

Recent updates in LS-DYNA[®] frequency domain solvers

Yun Huang, Zhe Cui

Livermore Software Technology Corporation

1 Introduction

A series of frequency domain features have been implemented to LS-DYNA, since the release of ls971 R6. The frequency domain features include

- FRF (Frequency Response Functions)
- SSD (Steady State Dynamics) and fatigue
- Random vibration and fatigue
- BEM (Boundary Element Method) Acoustics
- FEM (Finite Element Method) Acoustics
- Response Spectrum Analysis

The frequency domain features are developed to provide solutions to the vibration, acoustic and fatigue problems coming from a wide range of industrial areas. A typical application is NVH analysis for automobiles.

A bunch of updates have been made for the frequency domain features, since the last (13th) International LS-DYNA Users' Conference, Detroit, June 8-10 2014. The main updates include

- Incorporation of new boundary conditions in FEM acoustic analysis
- Incorporation of frequency dependent complex sound speed in acoustic analysis
- Acoustic eigenvalue analysis by FEM
- Incorporation of new stress indexes in SSD fatigue analysis
- Incorporation of multi-slope S-N fatigue curve for fatigue analysis
- Introduction of strain output in random vibration analysis

These updates in frequency domain solvers were made to meet the requirements from users, and to improve the capabilities of frequency domain analysis by LS-DYNA.

This paper provides a brief review on the updates.

2 FEM Acoustic analysis with new boundary conditions incorporated

For *FREQUENCY_DOMAIN_ACOUSTIC_FEM, two new boundary conditions are incorporated, in addition to the existing velocity boundary condition. The two new boundary conditions are

- acoustic pressure
- impedance

With all the 3 types of boundary conditions incorporated, LS-DYNA can provide complete solutions to general interior acoustic problems, which can be encountered in auto NVH analysis. The 3 types of boundary conditions can be present in one model, as shown in Figure 1.

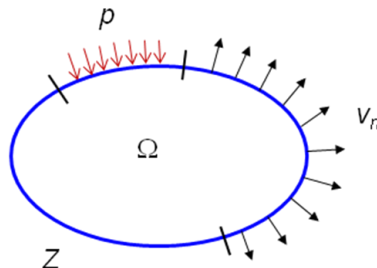


Fig. 1: An acoustic volume Ω with 3 types of boundary conditions (p : pressure, v_n : normal velocity and Z : impedance).

To validate the FEM acoustic solver, especially the incorporation of the new boundary conditions, an auto compartment model is adopted as case study here. This model has 114221 nodes and 643619 tetrahedron elements. Two loading cases are considered.

For the first case, the compartment model is subjected to the following two boundary conditions:

- Constant base excitation 4 mm/s for 10-500 Hz (see Figure 2)
- Open windows (which means the pressure is 0) (see Figure 3)

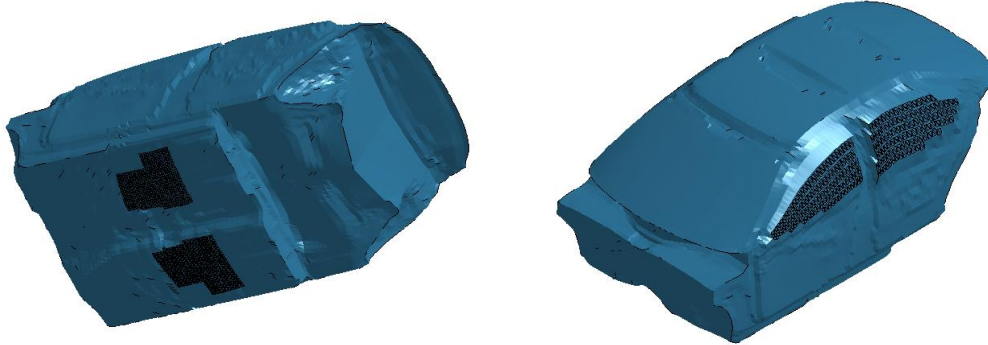


Fig.2: Base velocity at the shaded area (to model the excitation from ground). Fig.3: Opened windows

For a selected field point in the acoustic volume, the acoustic pressure (Pa), and Sound Pressure Level (SPL, dB) in the frequency range 10-500 Hz are computed using LS-DYNA, NASTRAN and ANSYS for cross validation. The results (SPL, dB) by the three softwares are plotted in Figure 4. One can see that a good match between the three softwares is achieved.

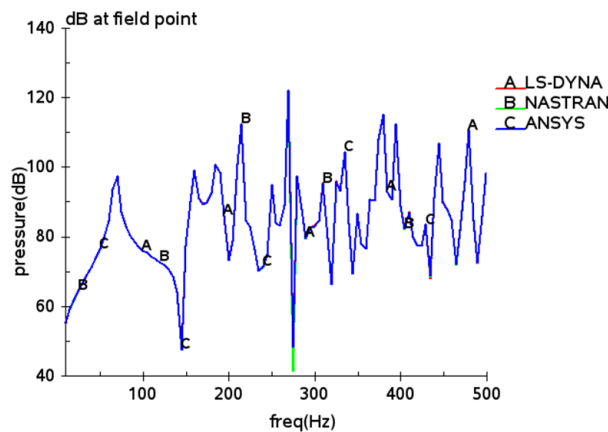


Fig.4: SPL (dB) by LS-DYNA, NASTRAN and ANSYS

To see the fringe plot of the acoustic pressure, one can use the binary plot database D3ACS. D3ACS is accessible to LS-PrePost®. With LS-PrePost, the acoustic pressure (real part, imaginary part and magnitude) and SPL (dB) can be plotted for a range of frequencies requested by user. Figures 5 and 6 show the acoustic pressure (magnitude) and SPL (dB) of the acoustic volume at frequency 100 Hz.

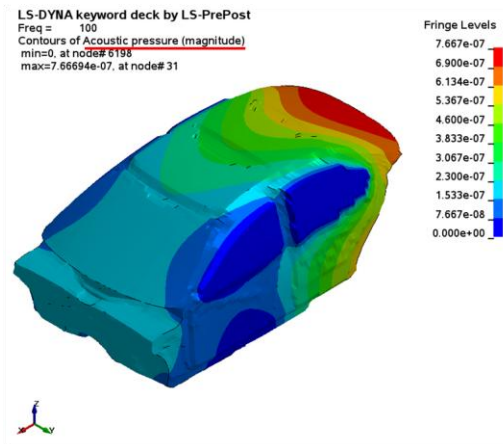


Fig.5: Acoustic pressure (magnitude) at frequency 100 Hz.

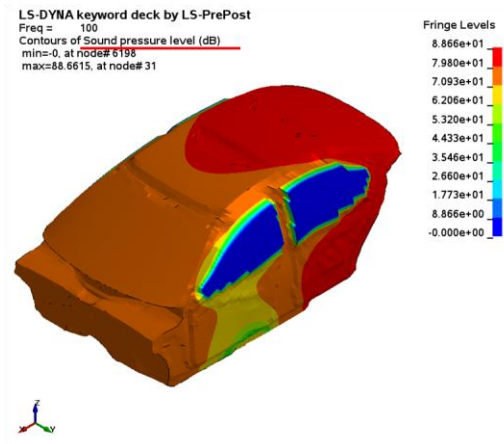


Fig.6: SPL at frequency 100 Hz.

The similar results at frequency 300 Hz are plotted in Figures 7 and 8.

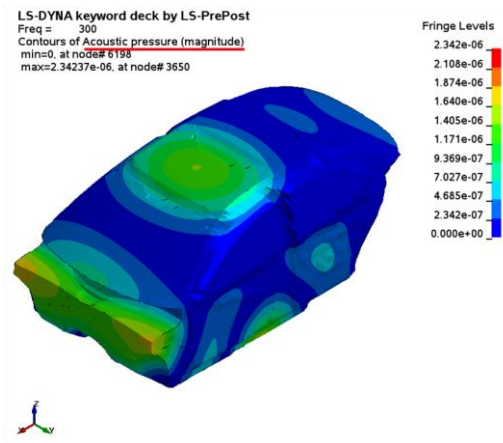


Fig.7: Acoustic pressure (magnitude) at frequency 300 Hz.

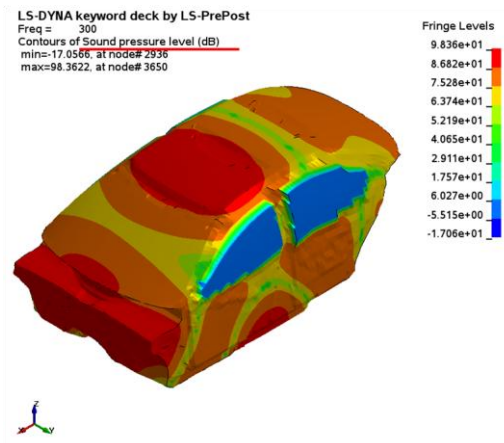


Fig.8: SPL at frequency 300 Hz.

As one can see, the pressure at the window areas is 0, which complies with the prescribed boundary condition.

For the 2nd case, the acoustic volume is subjected to the following two boundary conditions:

- Constant base excitation 4 mm/s for 10-500 Hz (see Figure 2)
- Impedance boundary condition on the roof (see Figure 9)

The impedance boundary condition is used to model the sound absorbing material in contact with the acoustic medium. It is a kind of passive boundary condition. The characteristic impedance is defined as

$$z = \frac{p}{v_n} = \rho c \tag{1}$$

For the compartment model shown in Figure 9, an impedance boundary condition is assumed for some area on the roof, to model the sound absorbing layer (foam, or fiberglass, etc.). Both LS-DYNA and NASTRAN are used to compute the SPL (dB) at a selected field point and their results are compared in Figure 10.

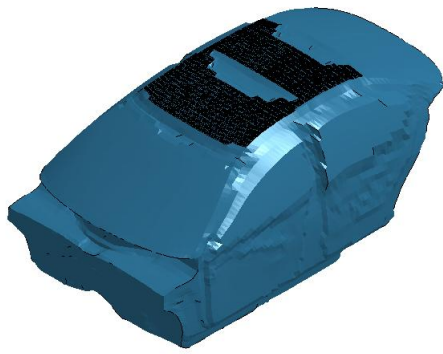


Fig.9: An auto compartment with impedance b.c. on the roof.

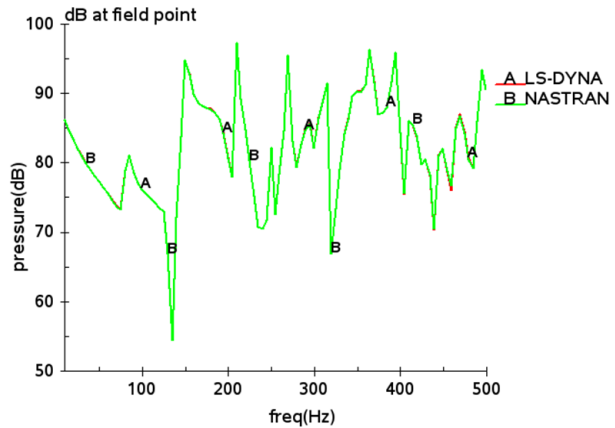


Fig.10: SPL (dB) by LS-DYNA and NASTRAN.

3 Acoustic computation with frequency dependent complex sound speed

To consider the damping in the acoustic medium, a frequency dependent complex sound speed can now be considered in FEM or BEM acoustic computation in LS-DYNA. A new keyword `*FREQUENCY_DOMAIN_ACOUSTIC_SOUND_SPEED` has been implemented so that user can define the frequency dependent complex sound speed. Two curves are needed to define the frequency dependent complex sound speed (one for the real part of the sound speed vs frequency, and the other for the imaginary part of the sound speed vs frequency). For more details about this keyword, please refer to the latest version LS-DYNA Keyword User's Manual [1].

To use the frequency dependent complex sound speed, one needs to define `C = -LCID` in the keyword `*FREQUENCY_DOMAIN_ACOUSTIC_BEM` or `*FREQUENCY_DOMAIN_ACOUSTIC_FEM`, where LCID is the curve ID defining the frequency dependent complex sound speed.

In the following example, a simple muffler model is considered (Figure 11). In computing the acoustic pressure at one field point in the volume, two cases of sound speed are considered to study the effect of damping in the acoustic system. For the first one, the sound speed is real, constant (340 m/s); for the second one, the sound speed is defined as frequency dependent complex variables (with both real and imaginary parts).

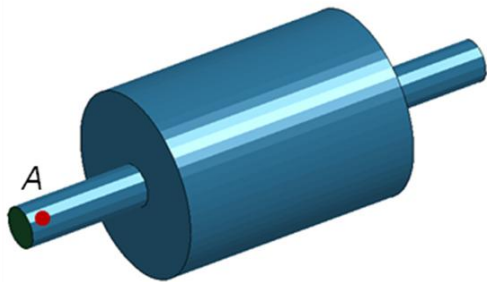


Fig.11: A simple muffler model.

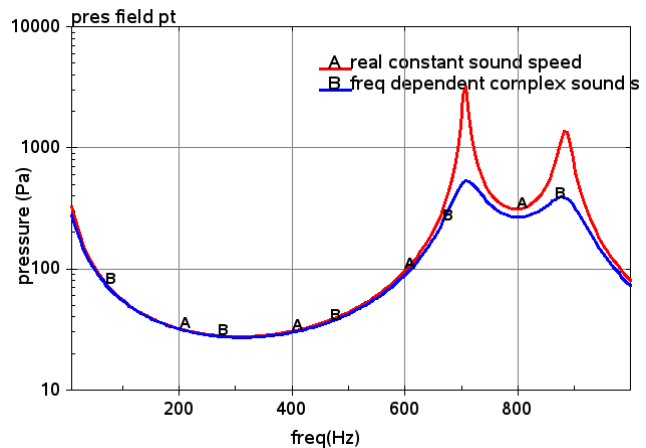


Fig.12: Acoustic pressure (Pa) for two cases of sound speed

From Figure 12, one can see that when changing sound speed from real to frequency dependent complex variables, the peak values of the acoustic pressure is reduced, which reflects the contribution from damping.

When the frequency dependent complex sound speed is used, the distribution of acoustic pressure (real part) is shown in Figure 13, for the frequency 1000 Hz.

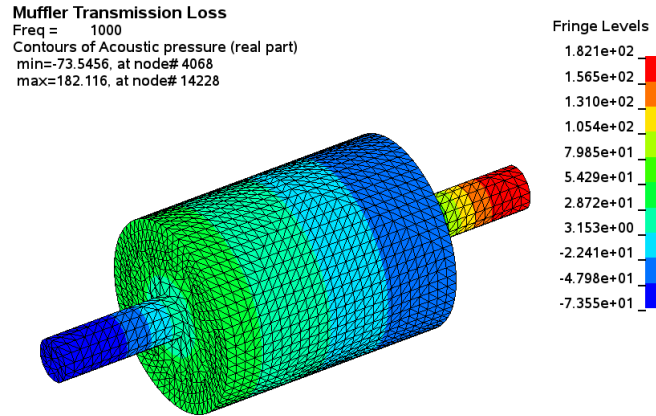


Fig. 13: Acoustic pressure (Pa, real part) at frequency 1000 Hz.

4 Acoustic eigenvalue analysis

The governing equation for the acoustic problems is the Helmholtz equation [2],

$$\nabla^2 p + k^2 p = 0 \quad (2)$$

Where p is the acoustic pressure; $k = \omega/c$ is called the wave number; $\omega = 2\pi f$ is the circular frequency of the acoustic wave; and c is the wave speed.

For vibro-acoustic problems, the boundary condition is given as

$$\partial p / \partial n = -i\rho\omega v_n \quad (3)$$

here n is the normal vector pointing outside from the acoustic volume; $i = \sqrt{-1}$ is the imaginary unit; ρ is the acoustic fluid density and v_n is the normal velocity.

Using the weighted residue technique and taking the shape function N_i as the weighting function, the governing equation can be written as

$$\int_V \nabla^2 p N_i dV + \int_V k^2 p N_i dV = 0 \quad (4)$$

Using the Green's theorem, equation (4) can be written as

$$\int_V \nabla p \nabla N_i dV - k^2 \int_V p N_i dV = \int_{\Gamma} \frac{\partial p}{\partial n} N_i d\Gamma \quad (5)$$

This can be expressed as

$$([K_a] - \omega^2 [M_a])\{p\} = \{F_a\} \quad (6)$$

Where, $[K_a]$, $[M_a]$ are acoustic stiffness and mass matrices. Thus an acoustic eigenvalue problem can be set up as

$$[K_a]\{\phi_i\} = \omega_i^2 [M_a]\{\phi_i\} \quad i = 1, \dots, N_a \quad (7)$$

Where ω_i is the eigenfrequency of i -th mode and $\{\phi_i\}$ is the i -th eigenvector.

To run acoustic eigenvalue analysis, a new option `_EIGENVALUE` has been added to the keyword `*FREQUENCY_DOMAIN_ACOUSTIC_FEM`. Eigen frequencies and corresponding eigenvectors are provided.

For the auto compartment model, shown in Figure 2, the natural frequencies for the first 20 eigen modes are computed, and compared with the results given by NASTRAN (Table 1). The first mode is rigid body mode and the eigen frequencies given by LS-DYNA and NASTRAN are both nearly 0. For the rest of the modes, the eigen frequencies given by LS-DYNA and NASTRAN match very well.

Mode	LS-DYNA	NASTRAN
1	3.93144E-06	6.698696E-06
2	8.10808E+01	8.108078E+01
3	1.25457E+02	1.254568E+02
4	1.47780E+02	1.477799E+02
5	1.52190E+02	1.521901E+02
6	1.72872E+02	1.728723E+02
7	1.98448E+02	1.984481E+02
8	2.08590E+02	2.085895E+02
9	2.14581E+02	2.145808E+02
10	2.23230E+02	2.232297E+02

Mode	LS-DYNA	NASTRAN
11	2.48374E+02	2.483744E+02
12	2.52407E+02	2.524068E+02
13	2.53542E+02	2.535415E+02
14	2.58978E+02	2.589778E+02
15	2.65955E+02	2.659551E+02
16	2.74129E+02	2.741291E+02
17	2.91410E+02	2.914401E+02
18	2.99179E+02	2.991786E+02
19	3.06640E+02	2.066399E+02
20	3.09067E+02	3.090668E+02

Table 1: Natural frequencies of the auto compartment

The eigenvectors are saved in the binary plot database `D3EIGV_AC`, which is accessible to LS-PrePost. The eigenvectors computed by LS-DYNA are also compared with those computed by NASTRAN, see Figures 14 and 15.

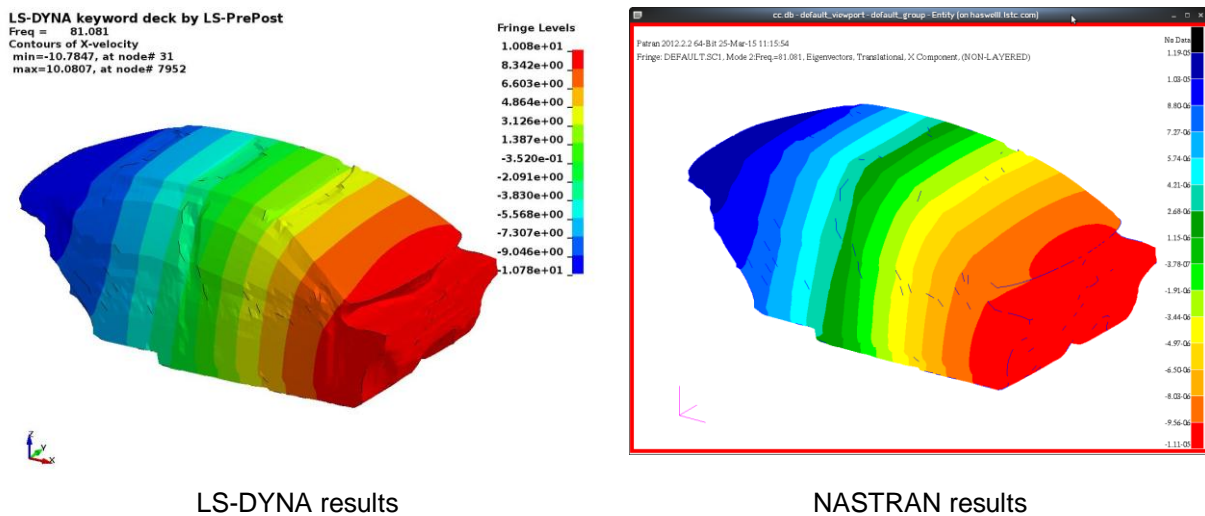


Fig. 14: Acoustic eigenvector for the 2nd mode

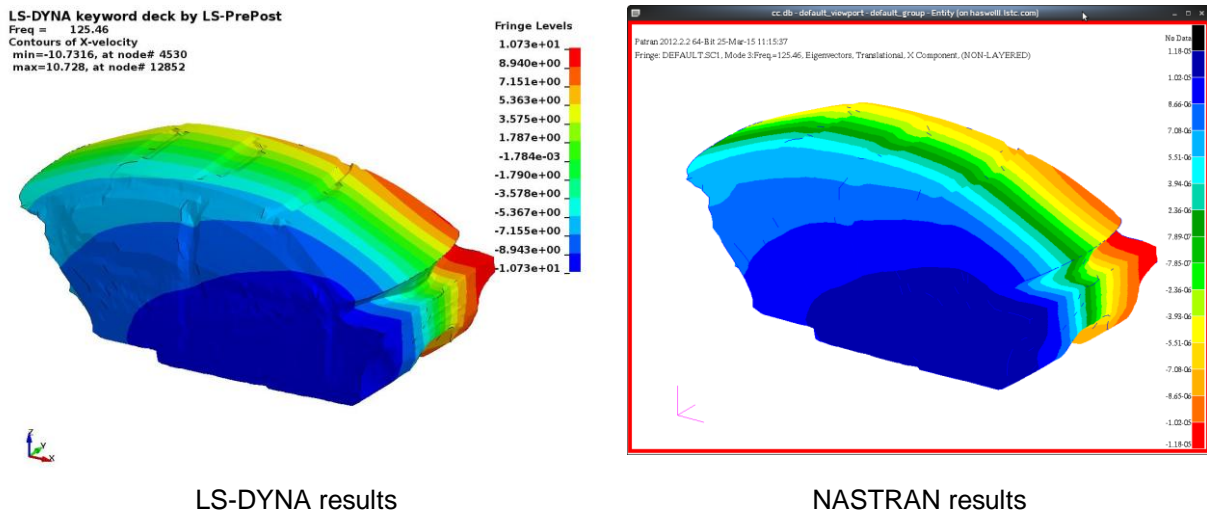


Fig. 15: Acoustic eigenvector for the 3rd mode

One can see, even though the values of the eigenvectors are different between LS-DYNA and NASTRAN (due to different scaling), the shape of the fringe plots are still very similar.

In acoustic eigenvalue analysis, various constraints on the boundary can be considered, like the prescribed pressure boundary condition. Table 2 gives the eigen frequencies for the first 20 modes, for the compartment model shown in Figure 3 (with windows open).

Mode	LS-DYNA	NASTRAN	Mode	LS-DYNA	NASTRAN
1	6.795510E+01	6.795506E+01	11	2.577480E+02	2.577482E+02
2	1.033460E+02	1.033460E+02	12	2.698970E+02	2.698974E+02
3	1.478530E+02	1.478530E+02	13	2.730180E+02	2.730175E+02
4	1.582110E+02	1.582106E+02	14	2.786900E+02	2.786899E+02
5	1.747810E+02	1.747807E+02	15	2.889850E+02	2.889853E+02
6	1.874600E+02	1.874595E+02	16	2.891140E+02	2.891136E+02
7	2.109060E+02	2.109057E+02	17	3.023890E+02	3.023888E+02
8	2.149930E+02	2.149934E+02	18	3.112120E+02	3.112121E+02
9	2.288610E+02	2.288609E+02	19	3.237700E+02	3.237698E+02
10	2.506670E+02	2.506669E+02	20	3.262210E+02	3.262206E+02

Table 2: Natural frequencies of the auto compartment with windows open

The acoustic eigenvalue analysis not only provides valuable characteristic information on the acoustic system itself, but also provides basis for modal solution method (comparing to direct solution method) for large scale problems.

5 Improved fatigue analysis in a sine sweep test environment

An option to run fatigue analysis has been implemented for SSD (Steady State Dynamics) in LS-DYNA. With this option LS-DYNA computes the cumulative damage ratio of structures, subjected to harmonic or steady state vibration. This vibration condition is very common for shaker table tests, e.g. in a sine sweep test.

Two new stress indexes (Maximum principal stress, and Maximum shear stress) have been incorporated in the fatigue analysis, in addition to the existing one (Von-Mises stress). The selection of the stress index in SSD fatigue analysis is enabled through the parameter STRTYP in the keyword *FREQUENCY_DOMAIN_SSD_FATIGUE.

For a simple bracket model, which is subjected to acceleration excitation through the two ends, the cumulative damage ratio is computed by using the three stress indexes. The loading condition and the

material's S-N fatigue curve are given in Tables 3 and 4. The computed cumulative damage ratios are plotted in the binary plot database D3FTG, as shown in Figure 16.

Frequency (Hz)	Acceleration (g)	Duration (min)
16	0.5	12
20	0.5	12
25	0.5	12
31.5	0.5	12
...
2000	0.5	12

Table 3: Loading condition

σ (MPa)	N
100	8×10^4
10	8×10^5
1.	8×10^6
0.1	8×10^7
0.01	8×10^8

Table 4: S-N fatigue curve

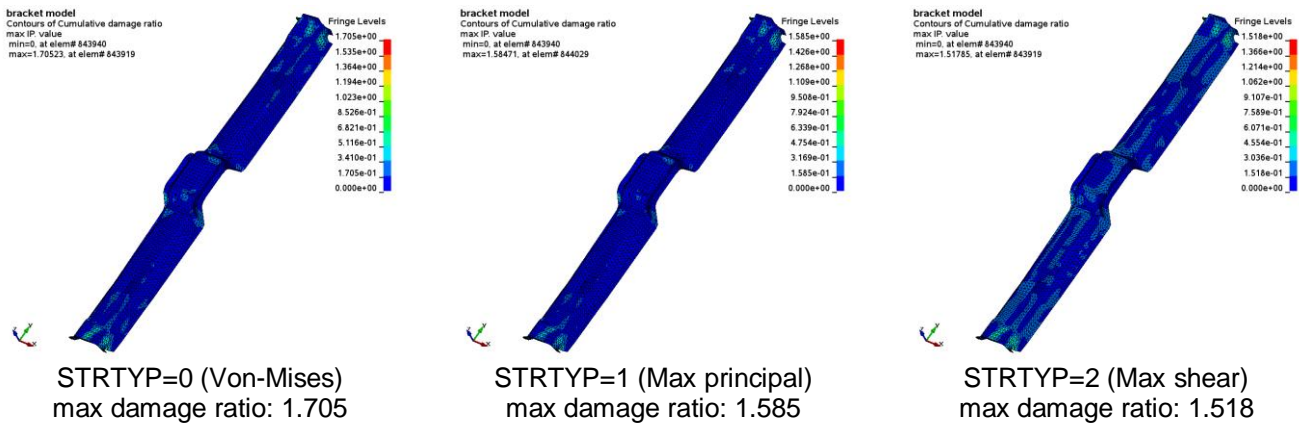


Fig. 16: Cumulative damage ratio computed with different stress index

One can see, though the max values of the cumulative damage ratio are different, they are pretty close, and the distribution of the cumulative damage ratio is also very similar.

6 Incorporation of multi-slope S-N fatigue curve in fatigue analysis

The random vibration fatigue and SSD fatigue features are both based on the materials' S-N fatigue curves. The material's S-N curve is a property of material, and can be defined by the keyword *MAT_ADD_FATIGUE for each material model.

With recent updates on the keyword *MAT_ADD_FATIGUE, user can define an S-N curve with multi slopes, as shown in Figure 17.

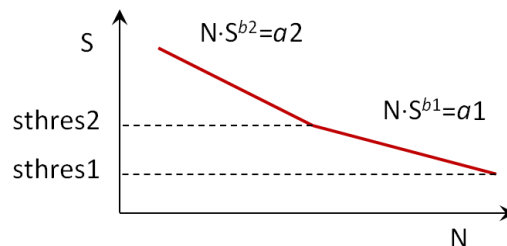


Fig. 17: Multi-slope S-N fatigue curve

The input cards for the keyword are updated as [1]:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	LCID	LTYPE	A	B	STHRES	SNLIMT	SNTYPE
Type	I	I	I	F	F	F	I	I
Default	none		0	0.0	0.0	none	0	0

Card 2	1	2	3	4	5	6	7	8
Variable				Ai	Bi	STHRESi		
Type				F	F	F		
Default				0.0	0.0	none		

To define S-N curve with multiple slopes, the S-N curve can be split into multiple segments and each segment is defined by a set of parameters A_i , B_i and $STHRES_i$. Up to 8 sets of the parameters (A_i , B_i and $STHRES_i$) can be defined. The lower limit of the i -th segment is represented by the threshold stress $STHRES_i$.

With the incorporation of multi-slope S-N fatigue curve, user can adopt more complicated material models in fatigue analysis.

7 Strain results in random vibration analysis

In random vibration analysis, strain results are added to D3PSD, and D3RMS. To get strain results in D3PSD and D3RMS for solids, shells and thick shells, one need to set STRFLG=1 in keyword *DATABASE_EXTENT_BINARY; To get beam stresses in D3PSD nad D3RMS, one need to set BEAMIP=1 in keyword *DATABASE_EXTENT_BINARY.

The example shown in Figure 18 is the xx strain results at frequency 500 Hz, given at lower integration point, for a bumper model.

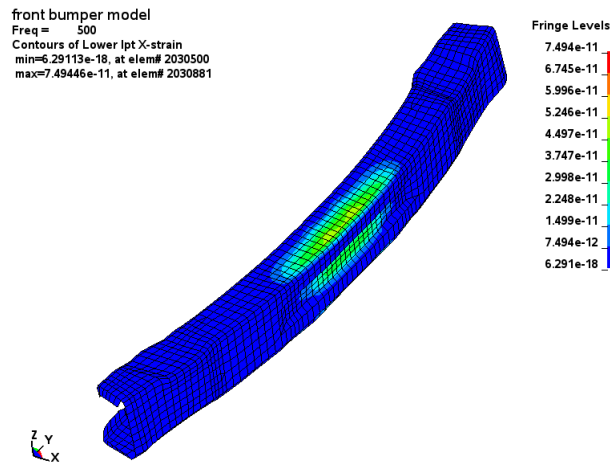


Fig. 18: Xx strain at lower integration point in D3PSD

8 Summary

A series of updates for the frequency domain features are described in this paper. These updates not only improve the current capabilities of the frequency domain solvers, but also extend greatly the application area of these solvers. Some examples are provided to demonstrate the application of these updated frequency domain solvers.

9 Literature

- [1] LS-DYNA Keyword User's Manual, Livermore Software Technology Corporation, 2015.
- [2] Wu, T.W.: "Boundary Element Acoustics: Fundamentals and Computer Codes", Advances in Boundary Elements, Southampton, Boston, WITpress, 2000.