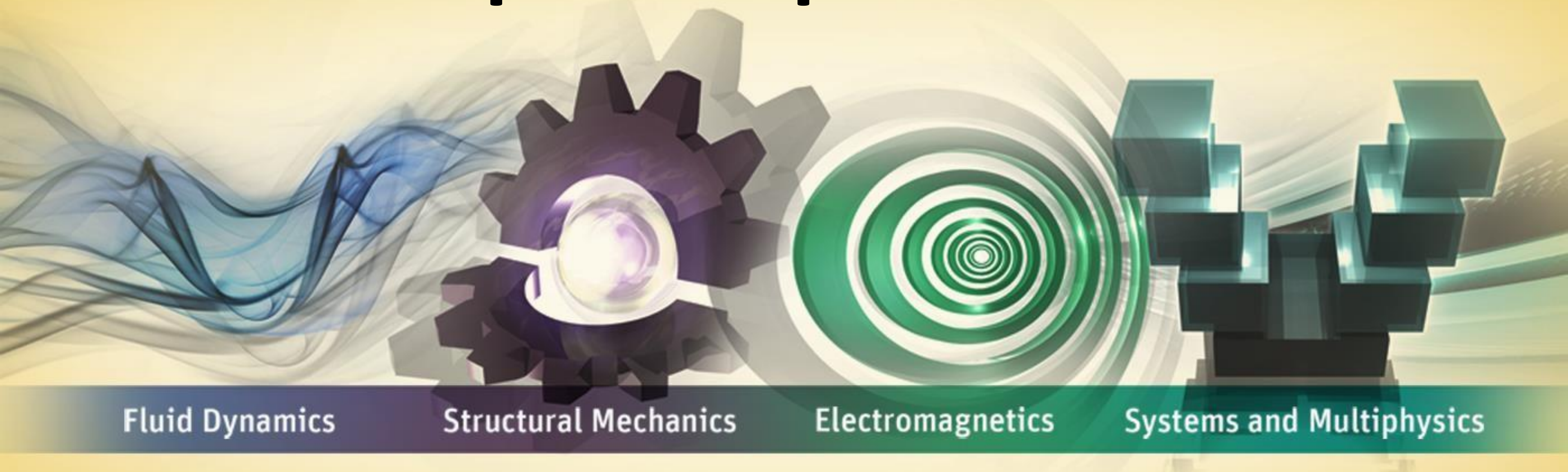
Realize Your Product Promise™

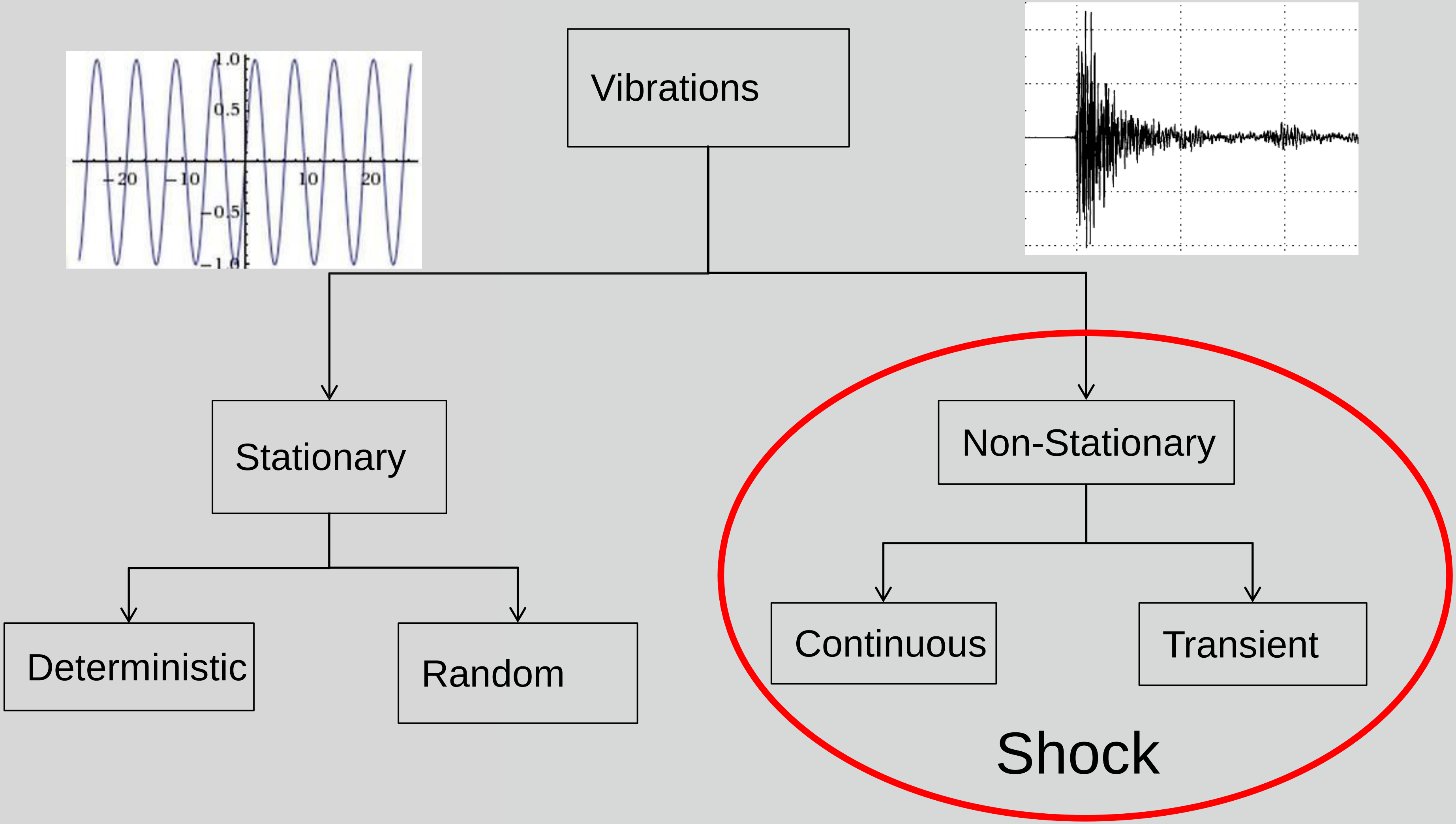
Shock Analysis using Transient & Response Spectrum



Anil Kumar

Contents

Types of Vibrations



Methods for Shock Analysis

Response Spectrum Method

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

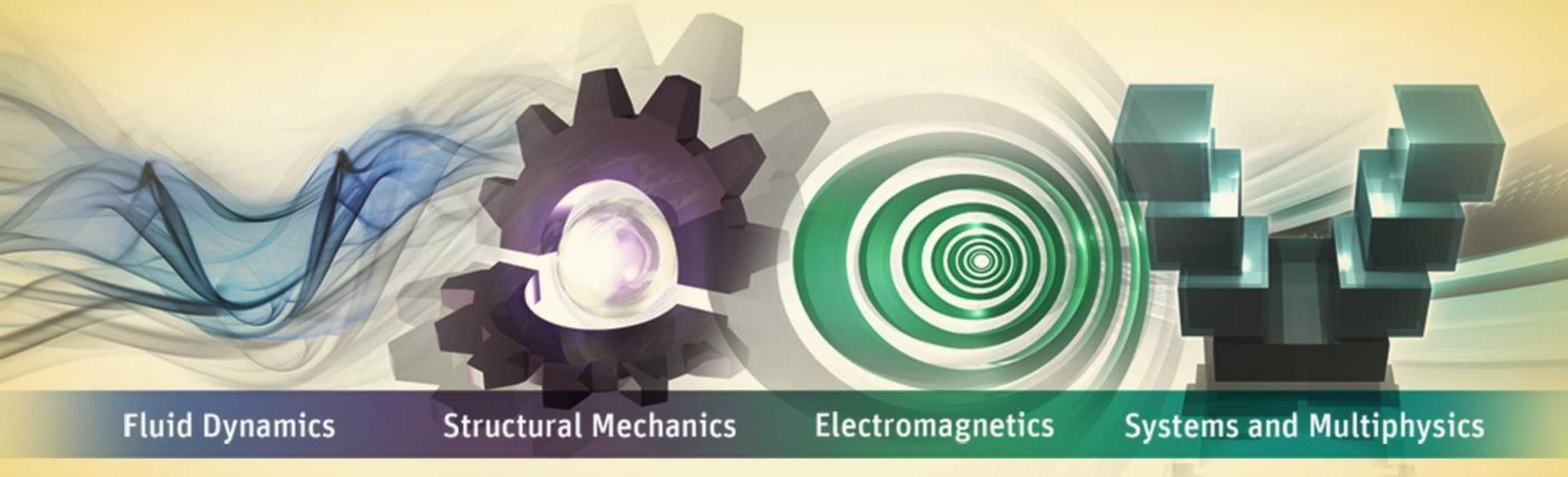
Transient (Time History Analysis)

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

ANSYS

Realize Your Product Promise™

Response Spectrum Analysis



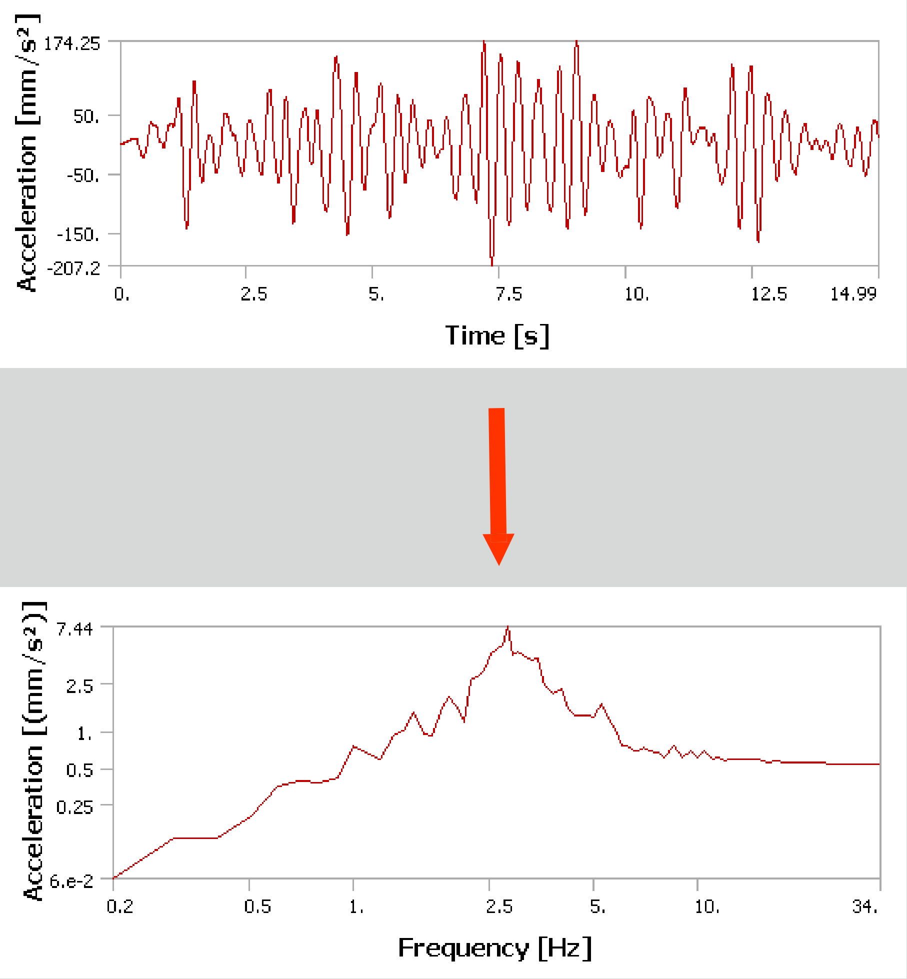
Description & Purpose

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.



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:

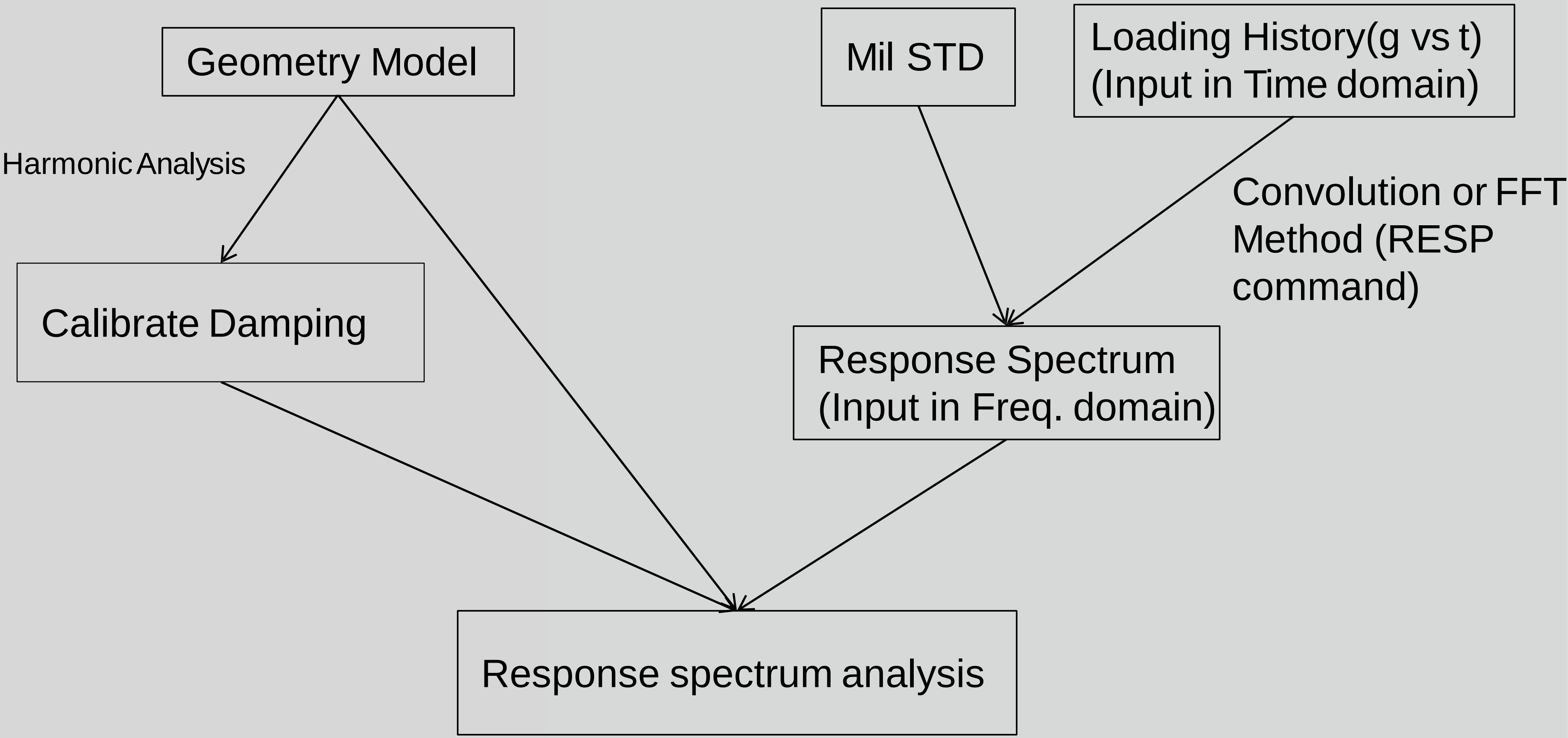
Single-point response spectrum

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

Multi-point response spectrum

- Different response spectra excite different points in the model.

Procedure


Geometry Model
Mil STD
Loading History(g vs t)
(Input in Time domain)
Harmonic Analysis
Convolution or FFT
Method (RESP
command)
Calibrate Damping
Response Spectrum
(Input in Freq. domain)
Response spectrum analysis

Calibrate Damping using Harmonic Analysis

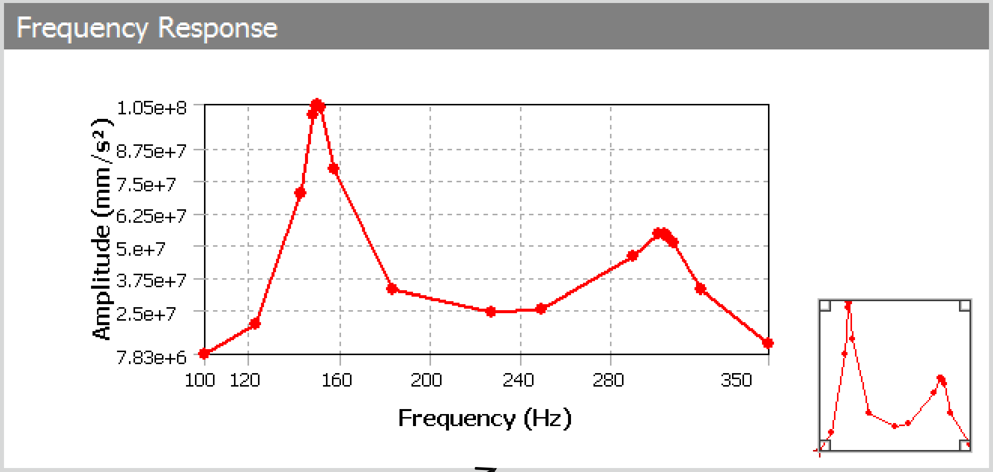
What is harmonic analysis?

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

Similar to a shaker table experiment.

Details of "Analysis Settings"

Options	
Range Minimum	100. Hz
Range Maximum	350. Hz
Cluster Number	5
Solution Method	Mode Superposition
Cluster Results	Yes
Modal Frequency Range	Program Controlled
Store Results At All Frequencies	Yes
Output Controls	
Damping Controls	
Constant Damping Ratio	5.e-002
Beta Damping Define By	Direct Input
Beta Damping Value	0.
Analysis Data Management	

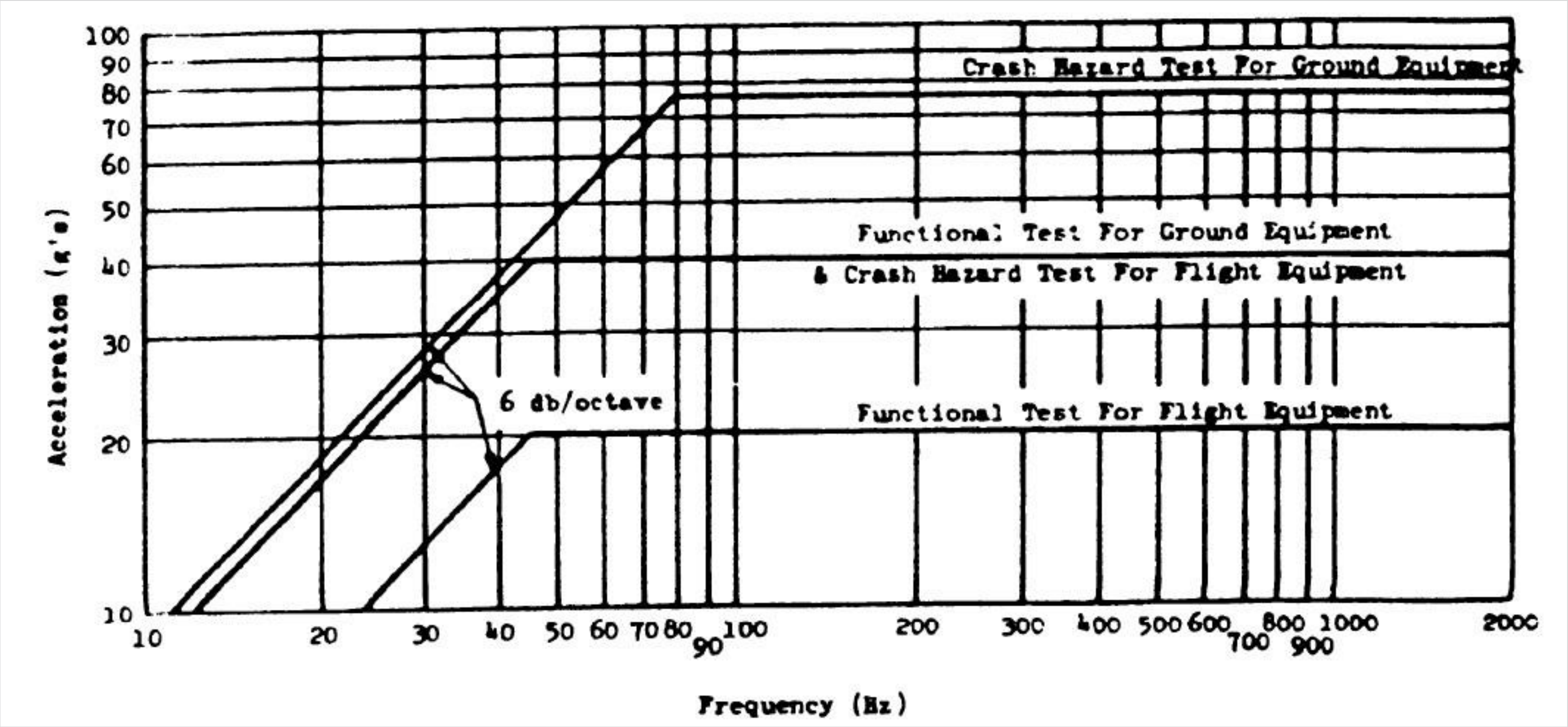


Adjust damping values until frequency sweep is similar to experiment

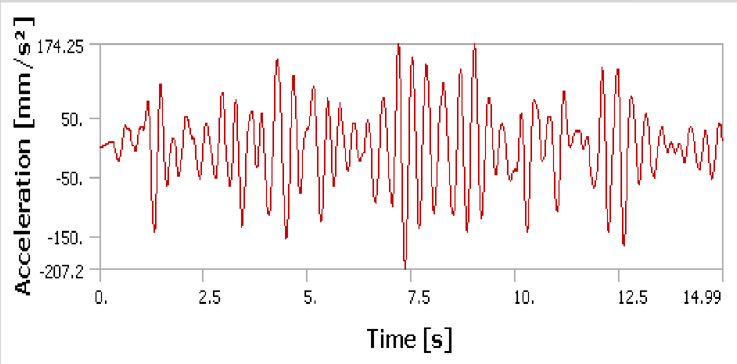
Response Spectrum Data

Follow industry standards

- MIL-STD-810E 516.4



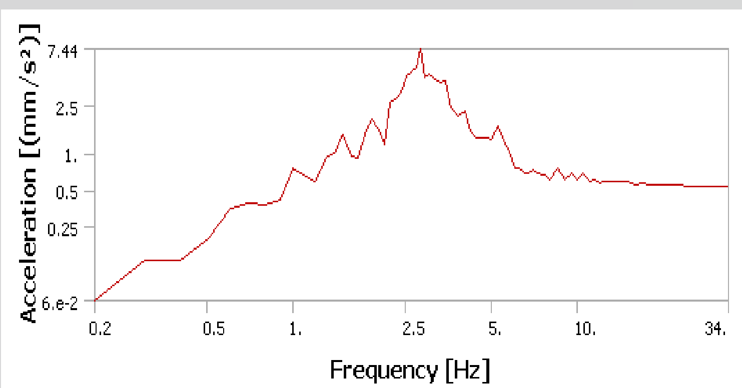
Converting time history to Response Spectrum Input data

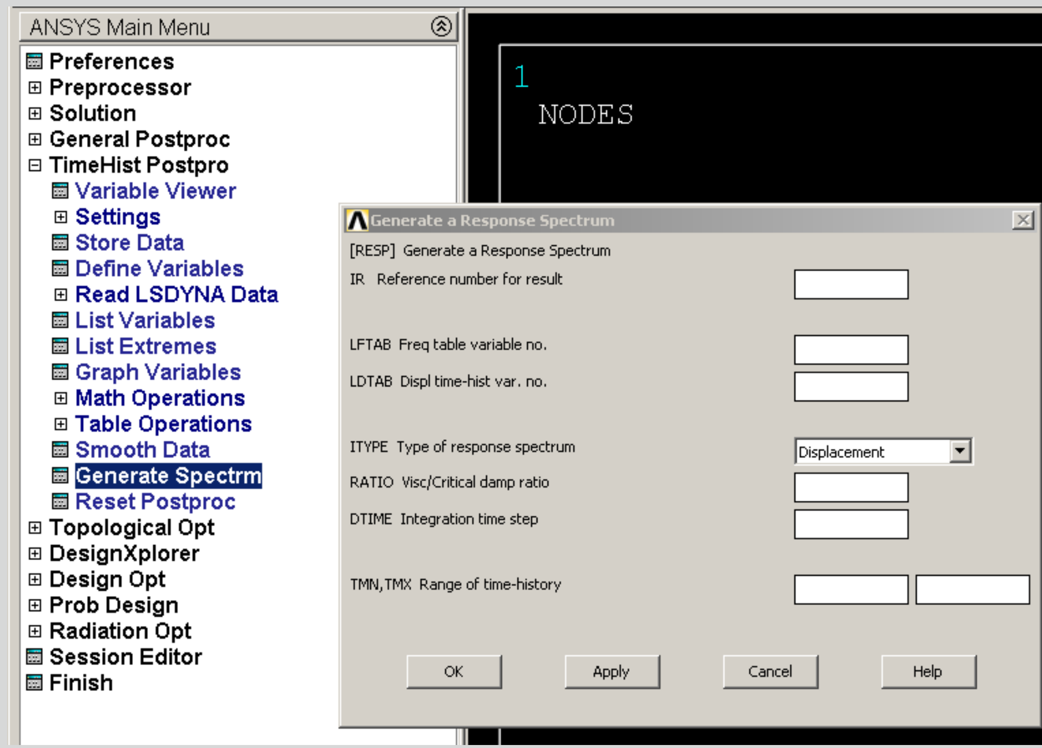


ANSYS MAPDL can be used to generate the response spectrum.
As of V13 this capability is not yet available in Workbench Mechanical

Convolution Integral or FFT

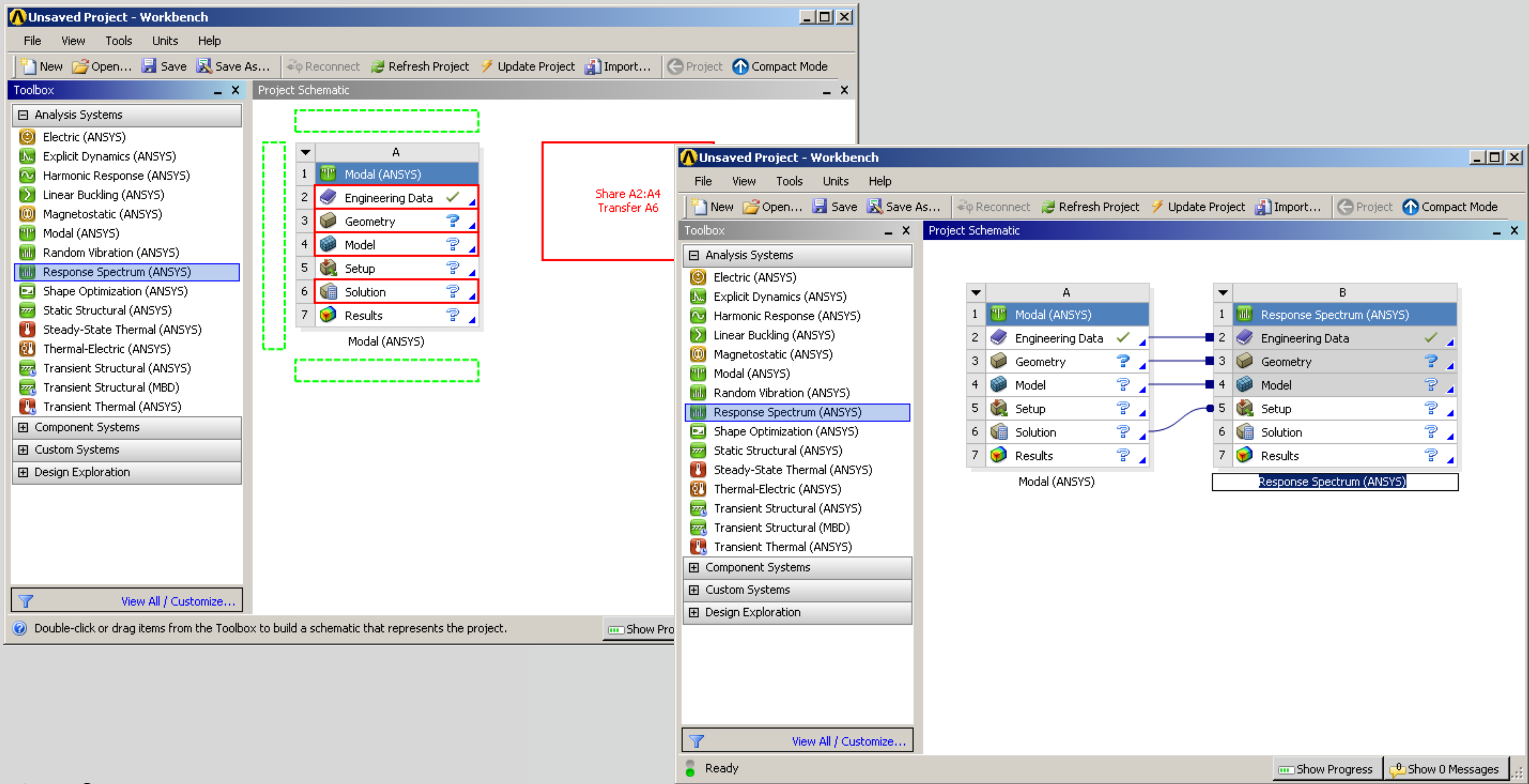
RESP command





Project Schematic

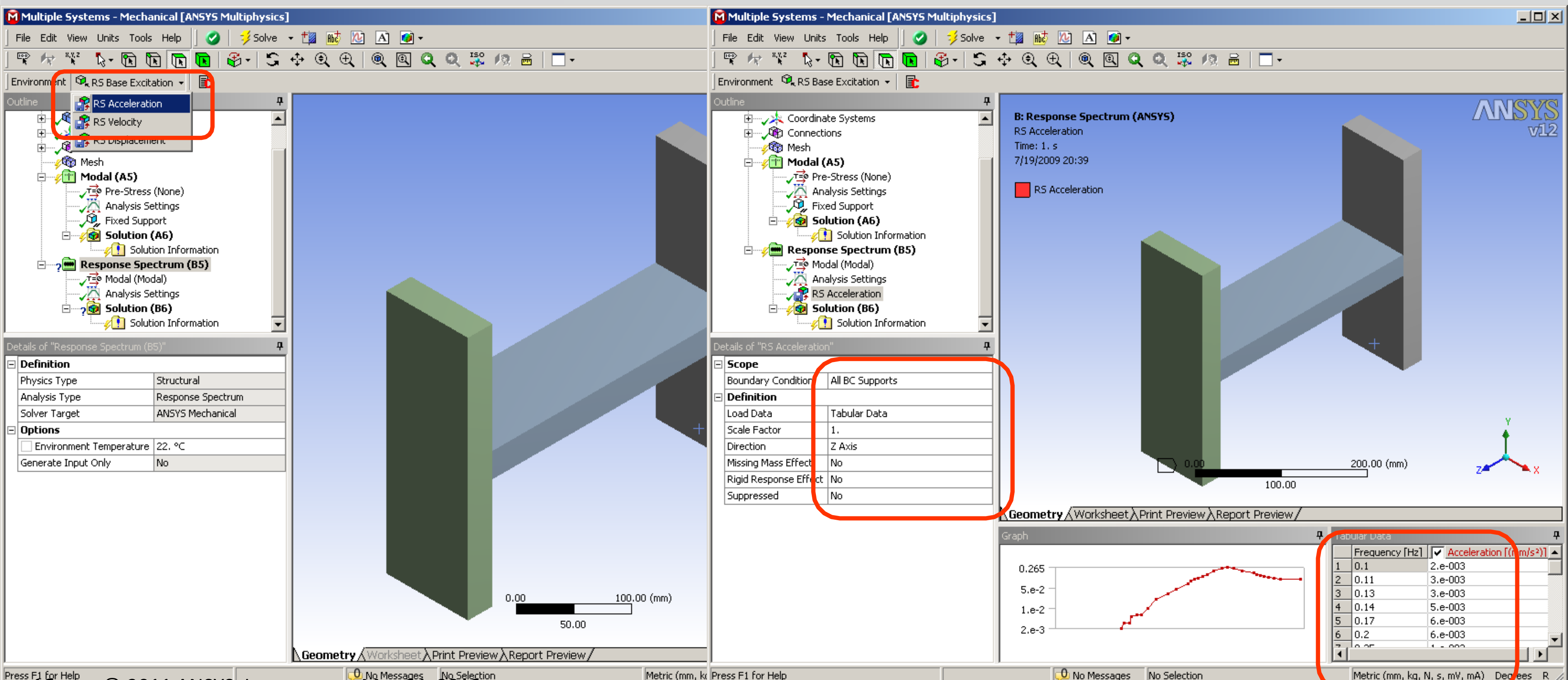
- Drop a Response Spectrum system onto the Solution cell of the Modal system.



Preprocessing

Insert an Acceleration, Velocity, or Direction response Input spectrum.

Set the Boundary Condition, Spectrum (Tabular) Data, and Direction.



Mode Combination

The phase information is lost in Response Spectrum method. Thus one need a way to combine the individual modal responses to get the combined response of the structure.

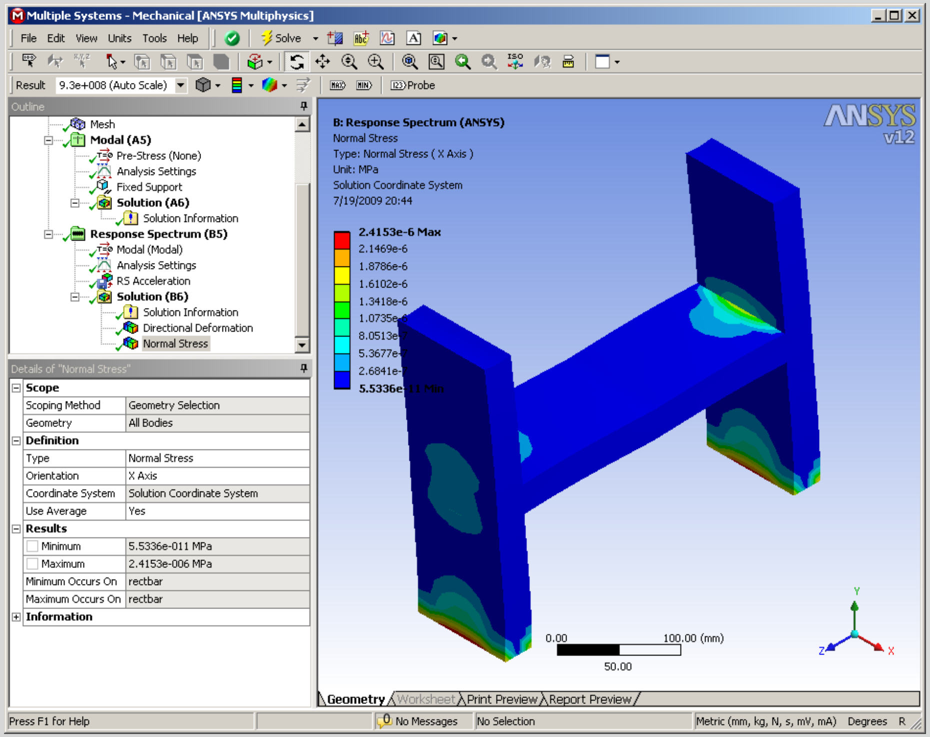
ANSYS have a number of different Mode combinations methods. The important/popular ones are described below:

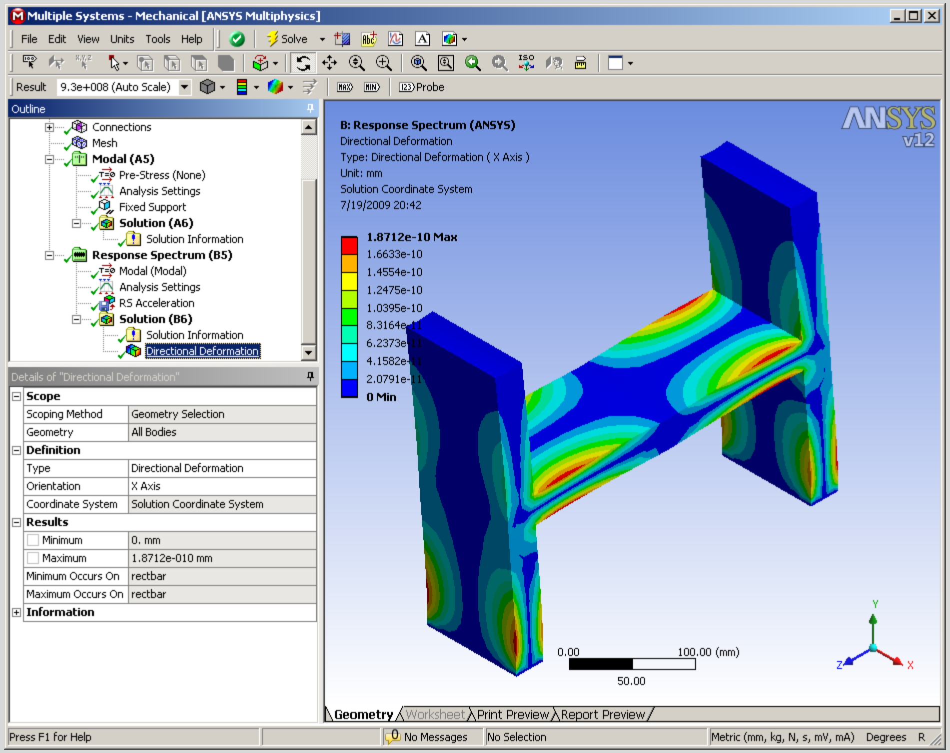
- The Square Root of Sum of Squares (**SRSS**) method is generally more conservative than the other methods.
- The Complete Quadratic (**CQC**) and the Rosenblueth (**ROSE**) methods providing a means of evaluating modal correlation for the response spectrum analysis.
  – **accounting for mode coupling makes the response estimate from these methods more realistic and closer to the exact time history solution**

Postprocessing

Results include

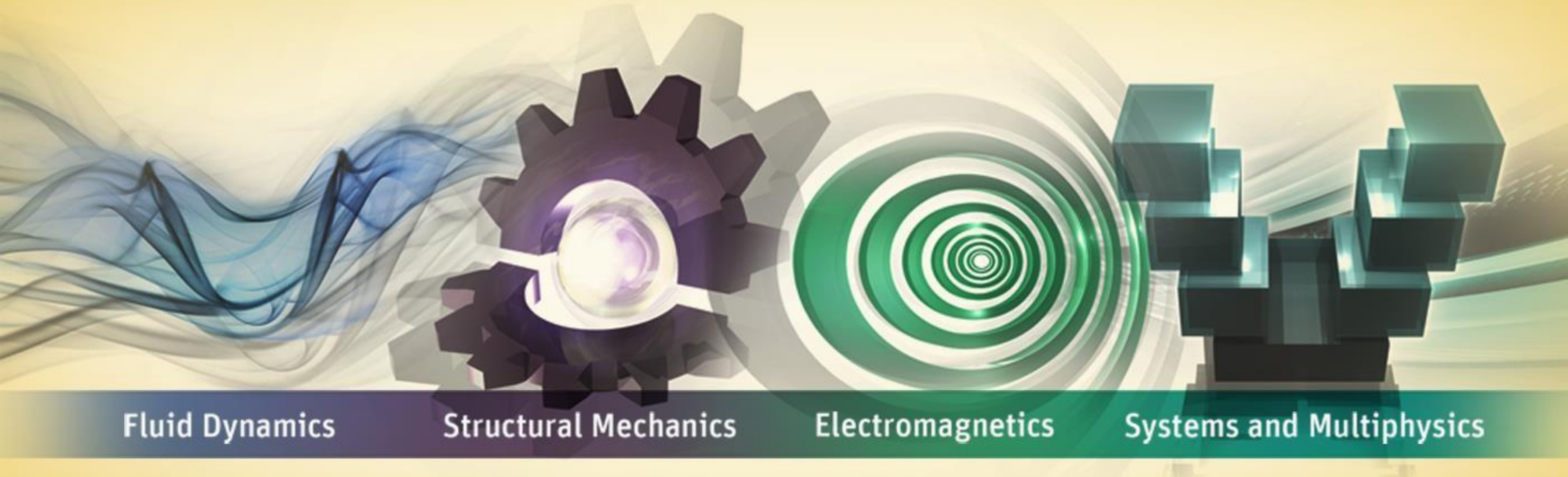
- Directional Deformation, Velocity, or Acceleration.
- Stress (normal, shear, equivalent) and Strain (normal, shear) results can also be reviewed.





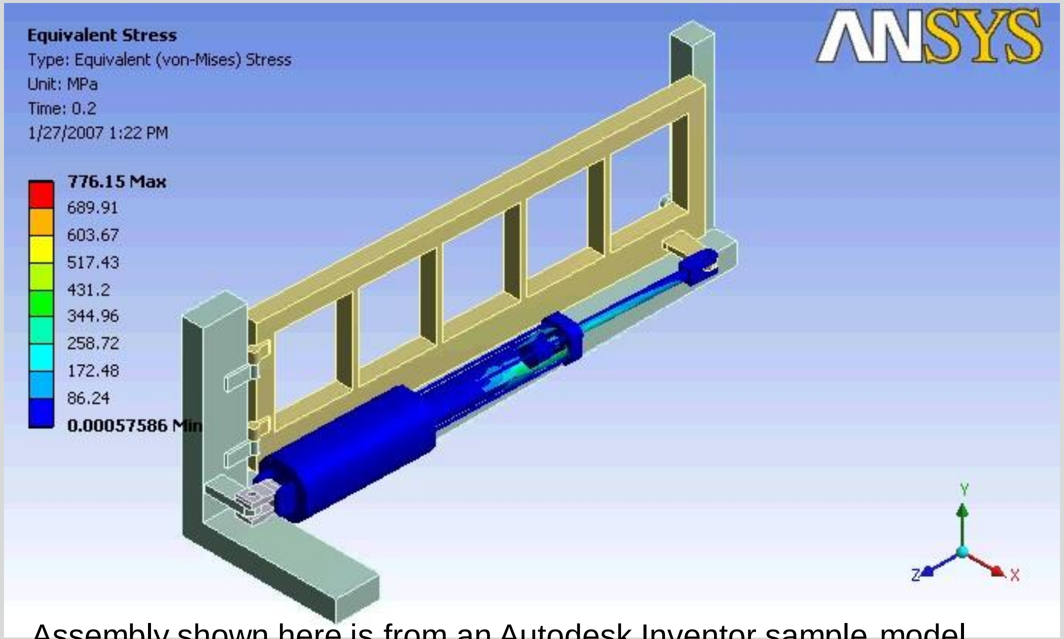
Realize Your Product Promise™

Transient Analysis



Overview

- ***Transient structural analysis*** **provides users with the ability to determine the dynamic response of the system under any type of time-varying loads.**
  - Unlike rigid dynamic analyses, bodies can be either rigid *or* flexible. For flexible bodies, nonlinear materials can be included, and stresses and strains can be output.
  - Transient structural analysis is also known as *time-history analysis* or *transient structural analysis*.

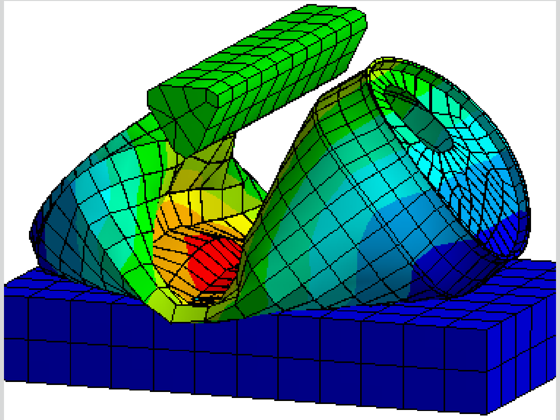


Assembly shown here is from an Autodesk Inventor sample model

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.

Implicit Dynamics Analysis details

Geometry

Damping – Same as previous section

Contacts

Time Step

Initial conditions

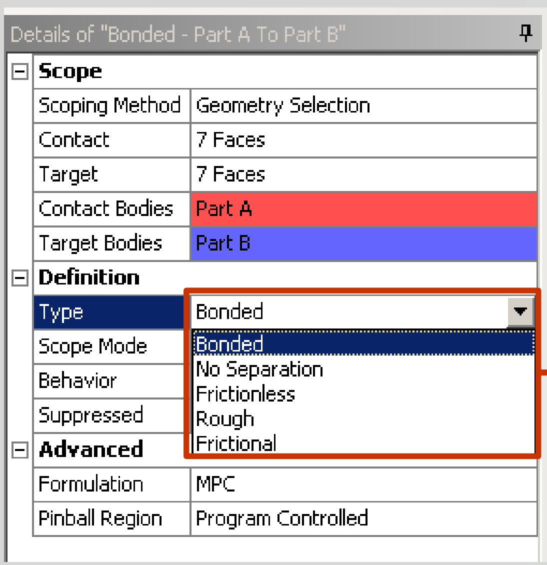
Load curve

Boundary conditions

Contact

In contact, parts are prevented from penetrating into each other. The different type of contact describe behavior in the *separation* and *sliding* directions:

Implicit dynamics analysis requires **all contacts** to be defined at the start of the analysis



	Normal Direction	Tangential Direction
Contact Type	**Separate**	**Slide**
Bonded	no	no
No Separation	no	yes
Rough	yes	no
Frictionless	yes	yes
Frictional	yes	yes (when $F_t \geq \mu N$)

Time Step Size

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/harmonic analysis should be performed prior to the transient structural analysis

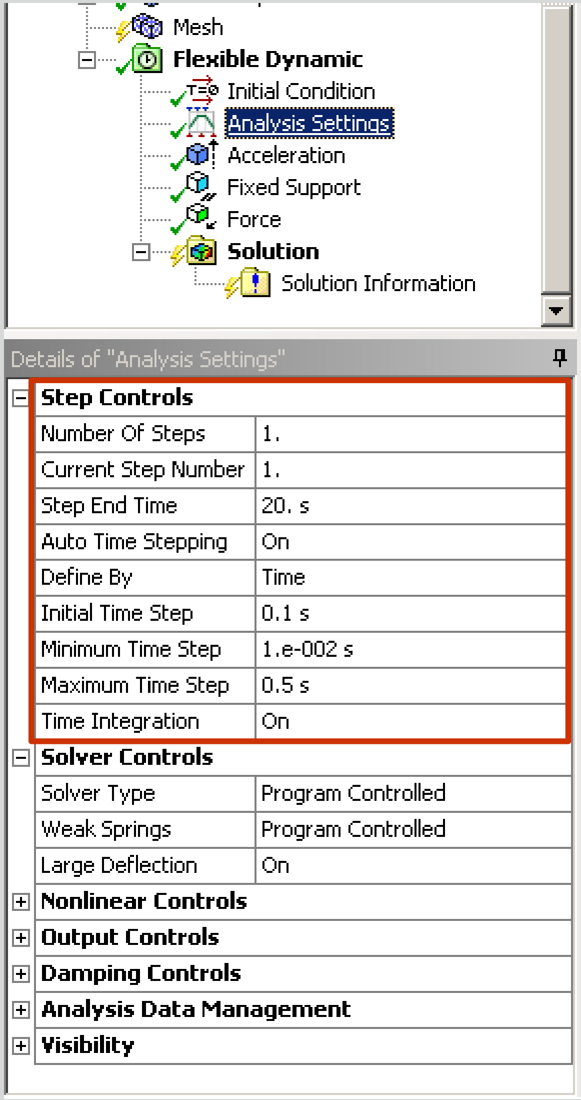
- **In this way, the user can determine what the mode shapes of the structure are (i.e.,how the structure may respond dynamically)**
- **The user can also then determine the value of $f_{response}$**

– If nonlinear effects dominate (i.e. contacts), the time step size may be dictated by nonlinear considerations rather than dynamic concerns.

Time Step Settings

It is important that the user specify the solution times in the “Step Controls” section

- The “Number of Steps” controls how the load history is divided.
- The “Step End Time” is the actual simulation ending time for the “Current Step Number”
- The initial, minimum, and maximum time steps should be defined to ensure
  - Accuracy
  - Convergence
  - Minimize solve time



Initial Conditions

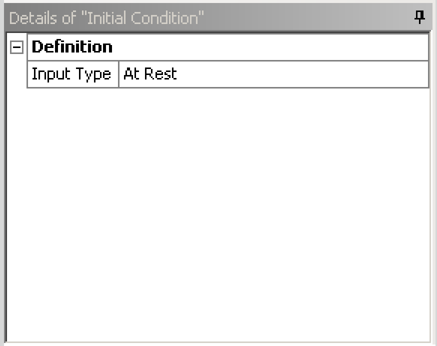
For a transient structural analysis, initial displacement and initial velocity is required:

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

Defining initial displacement & velocity with the "Initial Condition" object:

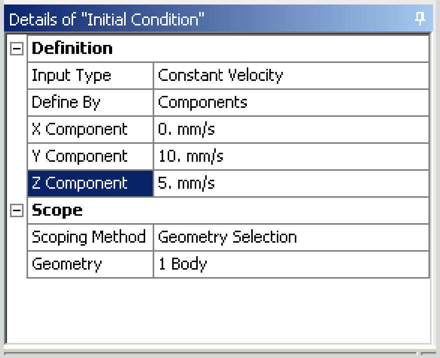
- Default condition is that all bodies are at rest
  - No additional action needs to be taken
- If some bodies have zero initial displacement but non-zero constant initial velocity, this can be input
  - Only bodies can be specified

– Enter constant initial velocity (Cannot specify more than one constant velocity value with this method)



Details of "Initial Condition"

Definition	
Input Type	At Rest



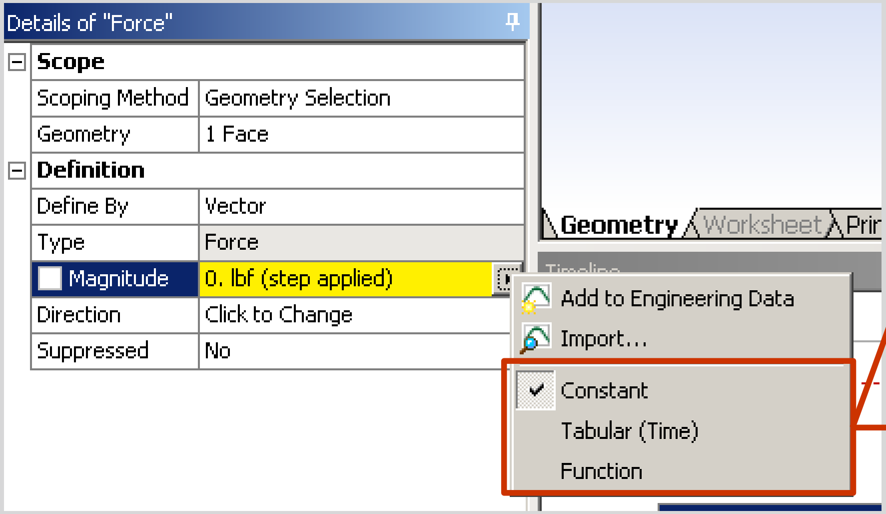
Details of "Initial Condition"

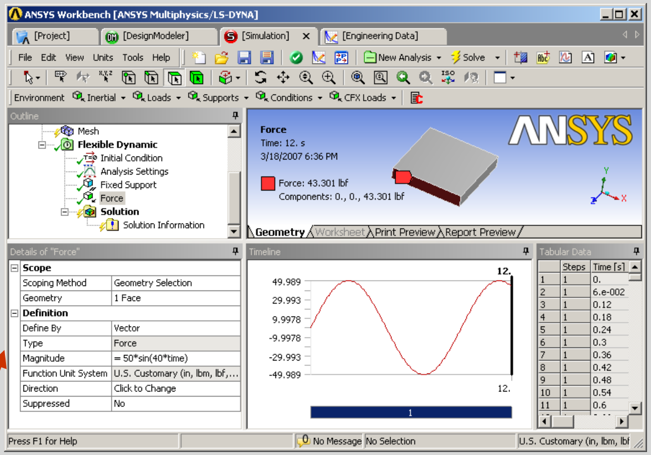
Definition	
Input Type	Constant Velocity
Define By	Components
X Component	0. mm/s
Y Component	10. mm/s
Z Component	5. mm/s
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body

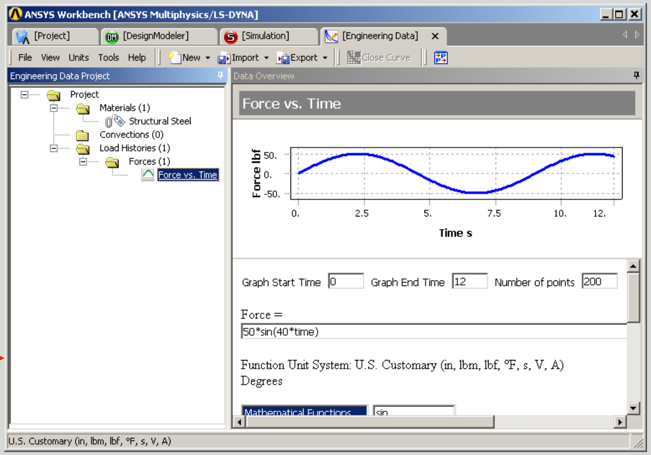
Time-Varying Loads

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 or entered in the Engineering Data page







  May 31, 2018

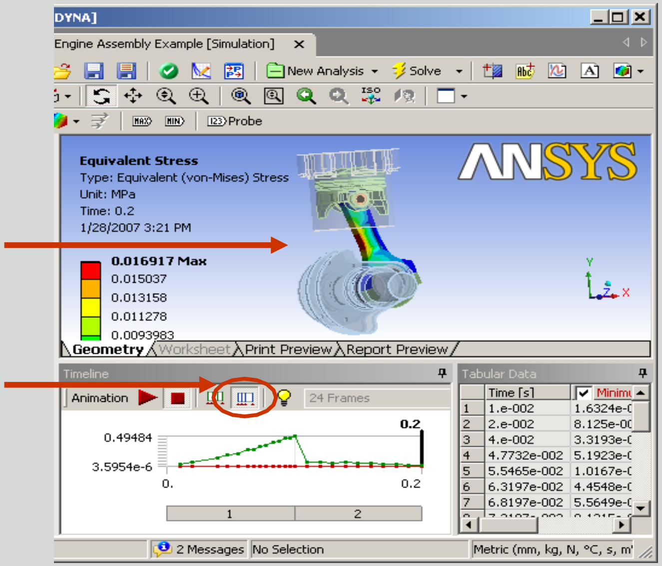
Reviewing Results

After completion of the solution, reviewing transient structural analysis results typically involves the following output:

- Contour plots and animations
- Probe plots and charts

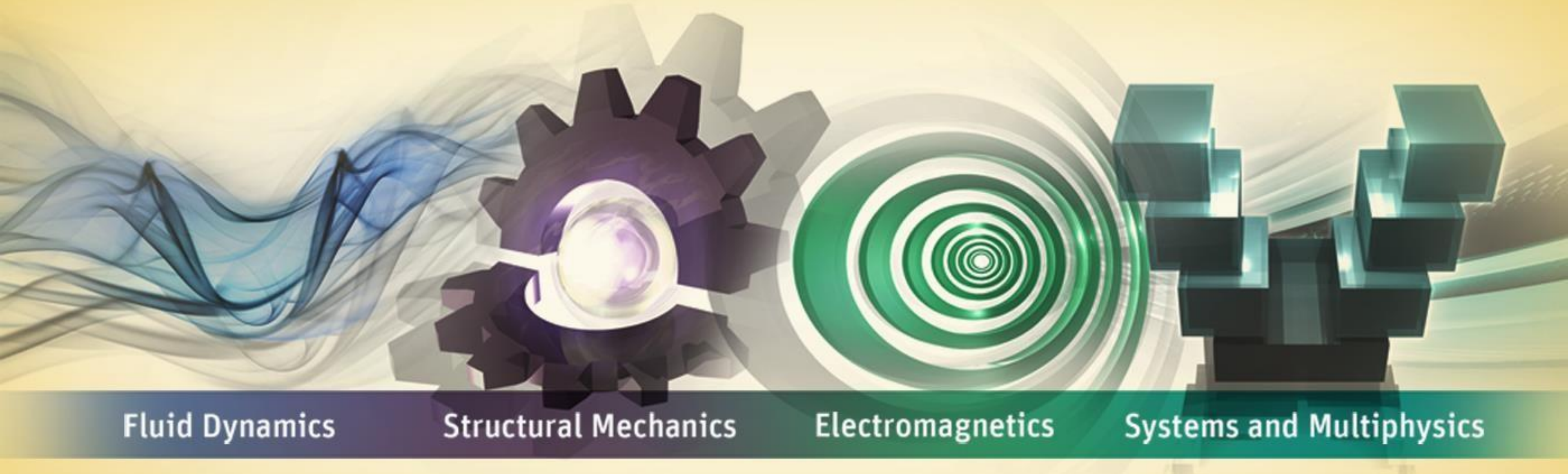
Generating contour plots and animations are similar to other structural analyses

- Note that the displaced position of rigid bodies will be shown in the contour result, but the rigid bodies will not show any contour result for deformation, stress, or strain since they are rigid entities
- Animations are generated using the actual result sets



Realize Your Product Promise™

Half sine shock on PCB



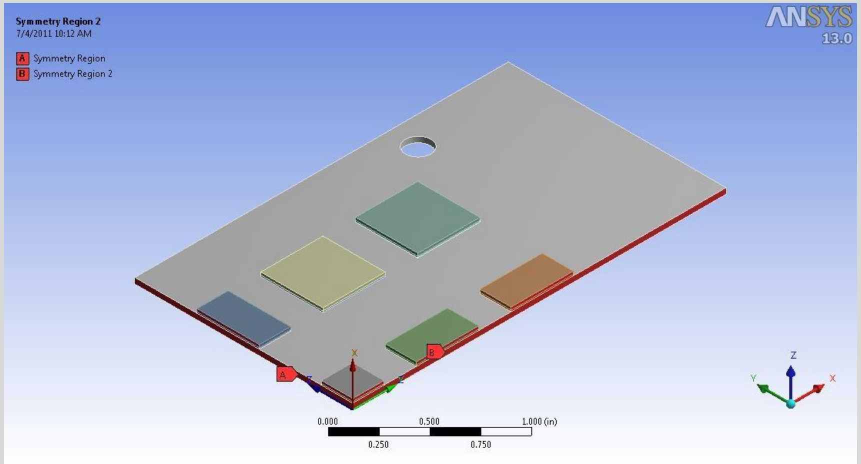
PCB model details

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

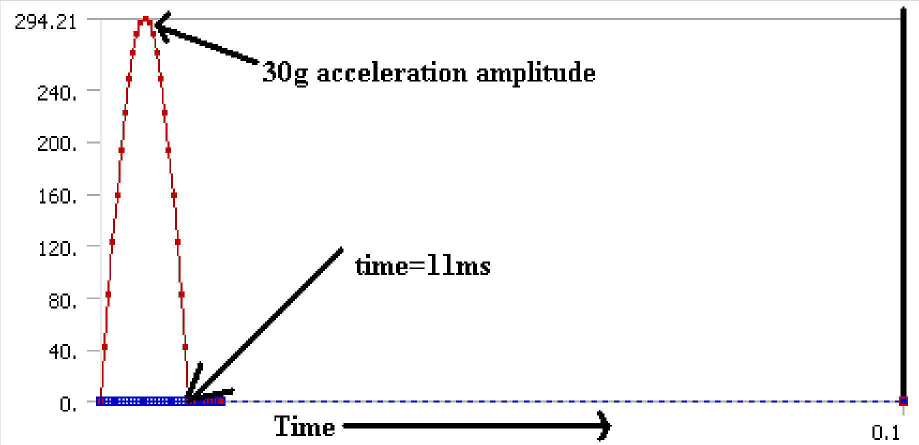
A real life example of a PCB subjected to standard 30g-11ms-half sine shock is considered as the basis of the study.

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 2.5e-2sec.

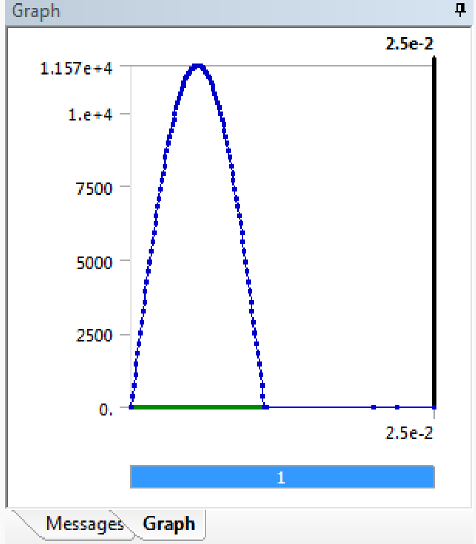
Initial conditions: Zero displacement and zero initial velocity.

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

Details of "Acceleration"

Scope	
Geometry	All Bodies
Definition	
Define By	Components
Coordinate System	Global Coordinate System
X Component	Tabular Data
Y Component	Tabular Data
Z Component	Tabular Data
Suppressed	No

Graph



Tabular Data

	Steps	Time [s]	X [in/s²]	Y [in/s²]	Z [in/s
1	1	0.	0.	0.	0.
2	1	1.1e-004	0.	0.	363.78
3	1	2.2e-004	0.	0.	728.35
4	1	3.3e-004	0.	0.	1090.6
5	1	4.4e-004	0.	0.	1452.8
6	1	5.5e-004	0.	0.	1811.
7	1	6.6e-004	0.	0.	2169.3
8	1	7.7e-004	0.	0.	2527.6
9	1	8.8e-004	0.	0.	2881.9
10	1	9.9e-004	0.	0.	3232.3
11	1	1.1e-003	0.	0.	3578.7
12	1	1.21e-003	0.	0.	3925.2
13	1	1.32e-003	0.	0.	4252.
14	1	1.43e-003	0.	0.	4606.3
15	1	1.54e-003	0.	0.	4921.3
16	1	1.65e-003	0.	0.	5275.6
17	1	1.76e-003	0.	0.	5590.6
18	1	1.87e-003	0.	0.	5905.5
19	1	1.98e-003	0.	0.	6220.5

Transient shock-load input data

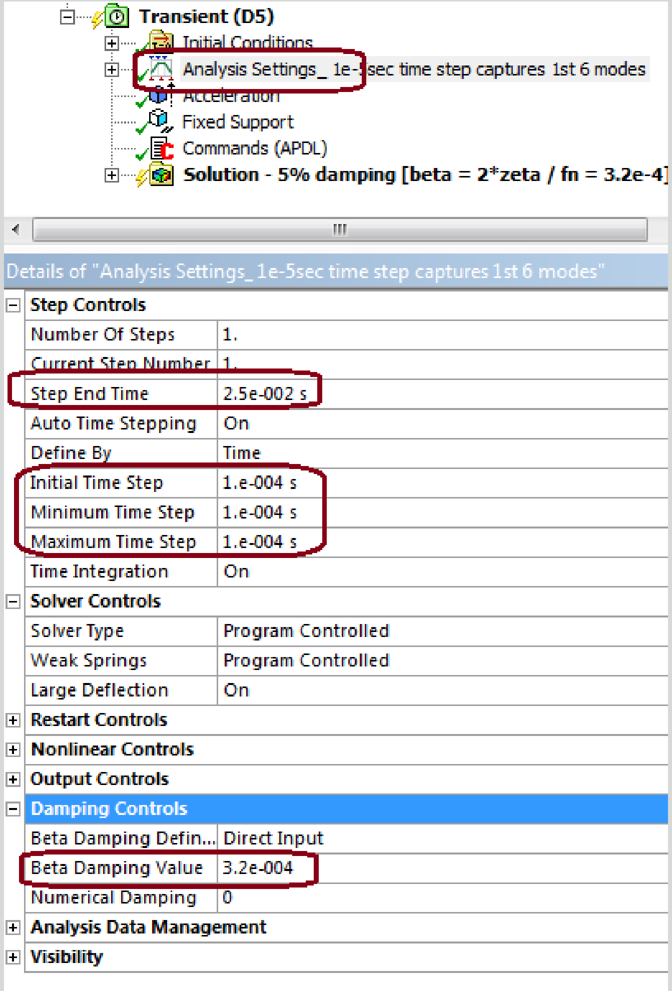
Analysis settings and time step definitions

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

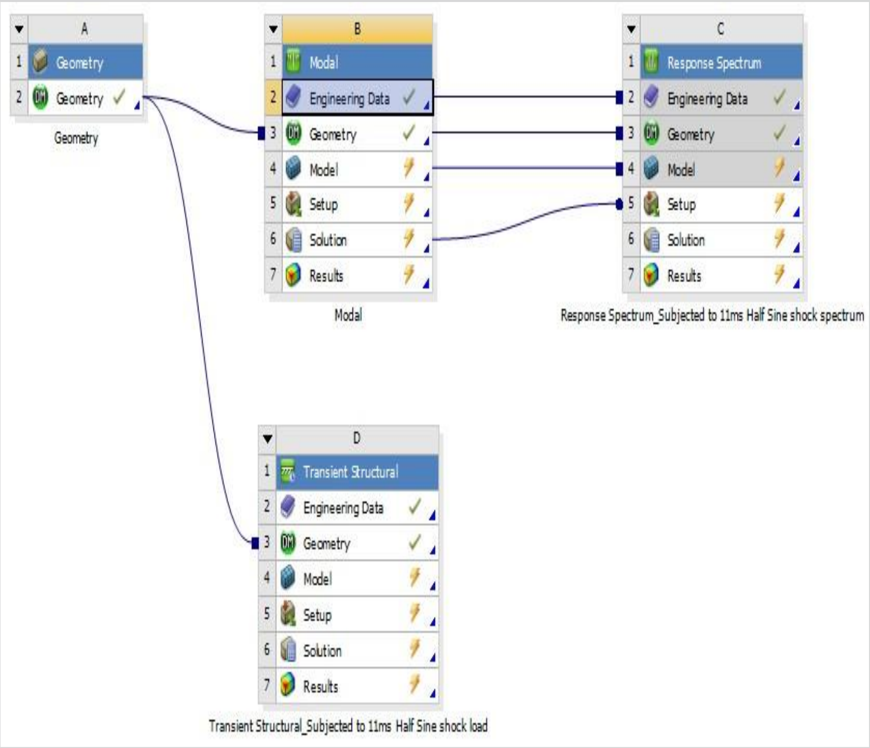
Including non-linear effects or not can also be defined.

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

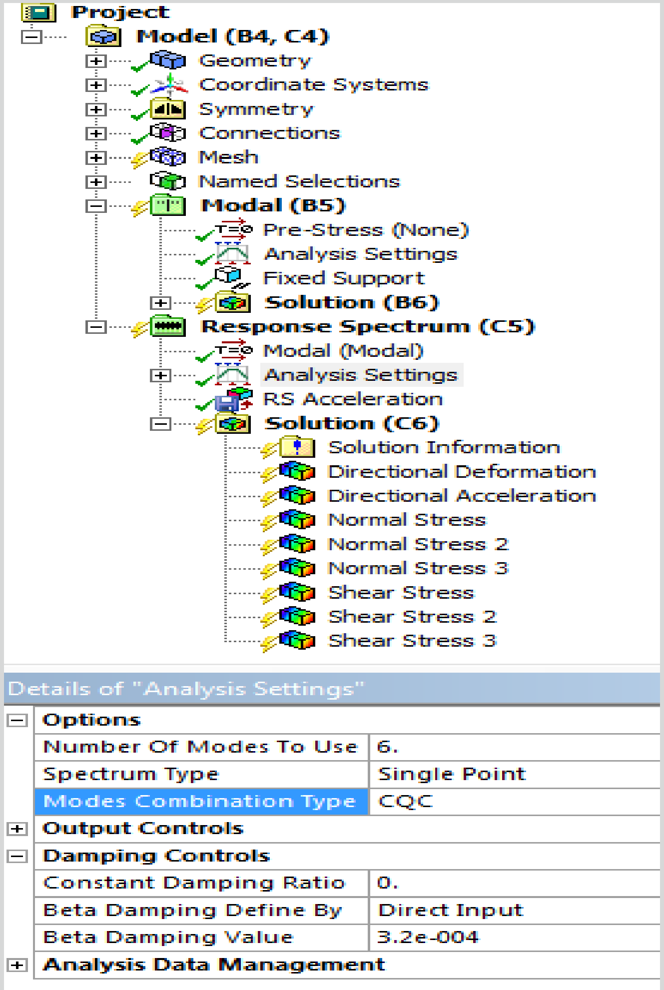
- 1st, 4th,5th mode freqs.=311, 3161, 4371 Hz, respectively.
- To capture 4th mode time step should be around (1/20*3161)=1.58e-5s. 1e-4 is taken as the time step for solution.



Spectrum analysis for Half-sine shock



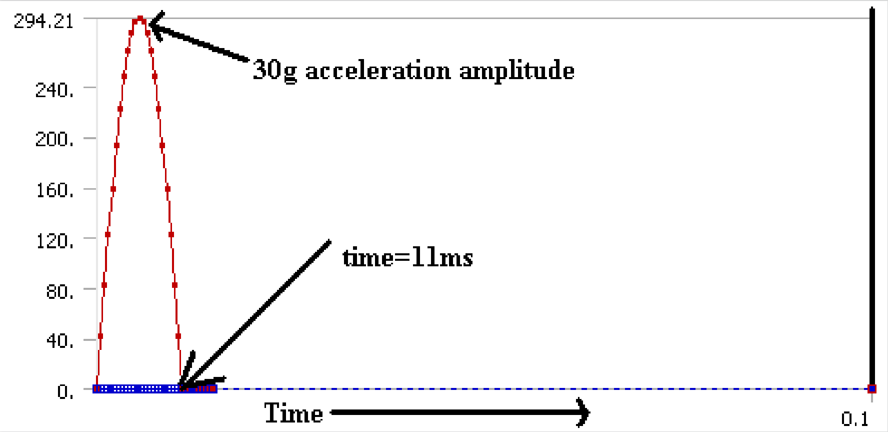
Project Schematic



Mechanical outline of Spectrum analysis

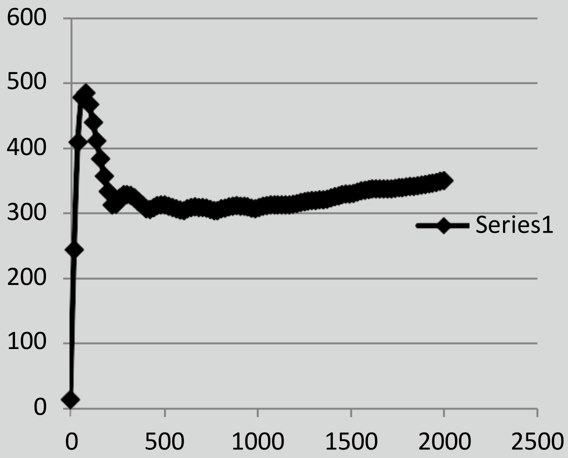
Converting Time domain load into Spectrum input

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



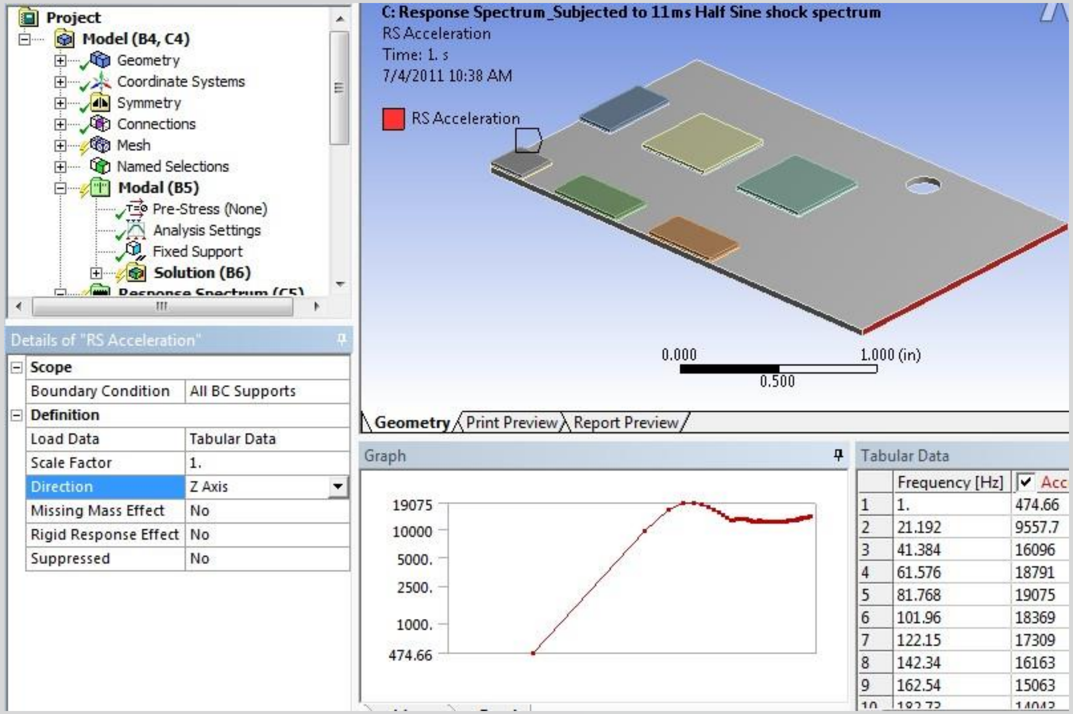
Time domain data
(Transient analysis)

RESP
command

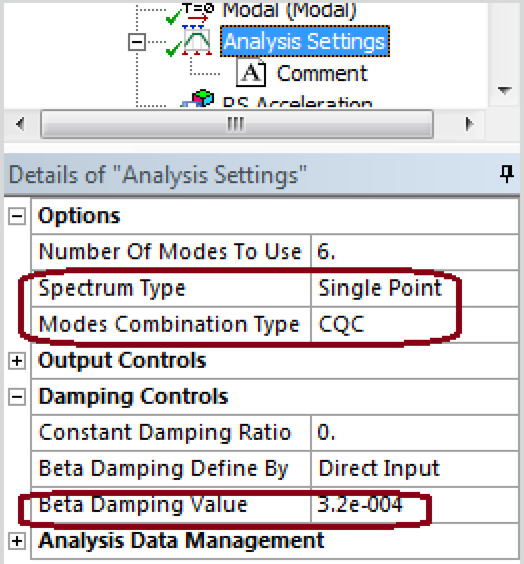


Frequency domain data
(Response Spectrum)

Applying acceleration Spectrum input in Spectrum analysis



RS acceleration input data



Settings of RS analysis

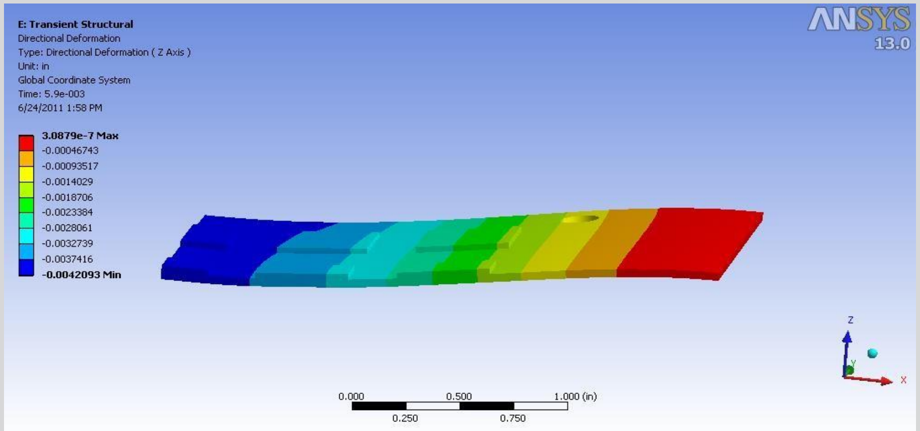
Transient versus RS

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

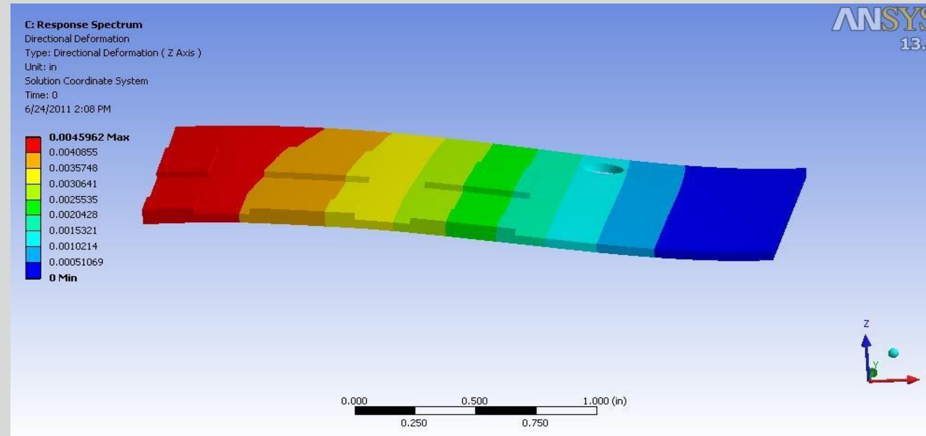
Transient results are assumed to be accurate and benchmark for accessing RS results.

Time taken for both the analysis are compared

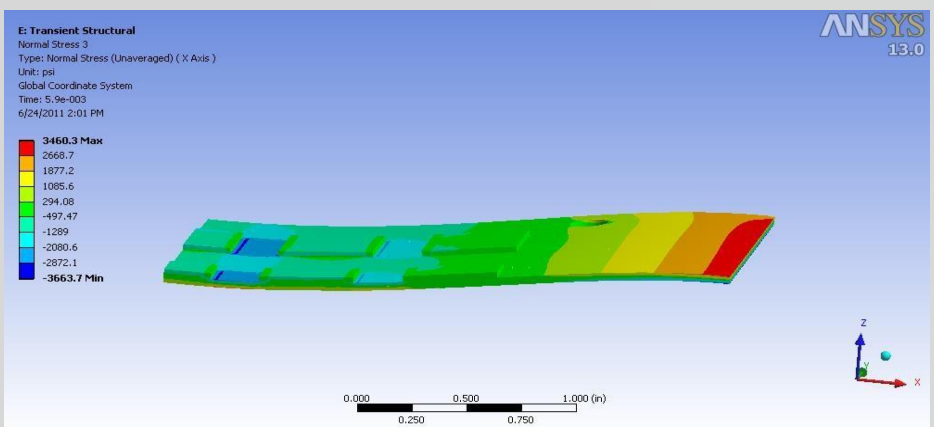
Results comparison- Maximum



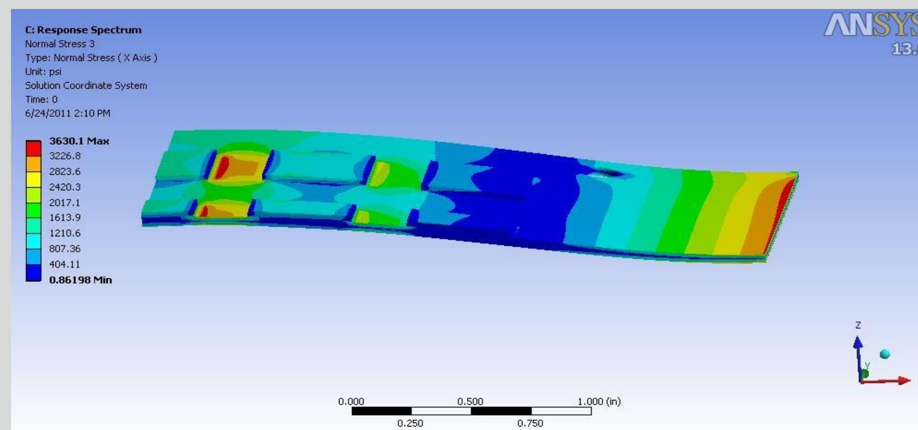
Transient directional deformation in z-dirn (Max= -4.2093in)



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

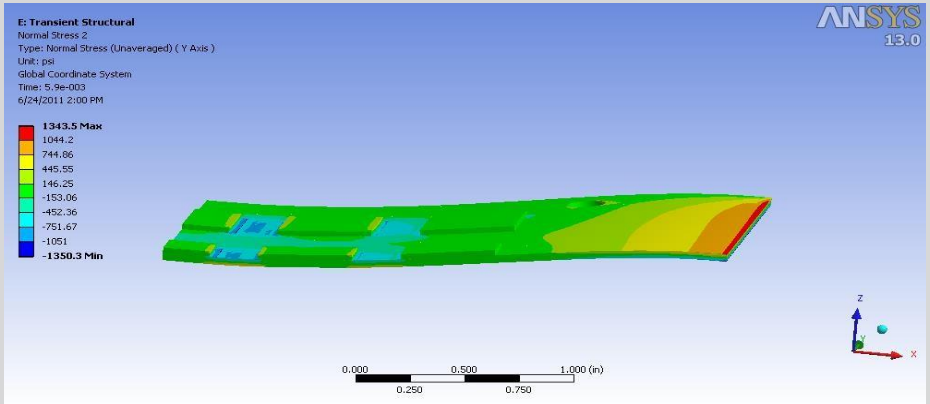


Transient Normal stress_x-dirn (Max=-3663.7psi)

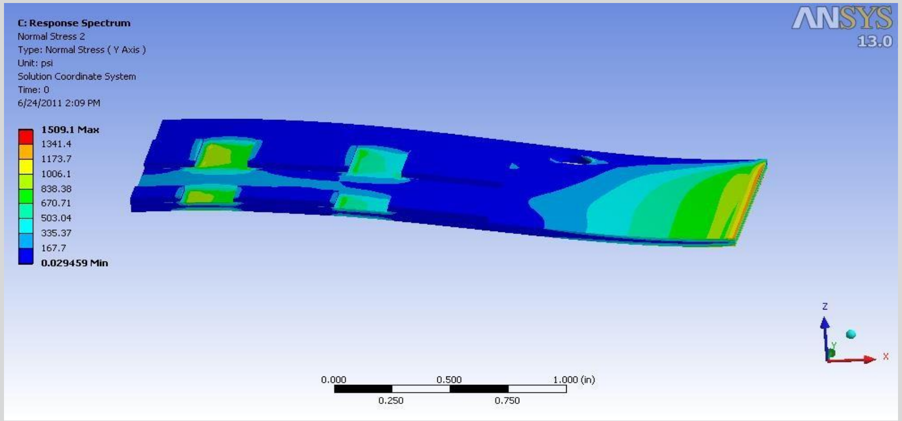


RS Normal stress_x-dirn (Max= 3630.1 psi)

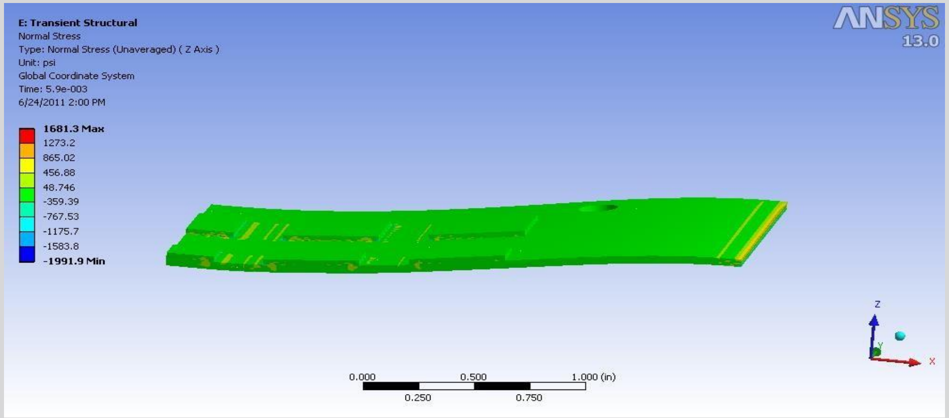
Results comparison- Maximum Normal Stress



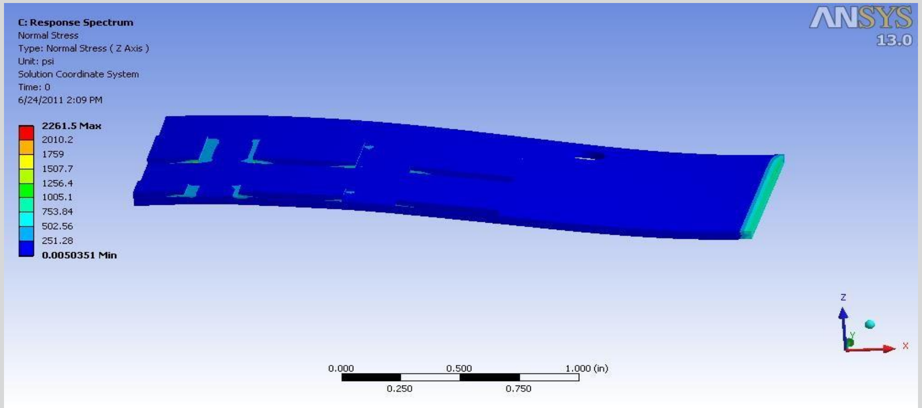
Transient Normal stress_y-dirn (Max= -1350.3 psi)



RS Normal stress_y-dirn (Max=1509.1 psi)

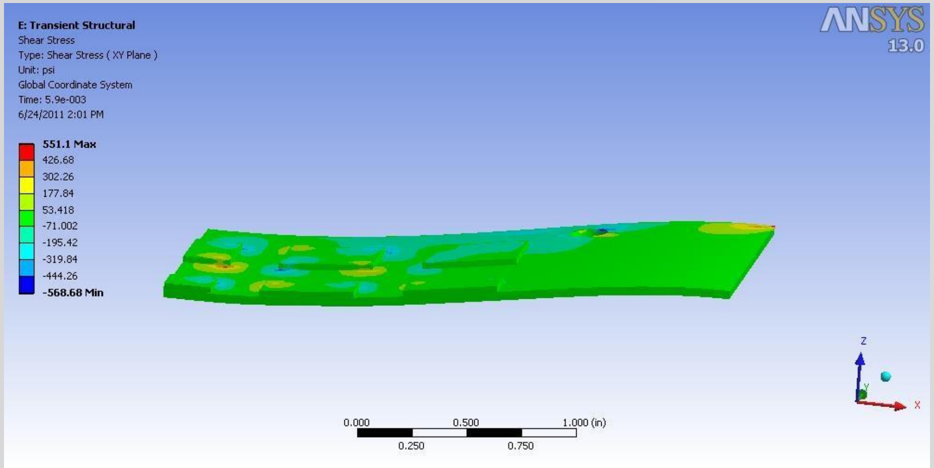


Transient Normal stress_z-dirn (Max= -1991.9 psi)

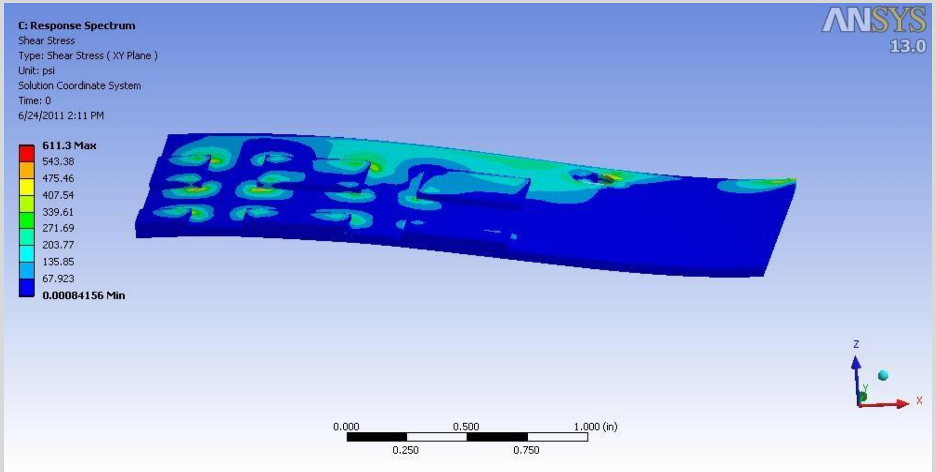


RS Normal stress_z-dirn (Max=2261.5 psi)

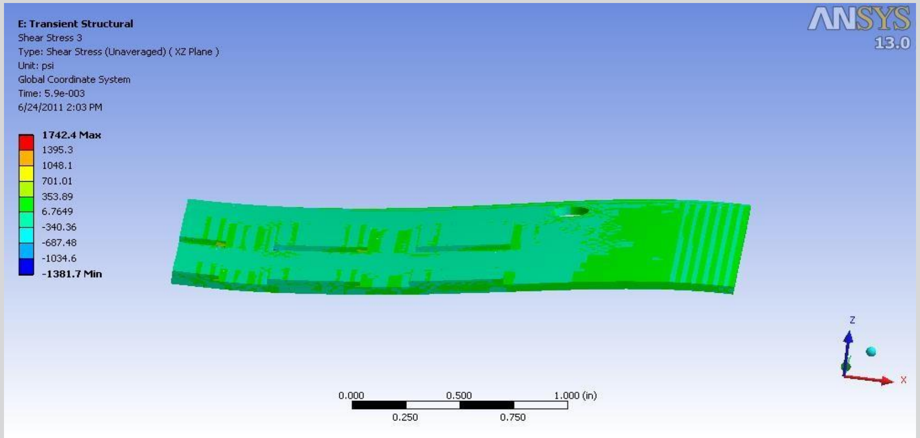
Results comparison- Shear stresses



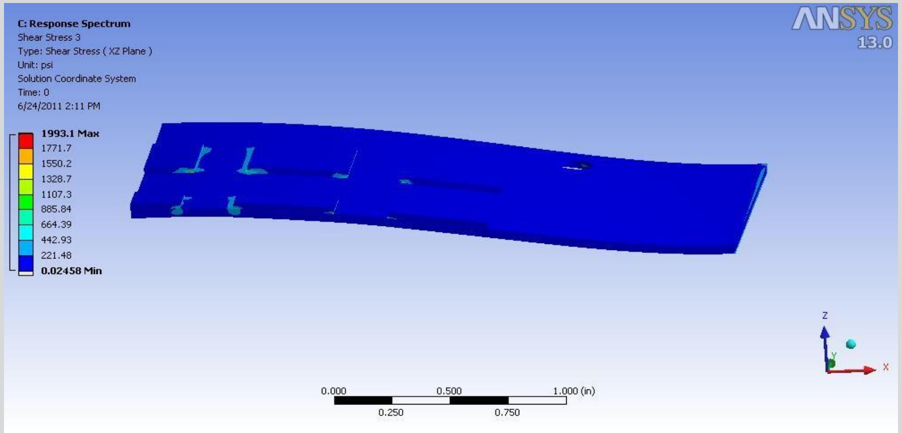
Transient Shear stress_xy-dirn (Max= -568.68psi)



RS Shear stress_xy-dirn (Max= 611.3psi)

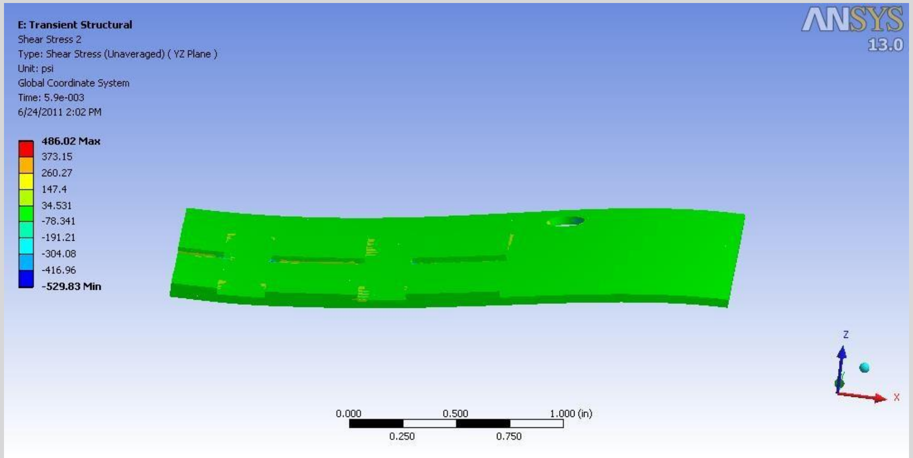


Transient Shear stress_xz-dirn (Max= 1742.4 psi)

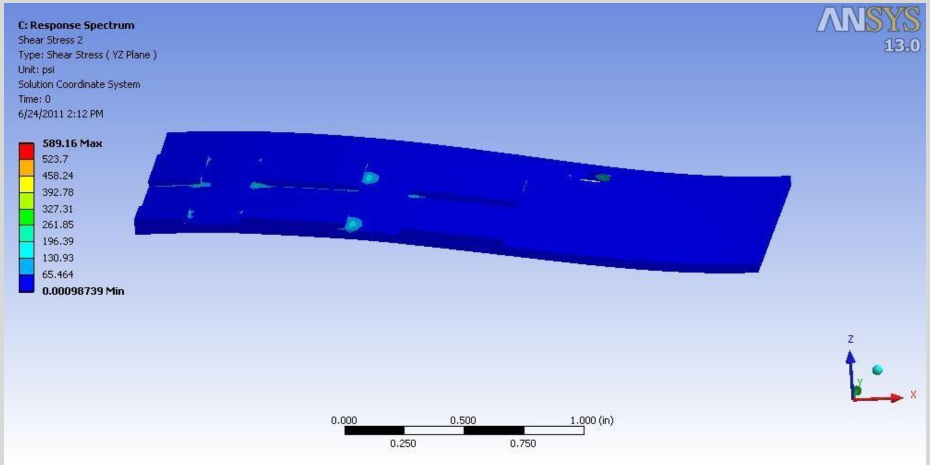


RS shear stress_xz-dirn (Max= 1993.1psi)

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)
Full Transient	-4.2093 e-3	- 3663.7	-1350.3	1991.9	- 568.68	1742.4	- 529.83	7192
MSUP Transient	-4.1766e-003	-3485.3 psi						49+136= 185
Spectrum (CQC)	4.5962e-3	3630.1	1509.1	2261.5	611.3	1993.1	589.16	49+8.375= 57.375
Percentage difference	9	0.09	11.3	13.5	7.4	14	11.1	7192/57.375= 125 times

ANSYS

Comments for usage in Electronic 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 indispensible.