Recent developments in NVH and fatigue solvers in Ansys LS-DYNA®

Yun Huang[1], Tom Littlewood[1], Zhe Cui[1], Ushnish Basu[1], Stefan Hartmann[2], David Benson[1]

[1]Ansys, Inc., US
[2]DYNAmore GmbH, Germany

Abstract

As one of the mainstream software packages widely used in automotive industry, LS-DYNA provides not only strong nonlinear capabilities for crashworthiness simulation, but also a suite of solvers for NVH and fatigue (durability) analysis. For NVH analysis, a series of vibration and acoustic solvers have been implemented to meet the need from CAE analysis of automotive of different levels and phases. They include FRF (Frequency Response Function), SSD (Steady State Dynamics), random vibration, response spectrum analysis, and acoustic analysis based on BEM (Boundary Element Method), FEM (Finite Element Method) and SEM (Spectral Element Method). The fatigue analysis features include fatigue damage solvers in both time domain and frequency domain (based on random vibration and steady state vibration).

Since the last European Users' Conference in Koblenz, Germany, 2019, many new solvers / options were implemented to the NVH and fatigue (durability) analysis modules, including

- Acoustic spectral element method
- Mode-based FEM acoustics
- New boundary / load conditions for transient acoustic analysis
- SSD analysis with multiple load cases
- SSD with coupled fluid-structure system
- Fatigue analysis based on modal dynamics
- Random vibration fatigue analysis based on IGA models
- D3MAX for stress and strain envelope

This paper gives a brief review of these new solvers / options for NVH and fatigue analysis with LS-DYNA and discusses the areas of application. Some examples are included for illustration purpose. The plan for future development in this area is also discussed.

1 Introduction

Since ls971 R6 version, a series of NVH and fatigue analysis features have been implemented to LS-DYNA. They include vibration solvers (FRF, SSD, Random vibration, and Response spectrum analysis), acoustic solvers, and fatigue solvers in both time domain and frequency domain. They were developed to answer the need from users from different industries. They have wide range of applications including NVH analysis of automobiles, noise prediction of engines, and fatigue life prediction of metal parts in defense industry.

During the past few years, some new options and capabilities have been implemented to answer the request from users. Some enhancements to the existing features were implemented too, to improve the performance of them, or to make the solution faster or easier. These new options / capabilities / enhancements are introduced in more details in the following sections.

2 Acoustic spectrum element method

The explicit transient spectral element method (SEM) is a sub-parametric finite element method which is especially effective for the simulation of ultrasonic acoustic waves. The SEM recently implemented in LS-DYNA for acoustics uses higher-order Lagrangian interpolants over unevenly spaced nodes.

Element integration is performed with integration orders ranging from 2 to 15 and Gauss-Legendre-Lobatto integration at points which overlap the nodes.

With its excellent convergence behavior, and high accuracy in the results, the SEM is a good choice for modelling high frequency acoustic wave propagation, scattering and reflection problems, such as the ultrasonic sensors used in autonomous parking / driving and USCT (UltraSonic computer Tomography).

So far, SEM is performed on the isoparametric acoustic finite element mesh (solid element 8), activated by the new keyword **`*CONTROL_ACOUSTIC_SPECTRAL`**. The time interval for nodal output of acoustic solution is defined by DT in **`*DATABASE_ACEOUT`**. Load and boundary conditions for SEM are defined by keywords **`*LOAD_ACOUSTIC_SOURCE`**, **`*BOUNDARY_ACOUSTIC_*`**. This solver has been enabled for both SMP and MPP.

Figure 1 shows a model for ultrasonic wave generation and propagation in a 1/8 sphere space behind rear bumper of a vehicle. The purpose of this model is to study the reflection of ultrasonic wave on a pole (or other obstacles) behind the vehicle. Pressure pulses are applied at the boundary near the bumper and the outer surface of the 1/8 sphere has a non-reflecting boundary condition. There are 38,104,448 SE elements (with 4,775,245,341 equations) for the reduced model.

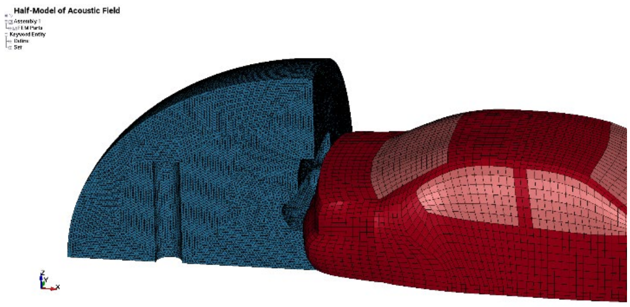


Fig.1: Acoustic volume behind the bumper modelled by SEM

An animation of acoustic pressure propagation in the domain can be obtained with SEM computation. For example, the fringe plot of acoustic pressure at time 1 ms can be found in Figure 2.

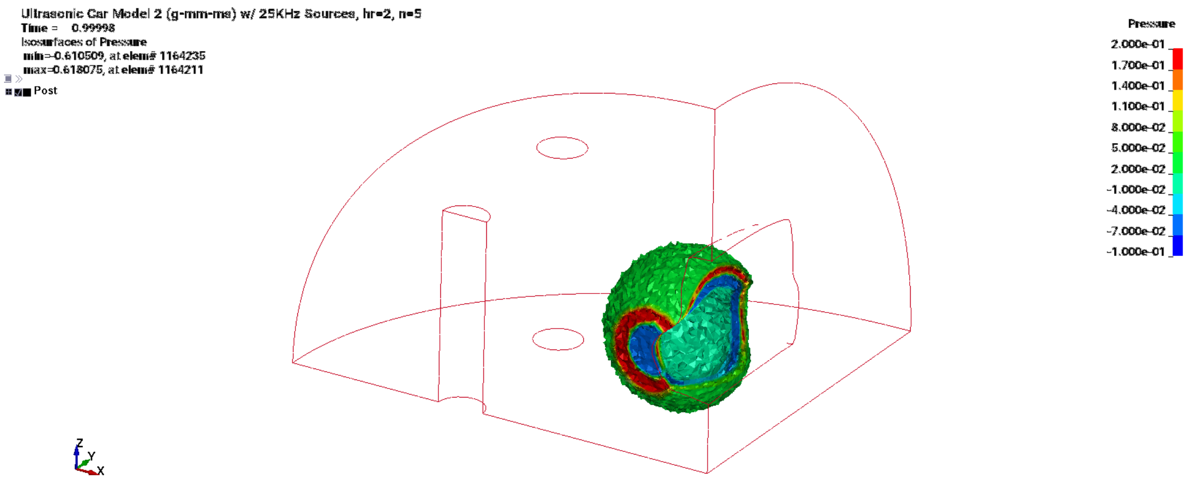


Fig.2: Propagation of acoustic pressure inside the domain (time = 1 ms)

A more detailed introduction of this method is given by [2].

3 Mode-based FEM acoustics

`*FREQUENCY_DOMAIN_ACOUSTIC_FEM` can run finite element method for acoustics, in a direct way, where the unknown variables are nodal acoustic pressure. In addition, acoustic eigenvalue analysis can be performed, with the option **_EIGENVALUE** for this keyword. As the result, acoustic eigenvalues and eigenvectors can be computed and saved in ASCII database EIGOUT_AC and binary plot database D3EIGV_AC.

A new option **_MODAL** has been implemented for ***FREQUENCY_DOMAIN_ACOUSTIC_FEM**, to run acoustic computation by modal superposition, using pre-computed acoustic eigenvectors. A simple example can be found in Figure 3, where the cabin model is excited by input normal velocity from the base.

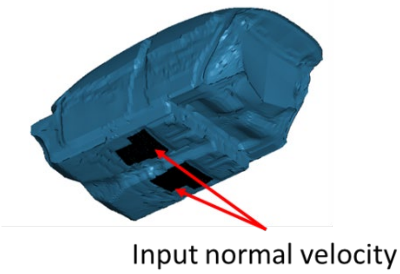


Fig.3: A cabin model with base velocity excitation

The solution of this problem is saved in binary plot database D3ACS, as shown in Figure 4. Please note that the results are dependent on excitation frequency and multiple states (one state for one frequency) can be found in D3ACS.

For cross validation, this problem was solved by direct method ***FREQUENCY_DOMAIN_ACOUSTIC_FEM** and mode-based method ***FREQUENCY_DOMAIN_ACOUSTIC_FEM_MODAL**. For the mode-based method, two cases with different number of acoustic eigenmodes are tested. For the 1st case, 400 eigenmodes are used and for the 2nd case, 1000 eigenmodes are used. Generally speaking, with more eigenmodes used in the modal superposition, the results are more accurate. For this problem, the results are plotted together in Figure 5 and one can see that the results of the three tests match very well, even for the one computed with only 400 eigenmodes.

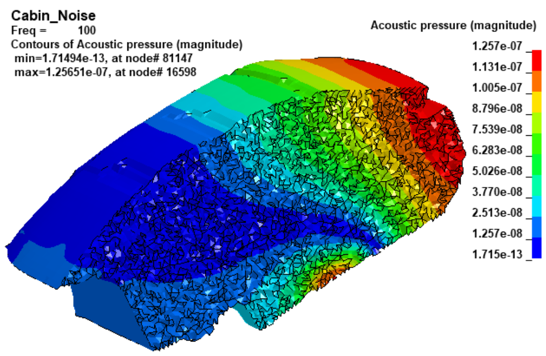


Fig.4: Acoustic pressure (magnitude) at frequency 100 Hz.

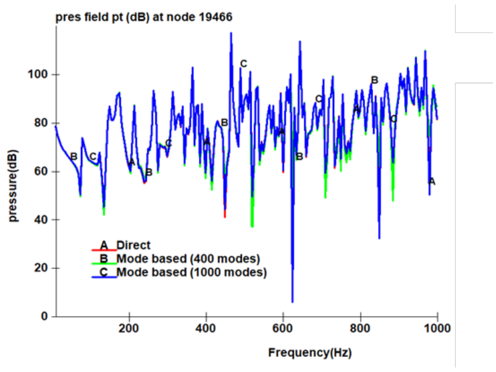


Fig.5: Sound pressure level at one node provided by direct method and mode-based method (with two different number of eigenmodes)

Table 1 compared the CPU cost for the two methods

Direct method	**Mode-based method**	
	400 modes	1000 modes
2283	308 = 270¹ + 38²	597 = 527¹ + 70²

1: CPU cost for acoustic eigenvalue analysis
2: CPU cost for modal superposition

Table 1: CPU cost (second) for the two methods

One can see from Table 1 that the mode-based method is much faster than the direct method. This can be explained by the fact that the number of unknown variables in the mode-based method (number of eigenmodes) is much less than the number of unknown variables in the direct method (number of nodes).

Besides, as shown in the notes below Table 1, the CPU cost for mode-based method is composed of two parts: CPU cost for acoustic eigenvalue analysis and CPU cost for modal superposition. For a model subjected to multiple loading cases, since one just needs to solve the acoustic eigenvalue problem only once and reuse the eigenvectors for each of the loading cases, the saving in the total CPU cost can be more significant.

4 New boundary / load condition keywords for acoustic analysis

In the recent version of LS-DYNA (R13 and dev), some new boundary / load conditions for transient and harmonic acoustic analysis have been implemented, so that users can solve more realistic problems with comprehensive boundary and load conditions, like the one in Figure 6.

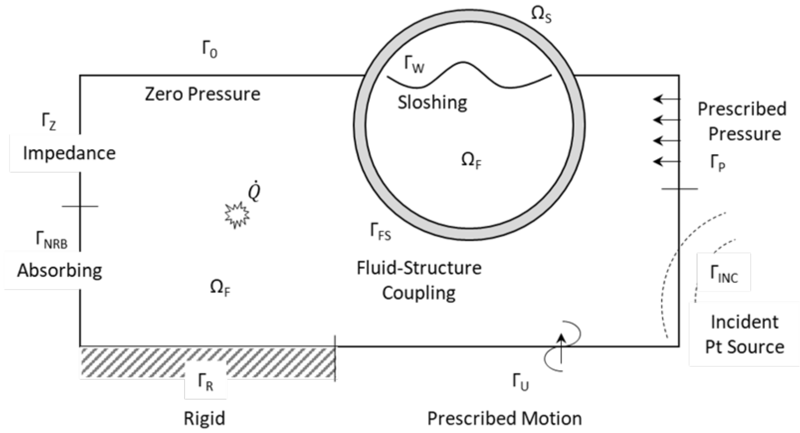


Fig.6: Various boundary conditions for acoustic problems

The new boundary / load conditions are defined by a series of new keywords listed below

Feature	Keyword
Structural Coupling - Γ_{FS}	***BOUNDARY_ACOUSTIC_COUPLING_MISMATCH** ***BOUNDARY_ACOUSTIC_COUPLING_SPECTRAL**
Weak Structural Coupling - Γ_{FS}	***INTERFACE_ACOUSTIC** ***BOUNDARY_ACOUSTIC_INTERFACE**
Prescribed Boundary Motion - Γ_U	***BOUNDARY_ACOUSTIC_PRESCRIBED_MOTION**
Prescribed Boundary Pressure - Γ_P	***BOUNDARY_ACOUSTIC_PRESSURE** ***BOUNDARY_ACOUSTIC_PRESSURE_SPECTRAL**
Rigid Boundary - Γ_R	This is a natural condition
Impedance Boundary - Γ_Z	***BOUNDARY_ACOUSTIC_IMPEDANCE** ***BOUNDARY_ACOUSTIC_COMPLEX** ***BOUNDARY_ACOUSTIC_MECHANICAL**
Absorbing Boundary - Γ_{NRB}	***BOUNDARY_ACOUSTIC_NON_REFLECTING**
Zero Pressure Boundary – Γ_0	***BOUNDARY_ACOUSTIC_FREE_SURFACE**
Linear Wave Boundary – Γ_W	***BOUNDARY_ACOUSTIC_FREE_SURFACE**
Internal Point Source – Q	***LOAD_ACOUSTIC_SOURCE**

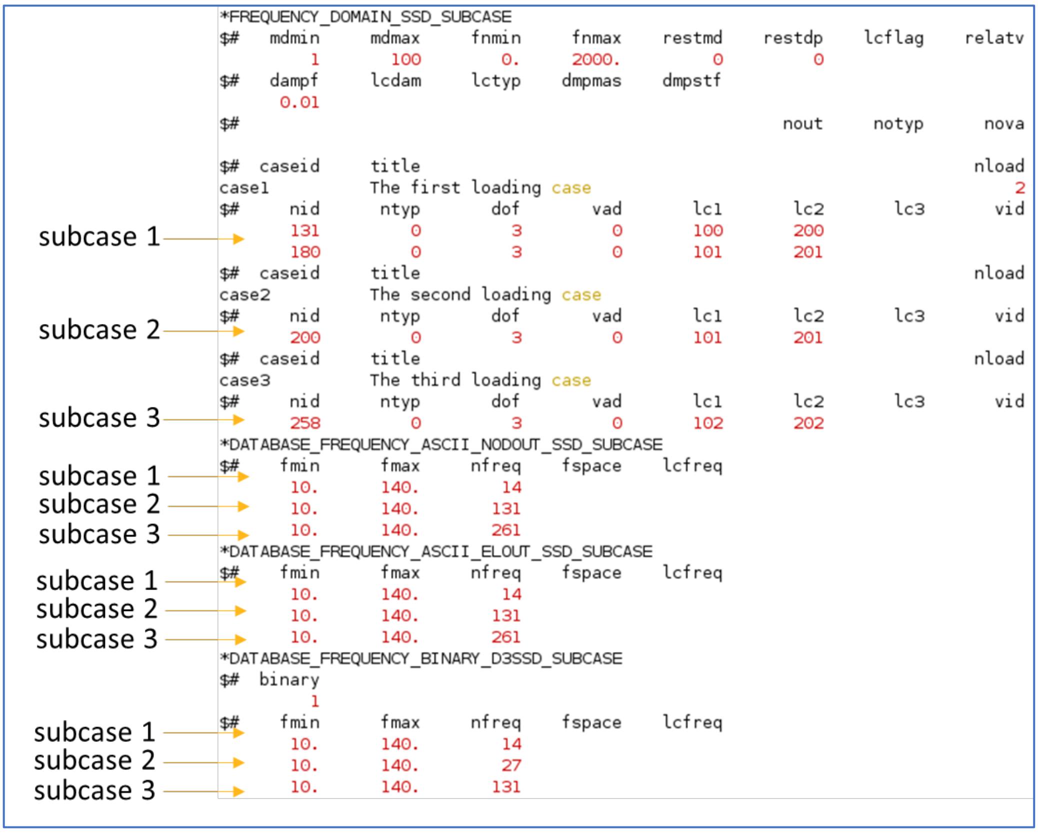
Incident Wave Point Source – P_{inc}	***LOAD_ACOUSTIC_SOURCE**

Table 2: Keywords for new boundary / load condition in acoustic analysis

5 SSD analysis with multiple load cases

LS-DYNA can run steady state dynamic analysis (or harmonic vibration analysis) using the keyword ***FREQUENCY_DOMAIN_SSD**. For many structures, it is possible that they need to be studied under multiple load cases. In the past, to include multiple load cases in a single input and get the simulation done in one run, one uses the ***CASE** commands. When ***CASE** commands are used to define multiple cases, some portions of the input will be shared by some, or all of the cases and other portions will be unique to each case. A series of input decks are generated automatically and sequentially (case*.inp), and LS-DYNA runs these input decks sequentially (each as a completely new run).

Based on suggestions from some users, a **_SUBCASE** option was implemented to the keyword ***FREQUENCY_DOMAIN_SSD**, to include multiple load cases under one keyword input. The same **_SUBCASE** option is also available to the keywords ***DATABASE_FREQUENCY_ASCII_NODOUT_SSD**; ***DATABASE_FREQUENCY_ASCII_ELOUT_SSD**, ***DATABASE_FREQUENCY_ASCII_NODFOR_SSD** and ***DATABASE_FREQUENCY_BINARY_D3SSD** so that user can define different output frequencies for each load case. A sample input deck is shown in Figure 7 below.


```
*FREQUENCY_DOMAIN_SSD_SUBCASE
$#   mdmin     mdmax     fnmin     fnmax    restmd    restdp    lcflag    relatv
         1       100        0.     2000.         0         0
$#   dampf     lcdam     lctyp    dmpmas    dmpstf
      0.01
$#                                                    nout     notyp      nova

$#  caseid     title                                                     nload
case1          The first loading case                                        2
$#     nid      ntyp       dof       vad       lc1       lc2       lc3       vid
       131         0         3         0       100       200
       180         0         3         0       101       201
$#  caseid     title                                                     nload
case2          The second loading case
$#     nid      ntyp       dof       vad       lc1       lc2       lc3       vid
       200         0         3         0       101       201
$#  caseid     title                                                     nload
case3          The third loading case
$#     nid      ntyp       dof       vad       lc1       lc2       lc3       vid
       258         0         3         0       102       202
*DATABASE_FREQUENCY_ASCII_NODOUT_SSD_SUBCASE
$#    fmin      fmax     nfreq    fspace    lcfreq
       10.      140.        14
       10.      140.       131
       10.      140.       261
*DATABASE_FREQUENCY_ASCII_ELOUT_SSD_SUBCASE
$#    fmin      fmax     nfreq    fspace    lcfreq
       10.      140.        14
       10.      140.       131
       10.      140.       261
*DATABASE_FREQUENCY_BINARY_D3SSD_SUBCASE
$#  binary
         1
$#    fmin      fmax     nfreq    fspace    lcfreq
       10.      140.        14
       10.      140.        27
       10.      140.       131
```


*Fig.7: Including multiple load cases with **_SUBCASE***

The **_SUBCASE** option brings significant savings in CPU cost, due to

- no generation of input deck (caseN.inp) for each load case;
- no input deck reading and variable initialization for each load case, except for the 1st one;
- no MPP partitioning for each load case (for MPP runs), except for the 1st one.

6 SSD with coupled fluid-structure system

For many applications, it is important to consider the full interaction between structures and fluids. For example, to model the tank sloshing problem, a full interaction between tank structure and gas inside is needed. Another example is a submarine surrounded by sea water. Since the water is heavy (comparing to air), the dynamic response of the submarine is affected by the added fluid mass from the water. For this type of analysis, a new keyword **`*CONTROL_IMPLICIT_SSD_DIRECT`** has been implemented to model the fluid-structure coupling in frequency domain directly.

This method established direct, complex solutions to steady state vibration of coupled acoustic fluid and structure system. For acoustic domain, solid elements 8 and 14 may be used. The coupling of the acoustic fluid and the structural elements is achieved with the keyword **`*BOUNDARY_ACOUSTIC_COUPLING_MISMATCH`** or by merging acoustic and structural nodes with compatible element faces. This method is useful for the cases when interaction between the fluid and the structure need to be considered.

For a simple problem in Figure 8, the solution given by LS-DYNA (Figure 9, left) is compared with the analytical solution given by a book (Figure 9, right). One can see that LS-DYNA results and the results from analytical solutions agree very well.

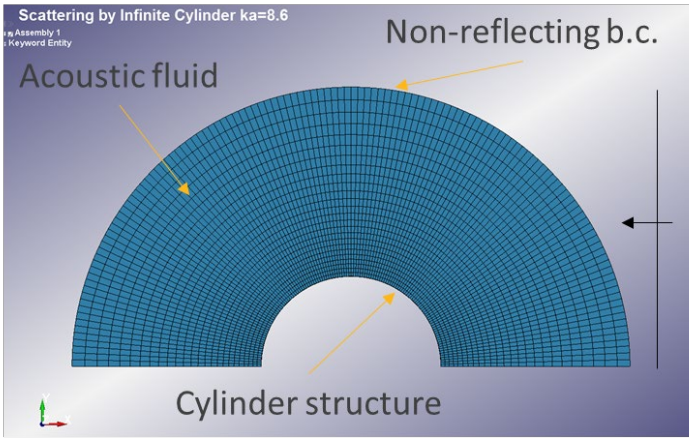


Fig.8: Interaction of cylinder structure and acoustic fluid surrounding the cylinder

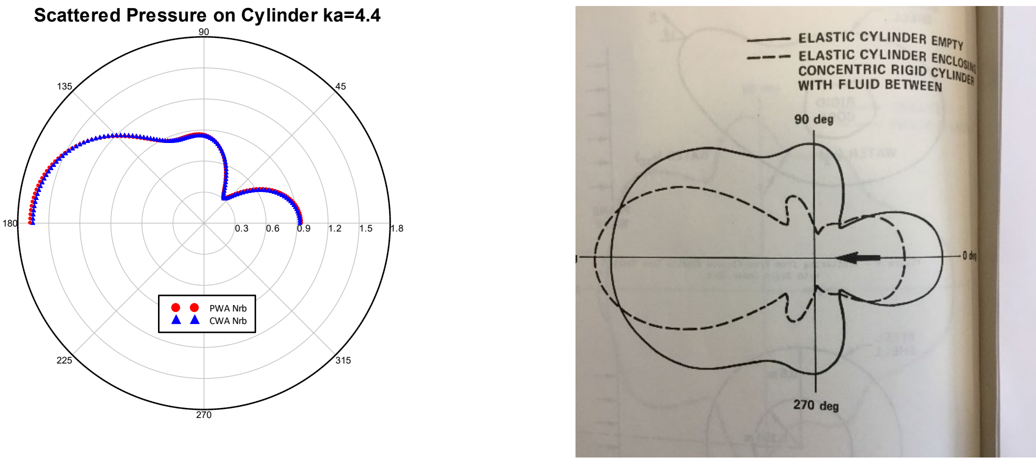


Fig.9: Solution by LS-DYNA (left) and Reference by Everstine (right)

7 Fatigue analysis based on modal dynamics

LS-DYNA can run transient fatigue analysis based on stress / strain from database ELOUT or D3PLOT (see keywords **`*FATIGUE_ELOUT`** and **`*FATIGUE_D3PLOT`**). Recently, a new option

_MODAL_DYNAMIC was implemented to provide a faster method to run fatigue analysis based on the results from modal dynamics.

This method is based on using the binary database D3EIGV from implicit eigenvalue analysis (keyword ***CONTROL_IMPLICIT_EIGENVALUE**), and the ASCII database MODDYNOUT from modal dynamics (***CONTROL_IMPLICIT_MODAL_DYNAMIC**). With D3EIGV, LS-DYNA extracts modal stress for each eigenmode; with Moddynout, LS-DYNA gets an array of modal coefficients at each time point. After that, LS-DYNA can run modal superposition to reconstruct the stress state for the whole time history and use it for fatigue analysis.

This method is faster than ***FATIGUE_ELOUT** and ***FATIGUE_D3PLOT** because no any stress output is needed in the transient dynamic analysis phase. As comparison, with ***FATIGUE_ELOUT** and ***FATIGUE_D3PLOT**, the stress results must be dumped to ELOUT or D3PLOT at a very high frequency, to get sufficient stress data for subsequent fatigue analysis. In addition, since no ELOUT and D3PLOT is required (users may still write out a limited number of D3PLOT for other purpose), this method shows a big saving in hard drive space usage.

As a demo, time domain fatigue analysis is performed for a metal bracket model shown below

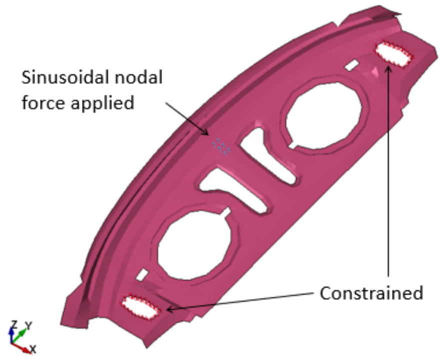


Fig.10: A metal bracket subjected to sinusoidal nodal force for 10 seconds

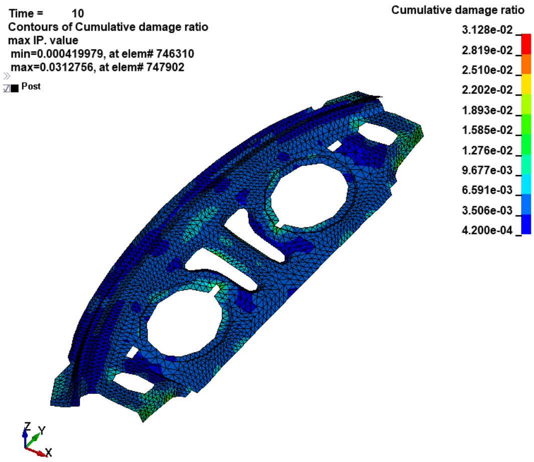


Fig.11: Cumulative damage ratio plot at the termination time (10 seconds)

And Table 3 compared the CPU cost for running this problem with different methods.

Method	**CPU time (sec, 1 core)**
***FATIGUE_MODAL_DYNAMIC**	255
***FATIGUE_ELOUT**	659 (with 7G binout files)
***FATIGUE_D3PLOT**	462 (with 15G D3PLOT files)

Table 3: CPU time for fatigue analysis of the bracket model by different methods.

8 Random vibration fatigue analysis based on IGA models

Isogeometric Analysis (IGA) is a new computational method which performs finite element analysis using mathematical geometry descriptions used in CAD tools, like NURBS (non-uniform rational B-spline). Therefore, it replaces the piecewise continuous Lagrangian polynomial interpolation functions in FEM with higher order spline basis functions. As a result, this method can be more accurate than traditional FEM. In the past few years, much progress in the implementation of IGA in LS-DYNA has been made. This also brings new opportunities for other applications like fatigue analysis.

A new scheme for random vibration fatigue analysis has been developed in LS-DYNA, based on using the IGA technique for eigenvalue analysis and modal stress computation. Random vibration and fatigue damage ratio computation are performed on the interpolated finite element mesh, using the mapped eigenvectors and modal stress. The fatigue results are accurate and can match very well with the results obtained by traditional finite element method.

The model shown in Figure 12 is a metal bracket, subject to base acceleration PSD excitation. This type of random excitation is very common, for parts or components used in a road or railway vehicle.

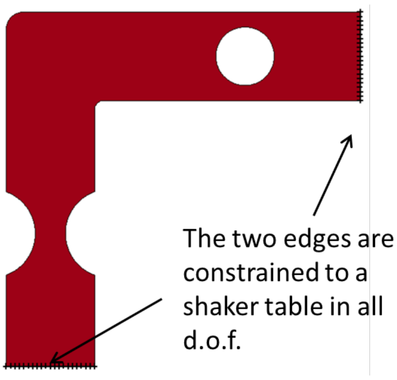


Fig.12: A metal bracket subjected to base acceleration excitation

The same model was computed using FEM and IGA methods. The RMS (Root Mean Square) value of effective stress by FEM and by IGA are plotted in Figure 13 and Figure 14.

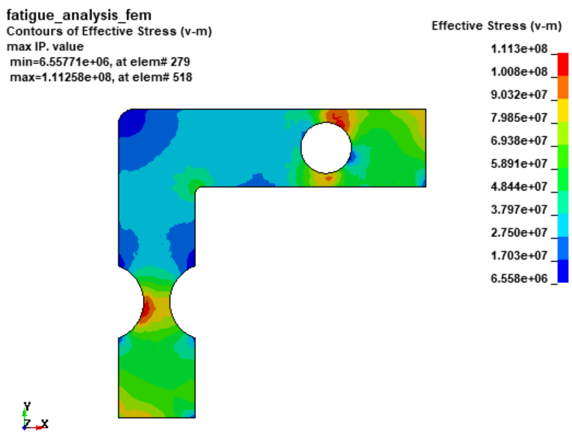


Fig.13: Effective stress RMS from FEM

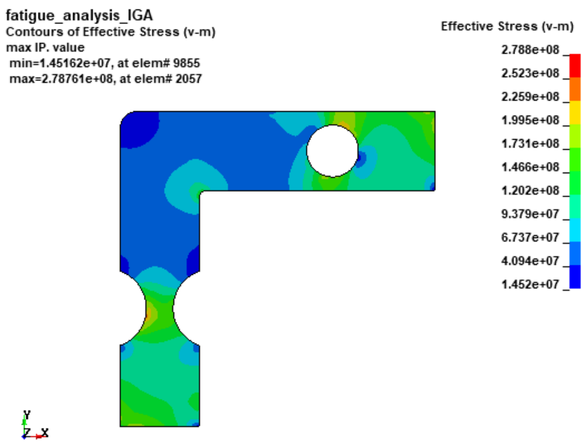
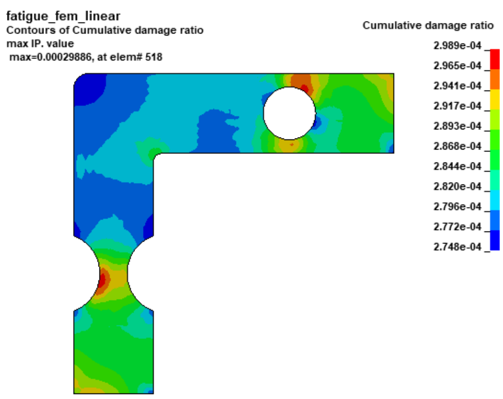


Fig.14: Effective stress RMS from IGA

And the Figures 15 and 16 compare the cumulative damage ratio given by FEM and IGA.



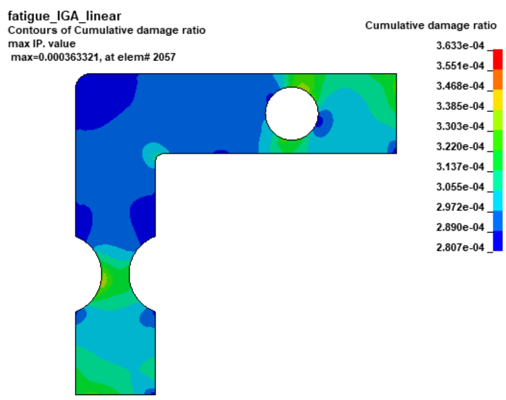


Fig.15: Cumulative damage ratio from FEM *Fig.16: Cumulative damage ratio from IGA*

For this example, the stress results and cumulative damage ratio results given by FEM and IGA are close, as shown in the above figures.

For the next step, the interpolation finite element mesh will be skipped, and all the random vibration and fatigue computation will be performed on the IGA patches directly. For example, we will compute the cumulative damage ratio on the integration points directly. Hopefully this can bring further saving in the CPU cost and make the results more accurate since the possible errors from the interpolation can be avoided.

9 D3MAX for stress and strain envelope

LS-DYNA provides a series of binary plot databases like D3PLOT and D3PART to save the response of structures in a transient event, like a drop test. For many users, they are more concerned with the maximum values of stress and strain during the transient event. To capture these maximum stress and strain with D3PLOT or D3PART, a very small time step for writing these files has to be used. This can make the IO overwhelming and result in a huge hard drive space consumption.

To solve this problem, a new binary plot database D3MAX has been introduced to LS-DYNA, to monitor the maximum values of stress and strain.

This feature is activated by a new keyword **`*DATABASE_BINARY_D3MAX`**. This database saves the maximum of stress and strain experienced in a transient event. A very small time step “dt” can be defined in this keyword, so that LS-DYNA can compare the saved stress / strain in D3MAX with the stress / strain in transient analysis very frequently (much more frequently than the stress / strain are dumped to D3PLOT) and update the maximum stress / strain if needed, as illustrated in Figure 17.

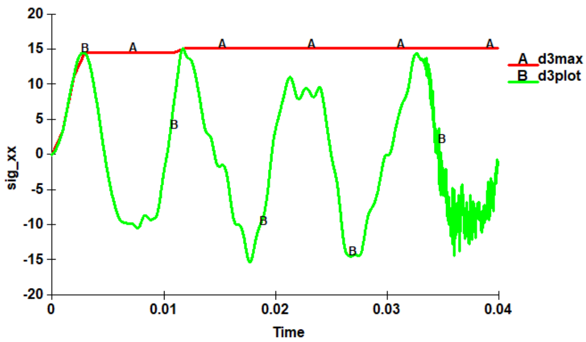


Fig.17: Stress (σ_{xx}) in D3MAX and D3PLOT

Figures 18 and 19 show the stress results from D3MAX and from D3PLOT, at the termination time of transient analysis. Please note, that Figure 18 saves D3MAX results to D3PART, as for some post-processing software, D3MAX is still not acceptable. So, we can choose to dump the D3MAX results to D3PART and use negative time stamps distinguish it from normal D3PART.

As one can see, that at the last time point, the stress results in D3MAX are higher than those in D3PLOT, as the stress in D3PLOT can go up and down along with time, as shown in Figure 17, but the stress in D3MAX will never go down.

To get a smoother result (no ripples), a low pass 2nd order Butterworth filter is also available with this feature.

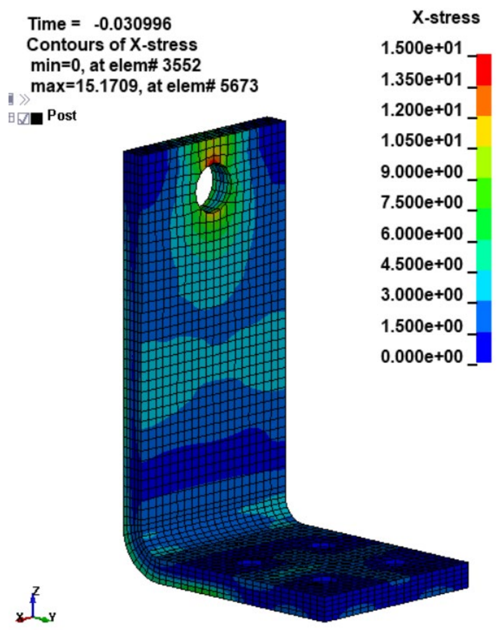


Fig.18: D3MAX saved in D3PART

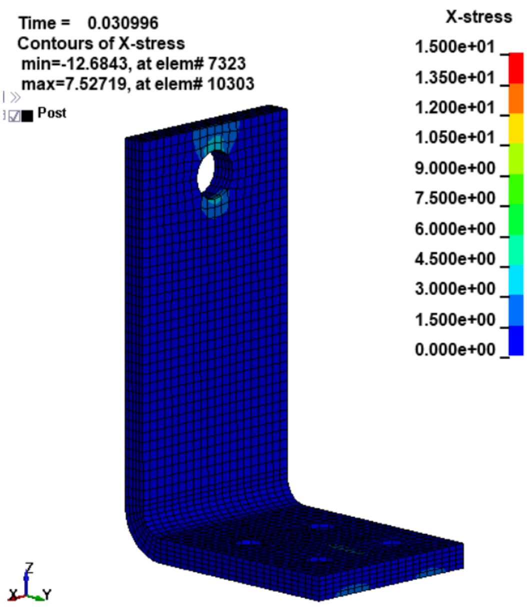


Fig.19: D3PLOT

With D3MAX, one can quickly and accurately identify the maximum stress and strain, and the distribution of these variables in the whole structure. This can be helpful for the safety evaluation of structures.

10 Conclusion and future work

This short paper reviews some recent developments in NVH and fatigue solvers in LS-DYNA. Some examples are included to illustrate the application of these new solvers and options.

For the future work, we will continue to improve and enhance the existing features in NVH and fatigue analysis in LS-DYNA, as well as add new features to meet the increasing need from the users.

Some focused area for future developments includes

- Further integration with Ansys Mechanical/WorkBench. This is needed to facilitate the users of Mechanical and WorkBench to use the solvers in LS-DYNA, and to couple LS-DYNA solvers with other products in Ansys family, like Fluent and Maxwell, to provide more analysis capabilities to the users. For example, the BEM acoustic solver in LS-DYNA could be potentially coupled with Ansys Fluent, to run far field acoustic pressure calculations, based on near field pressure results computed by Ansys Fluent. Hopefully this can provide a feasible solution for air-borne far field noise. Another possible coupling is between LS-DYNA SEA (statistical energy analysis) solver and Fluent, for predicting aero-vibro-acoustic noise inside cabin, similar to the DAVA (Deterministic Aero-Vibro-Acoustics) approach currently used in Ansys.
- Fast Multipole Method for BEM Acoustics. We have started collaborating with colleagues in EBU (Q3D team) to develop a fast multipole method package for Helmholtz integral equations.
- Skeletonized Interpolation for BEM matrix. This is needed to improve the current technology in BLR (block low rank approximation of kernel matrices) for BEM acoustic solvers.
- NVH and fatigue analysis using IGA technology. IGA is the new generation of numerical analysis. Extending the NVH and fatigue solvers from traditional FEM to IGA can help us stand at the front of the development, as well as providing advanced solutions to our users.
- Design optimization based on NVH and fatigue analysis results. As a comprehensive CAE package, LS-DYNA can provide multiphysics, multistage solvers in one code. This brings inherent advantage for multi-disciplinary optimization. For customers in automotive industry, they need to optimize the design of new vehicles not only for safety and crashworthiness, but also for NVH performance and fatigue life. With the NVH and fatigue solvers embedded in the crash simulation

software, this brings the unique opportunity to run MDO (multi-disciplinary optimization) for vehicle design more easily.

- Transfer path analysis. This is needed for source identification and optimization for NVH problems.
- More applications. We are trying to push for more applications of NVH and fatigue solves of LS-DYNA, in various industry areas, like the E-motor noise simulation, ultrasonic sensors for autonomous driving, integrated vibro-fatigue analysis for batteries for E-Vehicles …

We look forward to feedback and suggestions from the users, regarding the future developments in this area.

Acknowledgements

- The authors would like to thank Shubiao Wang of INSA Rouen of France for permitting us to use his model as an example (Section 8).
- The authors would also like to thank Roger Grimes (Ansys, Inc.) for his assistance in developing the direct SSD feature (Section 6) and Brian Wainscott (Ansys, Inc.) for his assistance in developing the D3MAX feature (Section 9).

Literature

[1] Livermore Software Technology, an ANSYS company, LS-DYNA® Keyword User's Manual, 2020.
[2] Tom Littlewood, Yun Huang, Zhe Cui, Spectral element methods for transient acoustics in Ansys LS-DYNA®, Proceedings of the 13th European LS-DYNA Conference 2021, Ulm, Germany.
[3] Yun Huang, Stefan Hartmann, David J. Benson, Random vibration fatigue analysis based on IGA model in LS-DYNA®, TechCon 2020.