Shock Analysis

MingYao Ding
Applications Engineer
ANSYS Inc.
Mingyao.ding@ansys.com
Contents

Shock Analysis Introduction
Response Spectrum Analysis
Transient (Implicit) Dynamics analysis
Half Sine shock example
Types of Vibrations

- Vibrations
  - Stationary
    - Deterministic
    - Random
  - Non-Stationary
    - Continuous
    - Transient
- Shock
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
Response Spectrum Analysis
A response-spectrum analysis calculates the **maximum response** of a structure to a transient load.

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:

• Nuclear power plant buildings and components, for seismic loading
• Airborne Electronic equipment for shock 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

Calibrate Damping

Response Spectrum
(Input in Freq. domain)

Mil STD

Loading History (g vs t)
(Input in Time domain)

Convolution or FFT Method (RESP command)

Response spectrum analysis
Viscous damping

- The value of $c$ in
  
  $F_d = c\dot{u} = i\omega_n u$

  can be input directly as element damping (Details section of Spring connection).

The value of $\beta$ in

$F_d = \beta k\dot{u} = i\beta k \omega_n u$

can be input directly as global damping value (Details section of Analysis Settings) or as material-dependent damping value (Material Damping Factor material property).
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.

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.

Convolution Integral or FFT

RESP command
• 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.
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*.
Implicit vs Explicit Dynamics?

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

<table>
<thead>
<tr>
<th>Solution</th>
<th>Impact Velocity (m/s)</th>
<th>Strain Rate (/s)</th>
<th>Effect</th>
</tr>
</thead>
<tbody>
<tr>
<td>Implicit</td>
<td>&lt; 50</td>
<td>&lt;10^{-5}</td>
<td>Static / Creep</td>
</tr>
<tr>
<td></td>
<td>50 - 1000</td>
<td>10^{-5} - 10^{-1}</td>
<td>Elastic</td>
</tr>
<tr>
<td></td>
<td>1000 - 3000</td>
<td>10^{-1} - 10^{1}</td>
<td>Elastic-Plastic (material strength significant)</td>
</tr>
<tr>
<td></td>
<td>3000 - 12000</td>
<td>10^{5} - 10^{6}</td>
<td>Primarily Plastic (pressure equals or exceeds material strength)</td>
</tr>
<tr>
<td>Explicit</td>
<td>&gt; 12000</td>
<td>&gt; 10^{8}</td>
<td>Hydrodynamic (pressure many times material strength)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Vaporization of colliding solids</td>
</tr>
</tbody>
</table>
Implicit vs Explicit Dynamics
Time step size

Implicit – no stability limit on time step size

Explicit -

\[ \Delta t \leq f \times \left[ \frac{h}{c} \right]_{\text{min}} \]

Where
- \( \Delta t \) is the time increment,
- \( f \) is the stability time step factor (\( \approx 0.9 \) by default),
- \( h \) is the characteristic dimension of an element and
- \( c \) is the local material sound speed in an element
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

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.

<table>
<thead>
<tr>
<th>Contact Type</th>
<th>Normal Direction</th>
<th>Tangential Direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bonded</td>
<td>no</td>
<td>no</td>
</tr>
<tr>
<td>No Separation</td>
<td>no</td>
<td>yes</td>
</tr>
<tr>
<td>Rough</td>
<td>yes</td>
<td>no</td>
</tr>
<tr>
<td>Frictionless</td>
<td>yes</td>
<td>yes</td>
</tr>
<tr>
<td>Frictional</td>
<td>yes (when $F_t \geq \mu N$)</td>
<td>yes</td>
</tr>
</tbody>
</table>
Time Step Size

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

\[ \Delta t_{\text{initial}} = \frac{1}{20 f_{\text{response}}} \]

where \( f_{\text{response}} \) 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 \( f_{\text{response}} \)

- 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)
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.
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
Thank You
Half sine shock example
Half sine shock

A cantilever beam subjected to Half-sine shock of 11ms.
Material properties and Boundary Conditions

- Initial conditions: Zero displacement and zero initial velocity.
- Fixed BC: The cantilever is fixed at one end.

![Properties of Outline Row 3: Structural Steel](image)

Material properties defined in Engineering data
Define Loading:

Acceleration Shock load: 30g-11ms, Half-sine shock pulse in x-dir.

Transient analysis is done for a total time of 0.1 sec.

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. = 31.551, 229.01, 544.08 Hz, respectively.
- To capture 4th mode time step should be around (1/20*229) = 22 ms
Results

X-dirn Acceleration response

Max. acceleration response (m/s²)

Time (sec)

X-dirn Deformation response

Max X-dirn deformation (m)
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)**
- **Frequency domain data (Response Spectrum)**
Applying acceleration Spectrum input in Spectrum analysis

RS acceleration input data

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

The elapsed time in RS is quite less than in the Transient run.

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

<table>
<thead>
<tr>
<th>Analysis Type</th>
<th>Max X-dirn deformation (m)</th>
<th>Max X-dirn acceleration (m/s²)</th>
<th>Elapsed time (sec) (8GB, 64bit_machine)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full Transient</td>
<td>-1.0979e-2 @12ms</td>
<td>620.8 @12ms</td>
<td>265</td>
</tr>
<tr>
<td>Spectrum (SRSS)</td>
<td>1.0887e-2 (0.83% error)</td>
<td>662.22 (6.67% error)</td>
<td>13 (modal) +5 (RS) =18</td>
</tr>
<tr>
<td>Spectrum (CQC)</td>
<td>1.0879e-2 (0.91% error)</td>
<td>625.13 (0.69% error)</td>
<td>13 (modal) +6 (RS) =19</td>
</tr>
<tr>
<td>Spectrum (ROSE)</td>
<td>1.0890e-2 (0.81% error)</td>
<td>673.98 (8.5% error)</td>
<td>13 (modal) +4 (RS) =17</td>
</tr>
</tbody>
</table>

*Comparison of Transient vs RS(diff mode combinations) results*