function

$$S_{out}(\omega) = \left(\frac{a_{out}}{a_{in}} \right)^2 S_{in}(\omega)$$

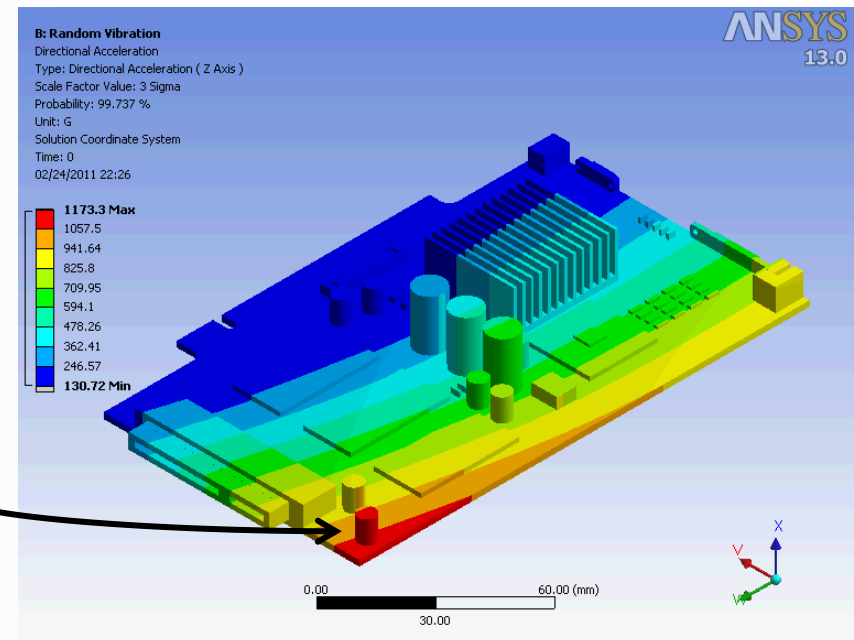
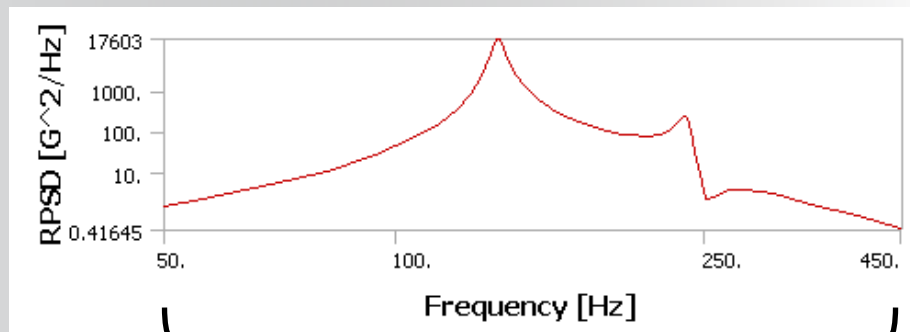


or

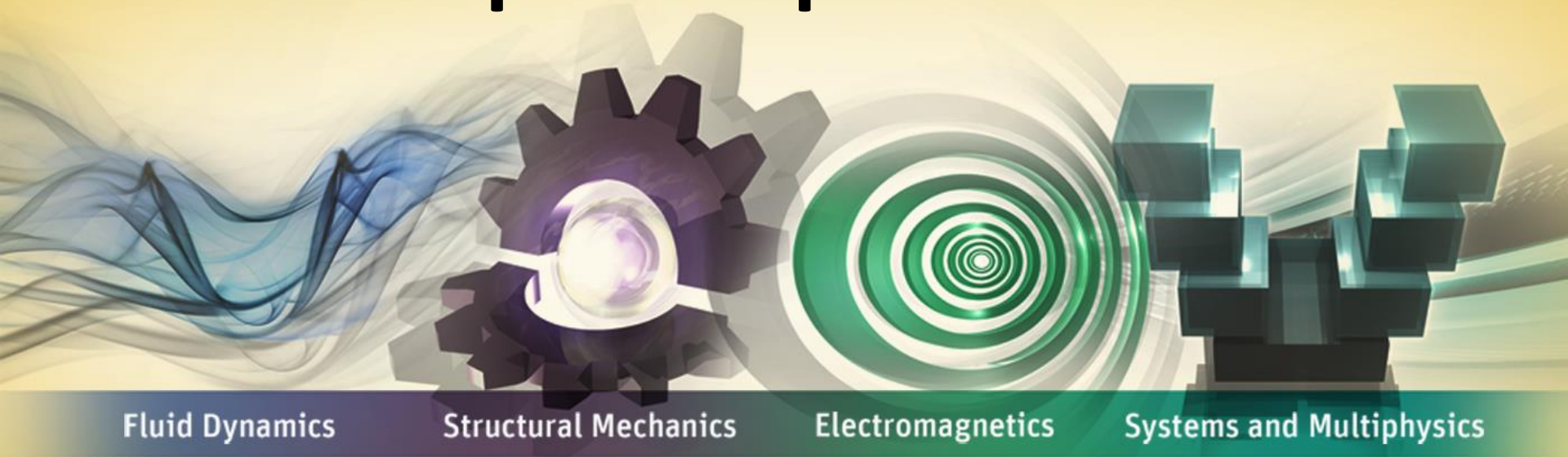
$$\text{RPSD} = \left(\frac{a_{out}}{a_{in}} \right)^2 (\text{input PSD})$$

For real models with multiple DOFs

- RPSDs are calculated for every node in every free direction at each frequency
 - RPSDs can be plotted for each node in a specific direction versus frequency
- a RMS value (sigma value) for the entire frequency range is calculated for every node in every free direction
 - sigma values can be plotted as a contour for the entire model for a specific direction.



Shock Analysis using Transient & Response Spectrum



Fluid Dynamics

Structural Mechanics

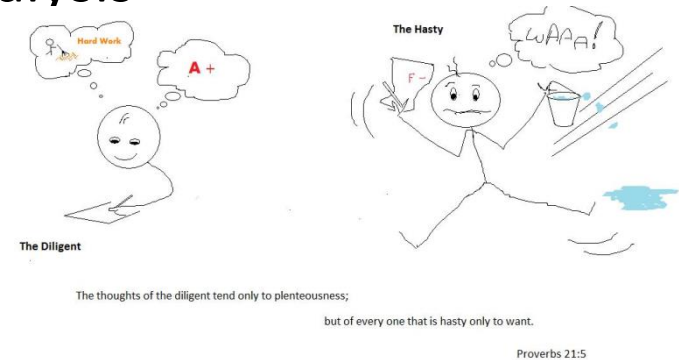
Electromagnetics

Systems and Multiphysics

Kelly Morgan
ANSYS Inc.

Response Spectrum Method

- Commonly used for large models
- Solve much faster than a full transient analysis
- Includes non-stationary excitations
- Linear analysis only



Transient (Time History Analysis)

- Include non-stationary and non-linear analysis
- Computationally quite expensive

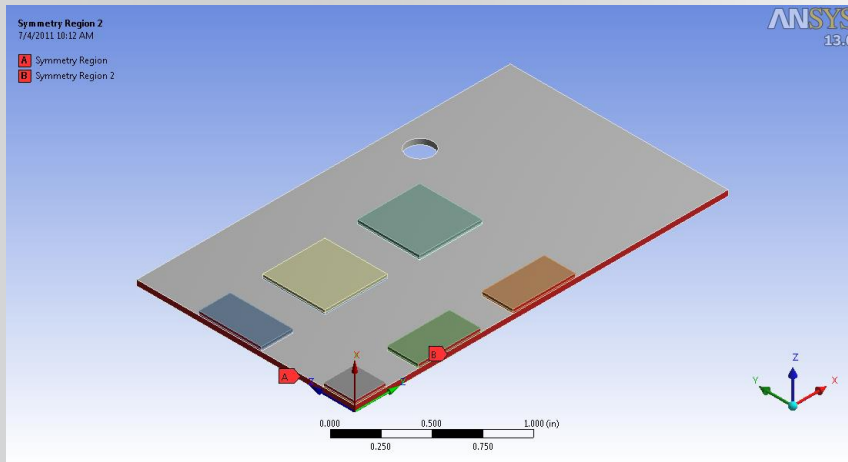
PCB model details

Basic guidelines for performing shock analysis based on Implicit Transient as well as Response spectrum in ANSYS WB is used

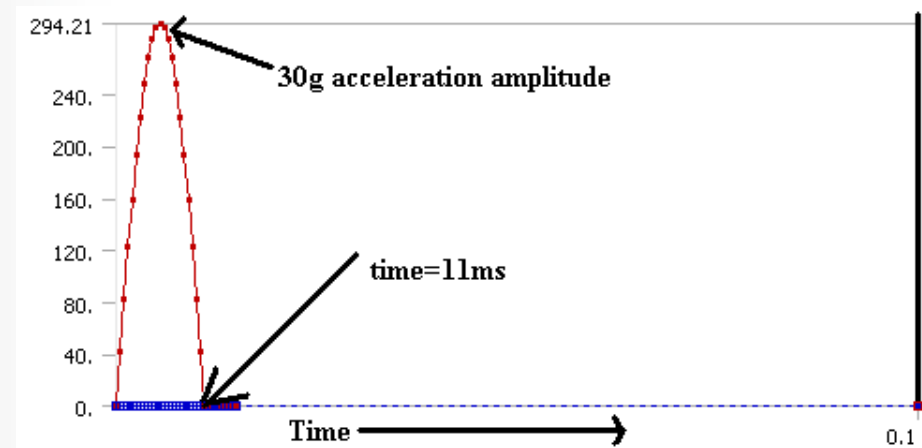
A real life example of a PCB subjected to standard **30g-11ms-half sine shock** is considered as the basis of the study.

Different parameters like results, memory requirements, solution time required for both the Transient & RS are compared.

Quarter symmetry model is taken



Quarter symmetric PCB



Shock load

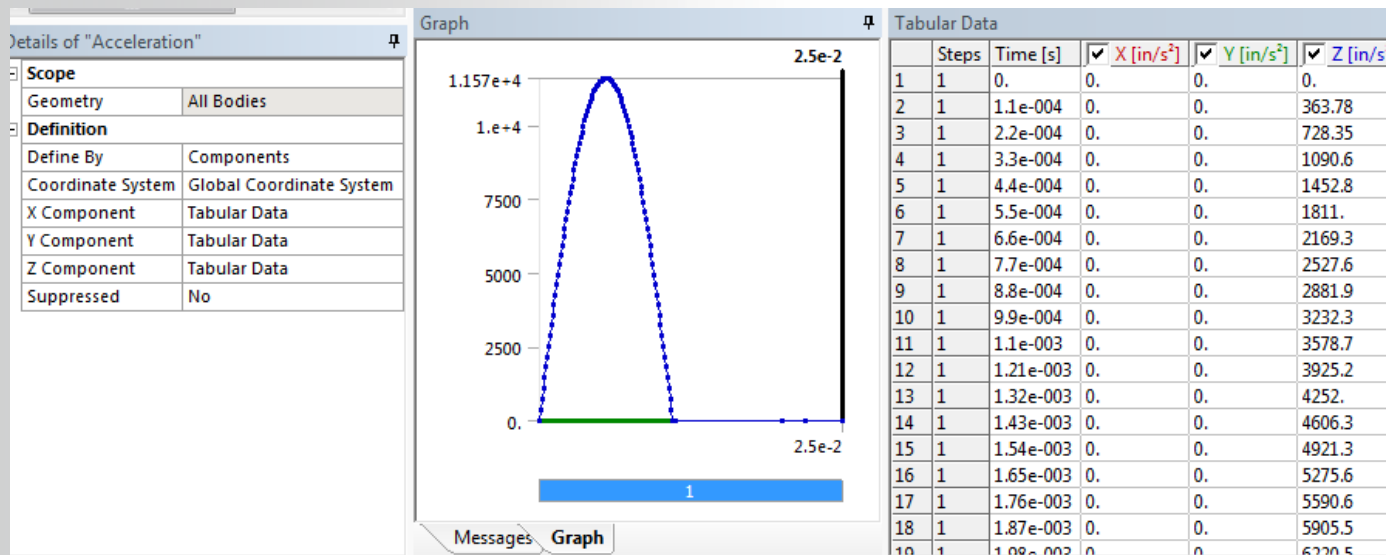
Define Materials, BCs, Loads:

Material properties of different components are defined in 'Engineering data'.

Transient analysis is done for a total time of 2.5×10^{-2} sec.

Initial conditions: Zero displacement and zero initial velocity.

- Fixed BC: The cantilever is fixed at one end.
- Acceleration Shock load: 30g-11ms, Half-sine shock pulse in transverse, z-dirn.



Transient shock-load input data

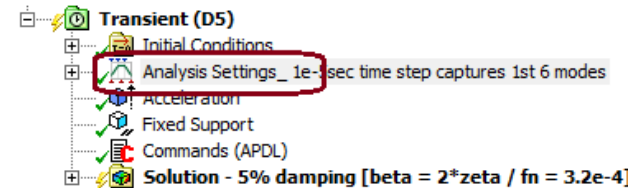
Analysis settings and time step definitions

Load steps, end time, time step size, damping, etc. are defined in the Analysis settings.

Including non-linear effects or not can also be defined.

Time step size is very **crucial** in Transient analysis as it determines the no. of dynamic modes one can capture.

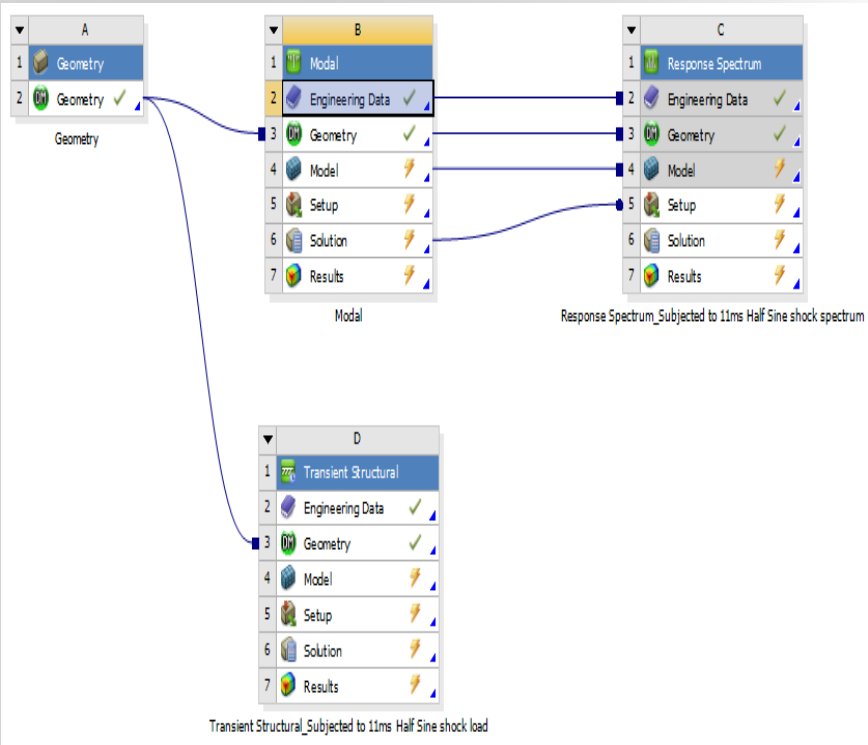
- 1st, 4th, 5th mode freqs.=311, 3161, 4371 Hz, respectively.
- To capture 4th mode time step should be around $(1/20 \times 3161) = 1.58 \times 10^{-5}$ s. 1×10^{-4} is taken as the time step for solution.



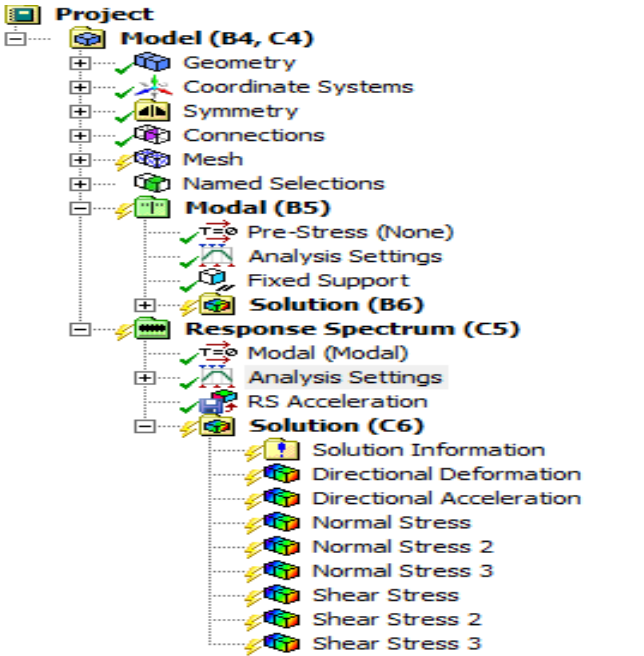
Details of "Analysis Settings_1e-5sec time step captures 1st 6 modes"	
Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	2.5e-002 s
Auto Time Stepping	On
Define By	Time
Initial Time Step	1.e-004 s
Minimum Time Step	1.e-004 s
Maximum Time Step	1.e-004 s
Time Integration	On
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Program Controlled
Large Deflection	On
Restart Controls	
Nonlinear Controls	
Output Controls	
Damping Controls	
Beta Damping Defin...	Direct Input
Beta Damping Value	3.2e-004
Numerical Damping	0
Analysis Data Management	
Visibility	



Spectrum analysis for Half-sine shock



Project Schematic



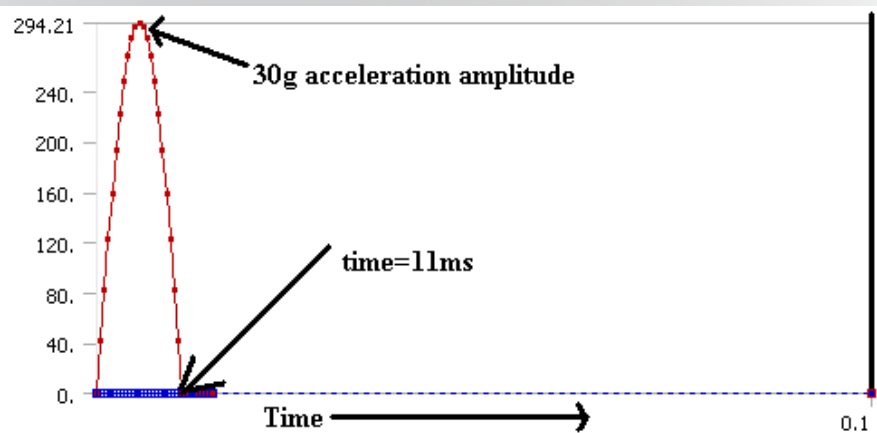
Details of "Analysis Settings"	
Options	
Number Of Modes To Use	6.
Spectrum Type	Single Point
Modes Combination Type	CQC
Output Controls	
Damping Controls	
Constant Damping Ratio	0.
Beta Damping Define By	Direct Input
Beta Damping Value	3.2e-004
Analysis Data Management	

Mechanical outline of Spectrum analysis



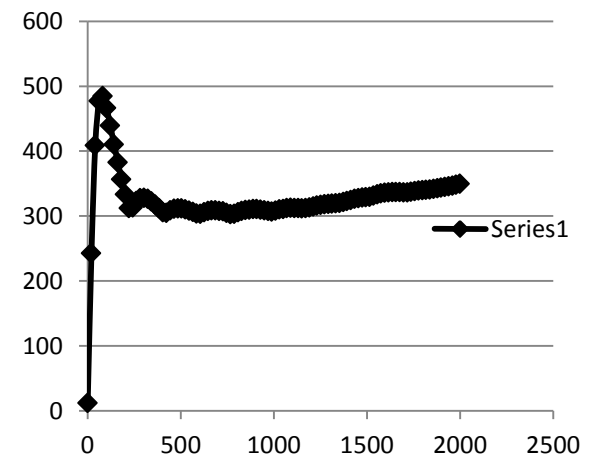
Converting Time domain load into Spectrum input

Time domain data can be converted into Spectrum data (Freq domain) through RESP command in ANSYS:



*Time domain data
(Transient analysis)*

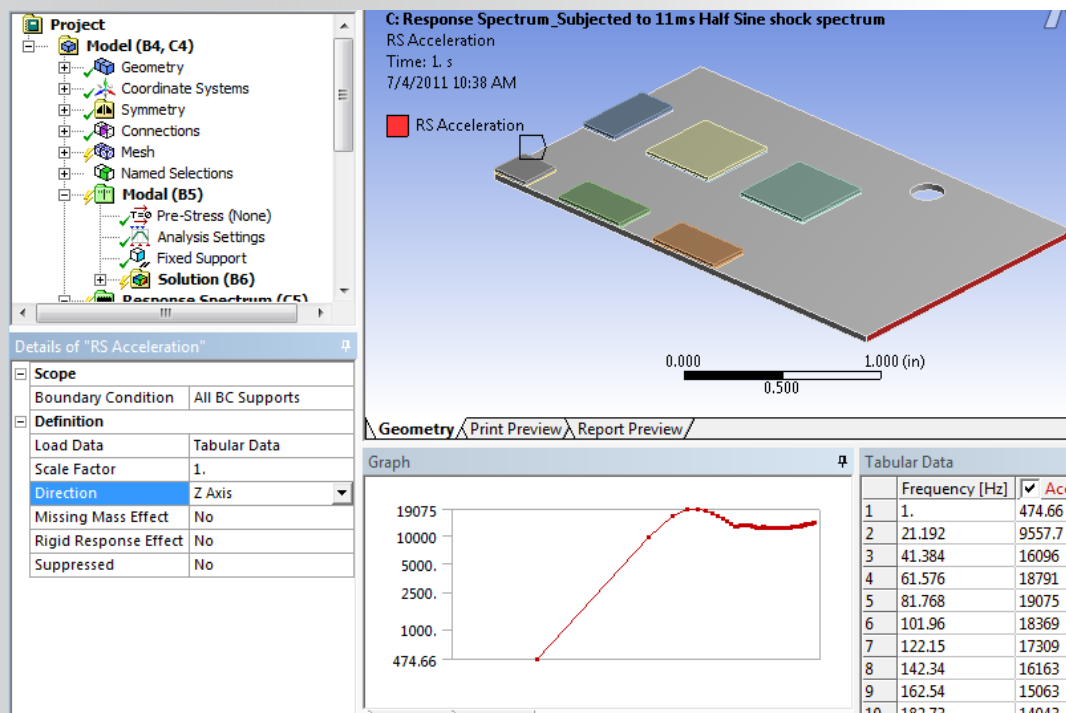
RESP
command →



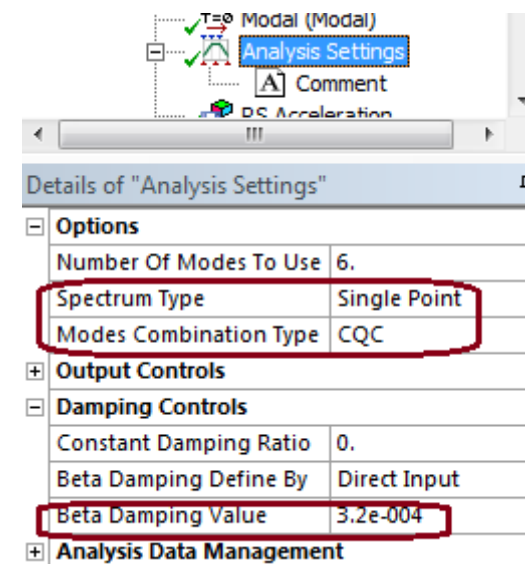
*Frequency domain data
(Response Spectrum)*



Applying acceleration Spectrum input in Spectrum analysis



RS acceleration input data



Settings of RS analysis

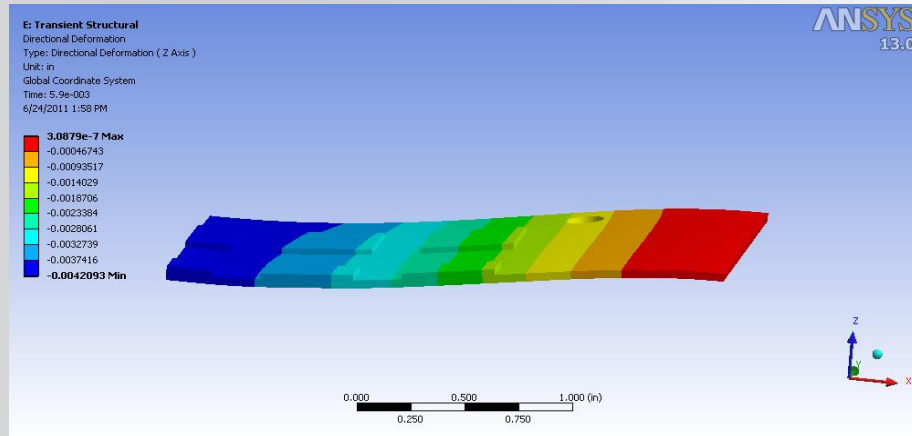
Transient versus RS

Maximum deformation, Normal & Shear stress are compared from the Transient & the RS analysis.

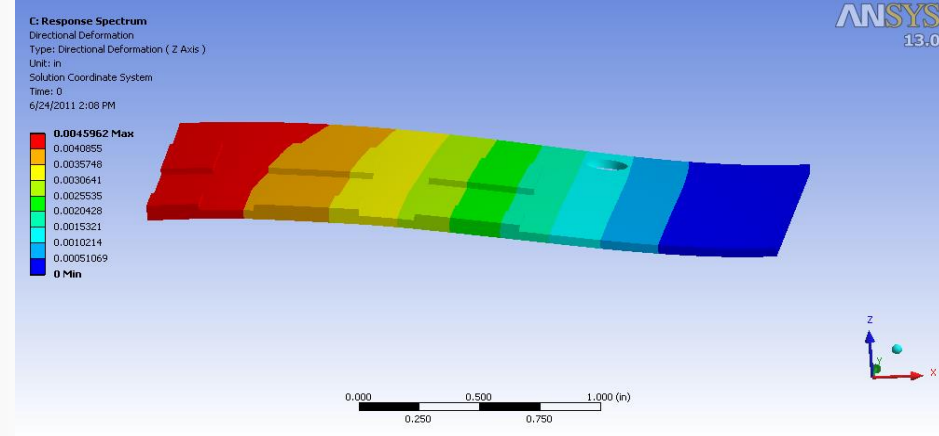
Transient results are assumed to be accurate and benchmark for accessing RS results.

Time taken for both the analysis are compared

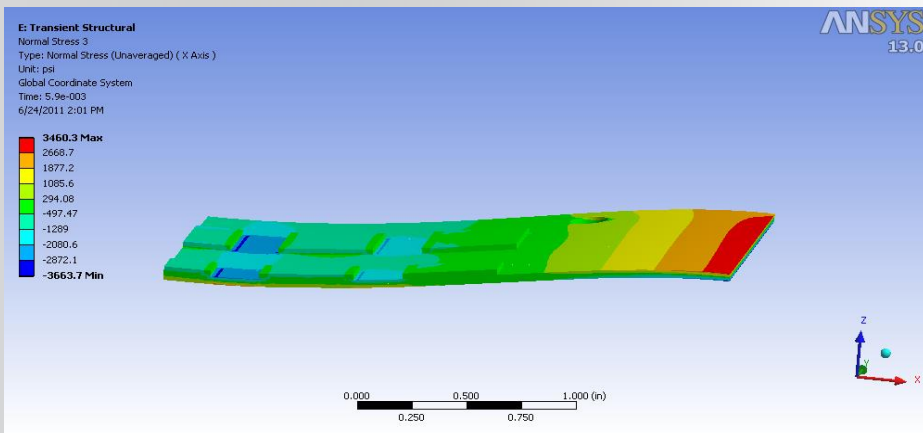
Results comparison- Maximum



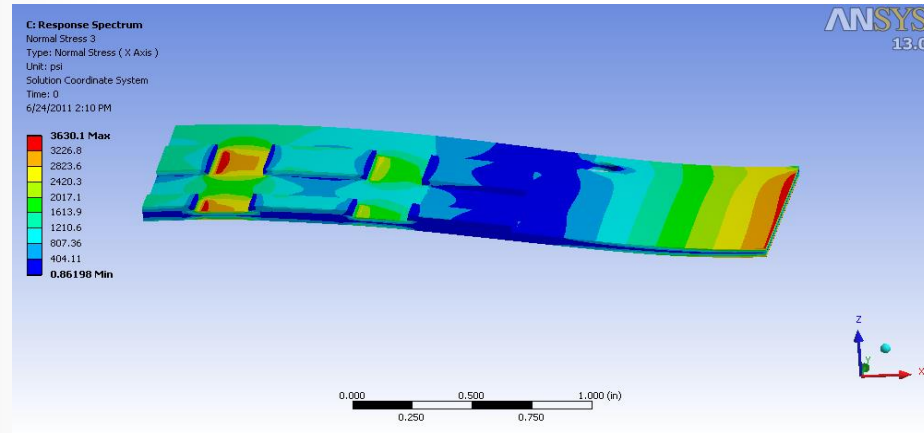
Transient directional deformation in z-dirn (Max= -4.2093 in)



RS directional deformation in z-dirn (Max= 4.5962e-3 in)

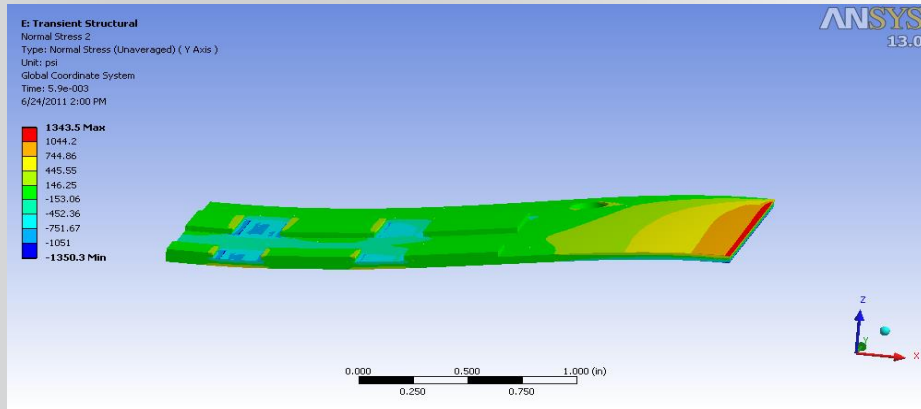


Transient Normal stress_x-dirn (Max=-3663.7 psi)

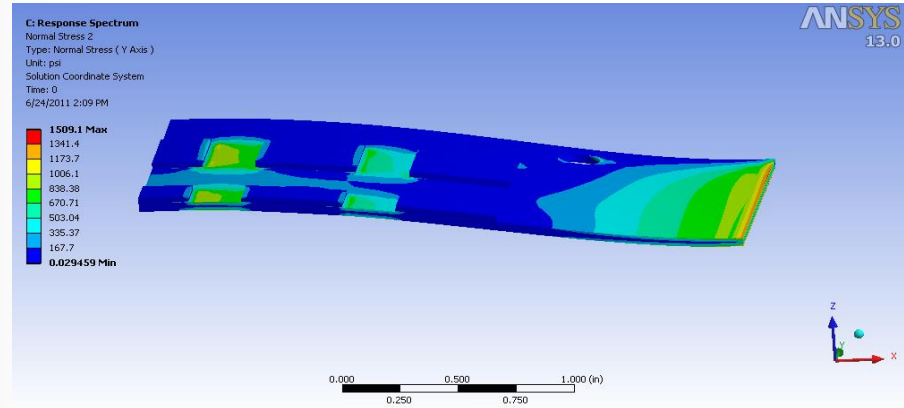


RS Normal stress_x-dirn (Max= 3630.1 psi)

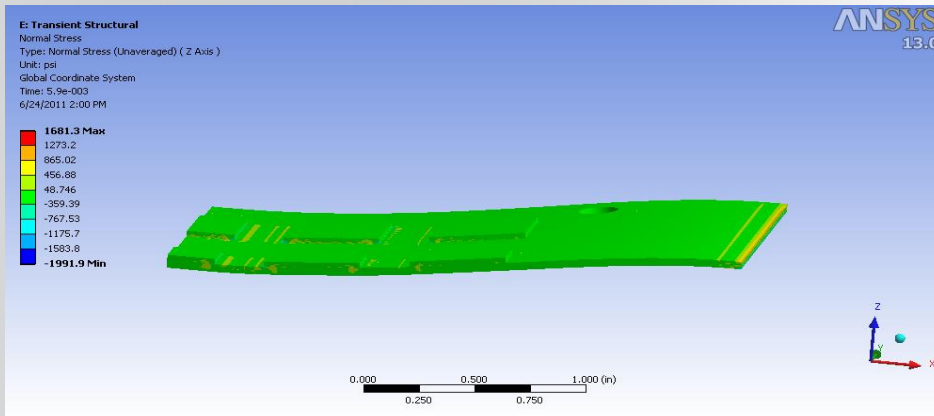
Results comparison- Maximum Normal Stress



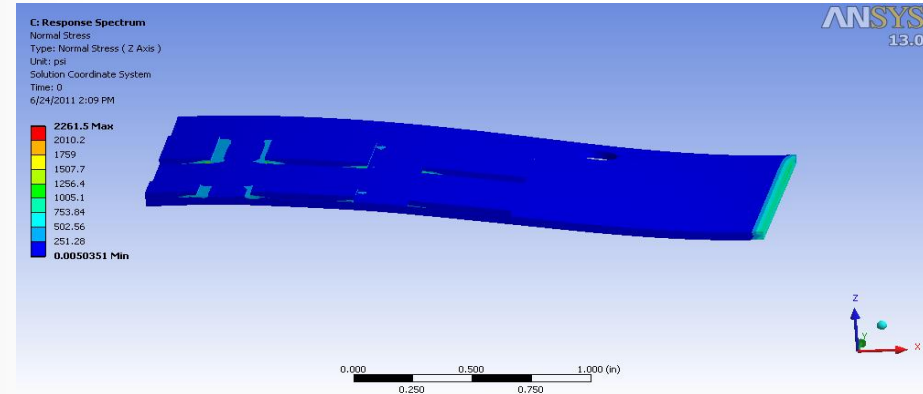
Transient Normal stress_y-dirn (Max= -1350.3 psi)



RS Normal stress_y-dirn (Max=1509.1 psi)

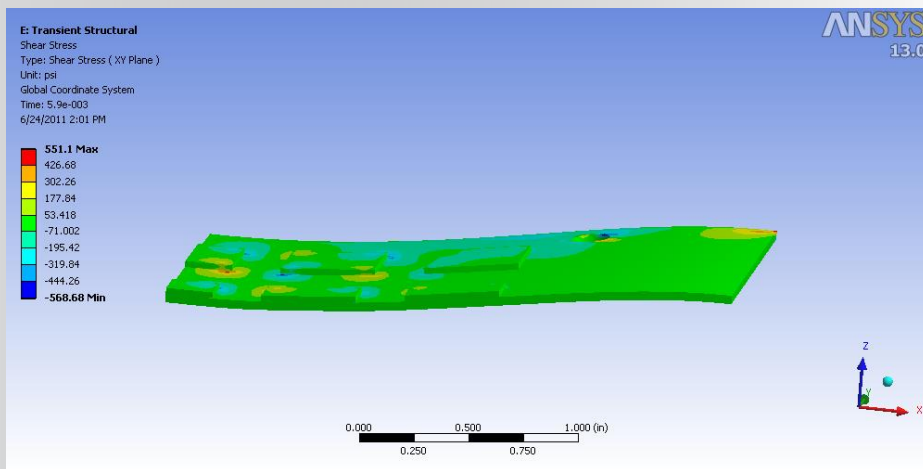


Transient Normal stress_z-dirn (Max= -1991.9 psi)

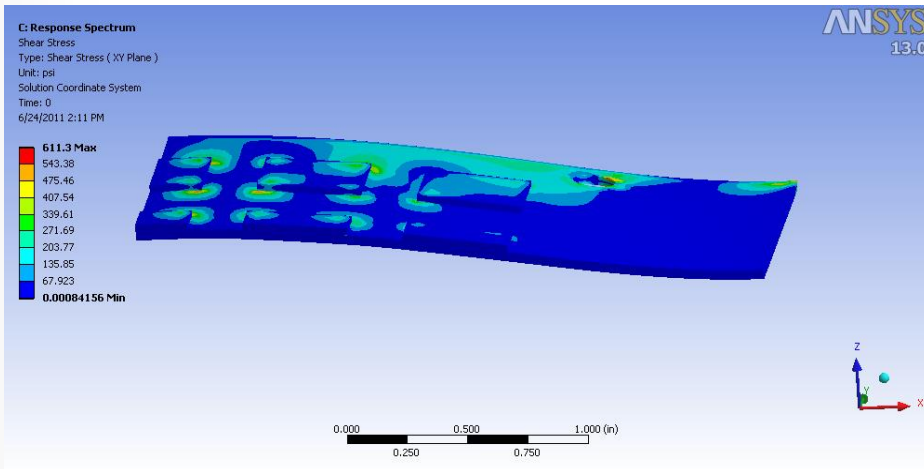


RS Normal stress_z-dirn (Max=2261.5 psi)

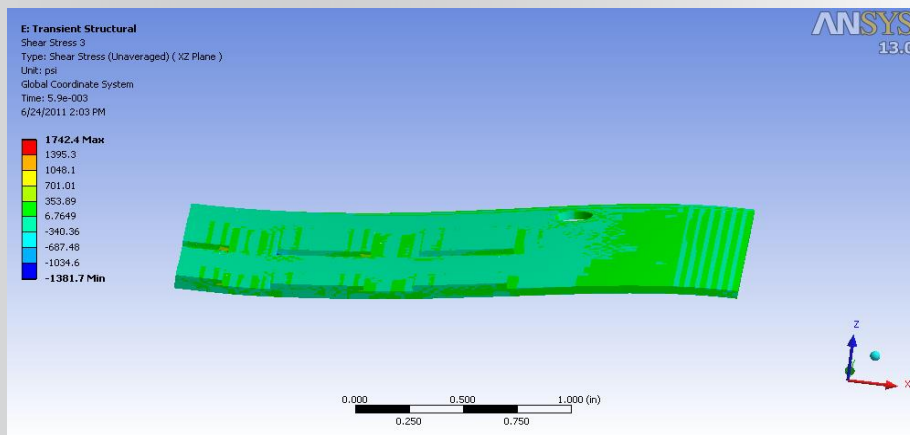
Results comparison- Shear stresses



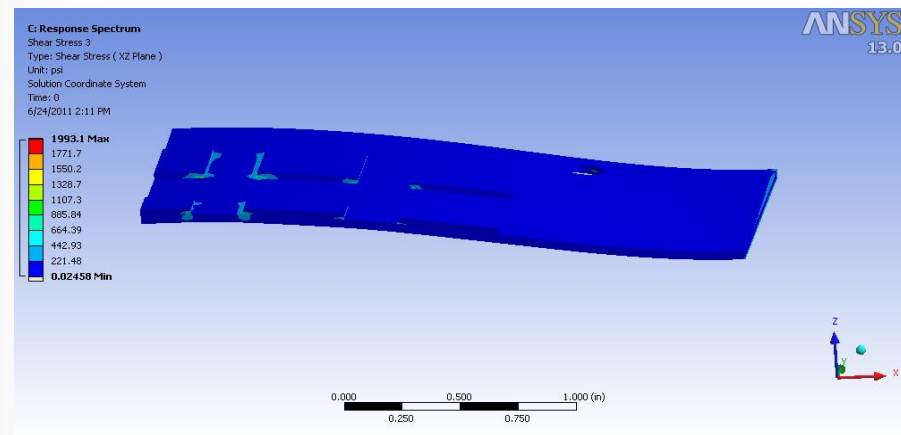
Transient Shear stress_{xy-dirn} (Max= -568.68 psi)



RS Shear stress_{xy-dirn} (Max= 611.3 psi)

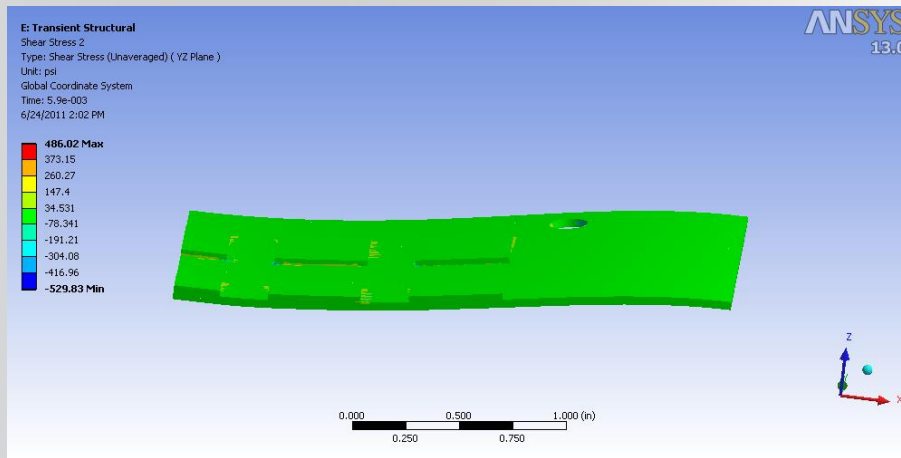


Transient Shear stress_{xz-dirn} (Max= 1742.4 psi)

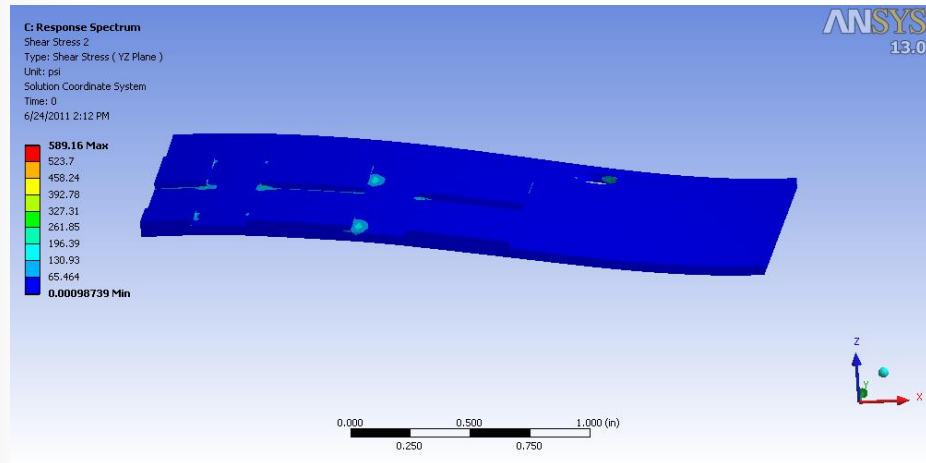


RS shear stress_{xz-dirn} (Max= 1993.1 psi)

Results comparison- Shear stresses



Transient Shear stress_yz-dirn (Max= -529.83 psi)



RS Shear stress_yz-dirn (Max= 589.16 psi)

Results summary:

- Results from RS are within 14% of the Transient results.
- As in RS, only absolute values of result quantities are taken, the plot colors look different. But considering only absolute values in Transient, the plots in RS & transient are comparable.
- Critical locations (high stress regions) are correctly predicted by RS.

Final observations & comments

The RS results are in reasonable limits of the Transient results.

The elapsed time in RS is quite less (125 times less in PCB model) than in the Transient run.

The time advantage is even more for big models, with contacts & complicated load histories typical in earthquakes, which justifies RS use in Shock and Earthquake analysis even at cost of some accuracy.

Result (Max)	Z-dirn deformation-(inch)	X-Normal Stress – (psi)	Y-Normal Stress – (psi)	Z-Normal Stress – (psi)	XY-Shear Stress – (psi)	YZ-Shear Stress – (psi)	ZX-Shear Stress – (psi)	CP Time – (sec)
<i>Full Transient</i>	-4.2093 e-3	- 3663.7	-1350.3	1991.9	- 568.68	1742.4	- 529.83	7192
<i>MSUP Transient</i>	-4.1766e-003	-3485.3 psi						49+136=185
<i>Spectrum (CQC)</i>	4.5962e-3	3630.1	1509.1	2261.5	611.3	1993.1	589.16	49+8.375=57.375
<i>Percentage difference</i>	9	0.09	11.3	13.5	7.4	14	11.1	7192/57.375= 125 times

Comments for usage in industry

When to go for Shock analysis using Response Spectrum?

- In industry, detailed, expensive analysis like Transient are not always necessary.
- For example, many times quick analysis of the systems is more important, even though if it comes with some compromise on accuracy.
- Also such quick analysis can be **a basis for selection** from a number of designs, and then carrying out detailed analysis of selected few.
- Detailed Transient analysis is nearly impossible, For example Seismic analysis of a complex component.

When to go for Shock analysis using implicit Transient ?

- When non-linearities play such an important role that they cannot be ignored for correct behavior of the model.
- Carrying out detailed analysis on selected specimens (based on prior analysis like RS) to finalize the design.
- When reliability of a particular design is very critical and accuracy is indispensable.

Thank You

Q/A