Random Vibration Fatigue Analysis for Bracket of PAB Module with LS-DYNA

Tengteng Wang, Qiang Liu

Autoliv (Shanghai) Vehicle Safety System Technical Center Co.Ltd. Shanghai 201807

Abstract

Random vibration fatigue analysis is performed with LS-DYNA to predict whether PAB (Passenger Air Bag) bracket can pass the vibration durability test. The FE (Finite Element) model is corrected in comparison with hammer test results. The bolt fastening force and contact between bracket and fixture are considered, and the intermittent eigenvalue computation method is used to solve the non-linear eigenvalue problem using LS-DYNA. The random vibration loading excitation and exposure time refers to ISO 12097 standard, and the durability simulation results are consistent with physical shaker-table test results. One code strategy to resolve the assemble, random vibration response and durability life of components with LS-DYNA, which provides effective and high precision solution for vibration fatigue analysis.

Key words: bracket, natural frequency, random vibration, fatigue

1 Introduction

PAB module plays an important role in passive safety system of vehicles. Bracket of PAB module as a connection part between PAB module and vehicle body, which always suffer different loads transferred from a rough road or power train. Make sure PAB bracket is no damage in vehicle's long-term life is the necessary premise for PAB module's well-functioning. The loading on PAB bracket is not known in a definite sense, and the vibration environments are not related to a specific driving frequency. All these features shown that PAB bracket is always working in a random vibration environment. Structural response to random vibration is usually treated using statistical or probabilistic approaches. Power spectral density (PSD) is commonly used to specify a random vibration process, which is an important input data for the research of components' fatigue life.

For many years, fatigue has been a significant and challenging problem for engineers, especially for those who design structures. But now, with the development of fatigue theory and commercial software, fatigue life of components can be predicted by the following two methods. One is accelerated vibration fatigue test with physical shaker-table, the other is CAE simulation method to get a virtual result. CAE simulation for prediction of fatigue life saves considerable time and cost compared with physical shaker-table tests. Fatigue simulation can be performed in time domain and frequency domain. A frequency domain approach based on random vibration theory has been implemented in LS-DYNA for fatigue and durability analysis [1].

In Autoliv, prediction of vibration fatigue life for bracket is a very important gate during product design and development phase of PAB module. PAB bracket shall have to satisfy corresponding vibration fatigue standard of Autoliv or OEM. This paper will share the prediction process of fatigue life for PAB bracket.

2 Model Information and Calibration Process

The main parts involved in PAB module are cover, housing, cushion, retainer, inflator and bracket. The bracket is defined as a simple structure fastened to vehicle body with the purpose of supporting mass of PAB module, which thickness is 2mm. The fixture is designed for vibration fatigue test, which is fastened on shaker-table to mimic the reality working environment of PAB module. The exploded view of PAB module is shown in Figure 1.

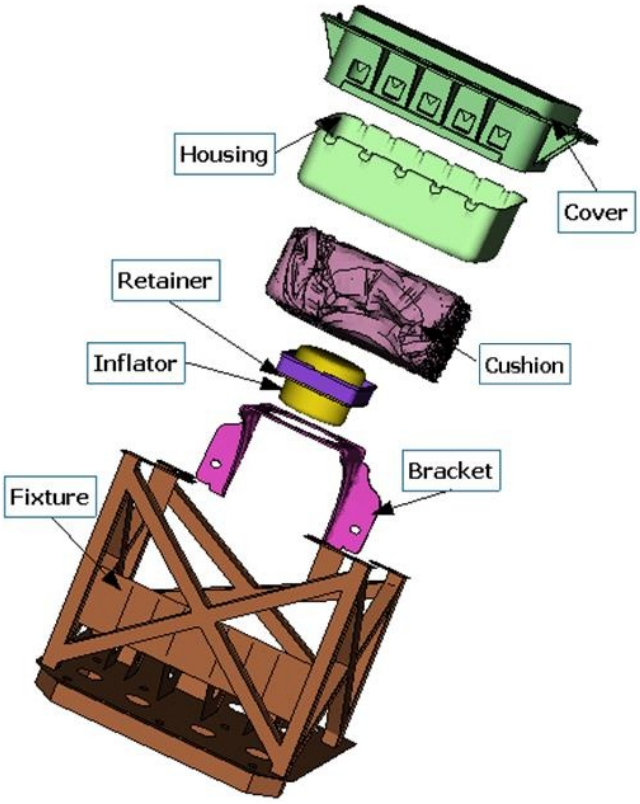


Figure 1 – Exploded View of PAB Module

The FE model of PAB module is composed of 376494 elements and 155871 nodes, which is modelled in ANSA software. The PAB bracket is modelled with hexahedron elements and coated with skin shell elements. Cover and inflator are modelled with tetrahedron elements, while the other parts of PAB module, housing, retainer and fixture are in shell elements. The average mesh size is 2mm with the minimum length of 0.5mm to accommodate details of the bracket in thickness direction. The cushion mass is implemented by non-structural mass on elements of the housing. The fixture is constrained 6 degrees of freedom in the bolt hole position. The FE model is shown in Figure 2.

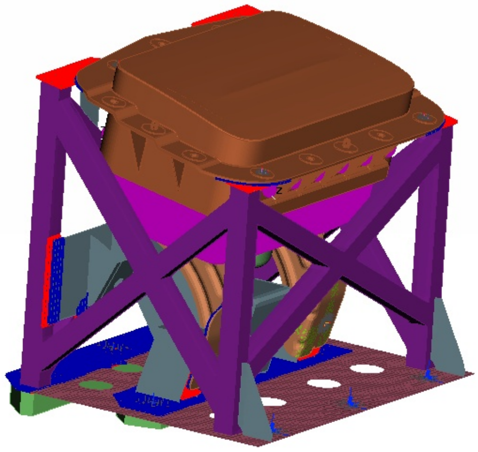


Figure 2 – The FE Model of PAB Module

Random vibration fatigue is solved in frequency domain, and the natural character which main parameter natural frequency is the basic of the whole simulation process. For the PAB module, there are different connections, which highly influence natural frequency of the FE model. How to define constrained among different parts of PAB module is closely related to prediction accuracy of the FE model. The FE model shall be calibrated with related hammer test results. Comparison of modal results with simulation and test can reflect the discrepancy of the FE model. Then the correct connection types for FE model can be found in the process of comparison. The work flow of calibration process for FE model is shown as below (in Figure 3).

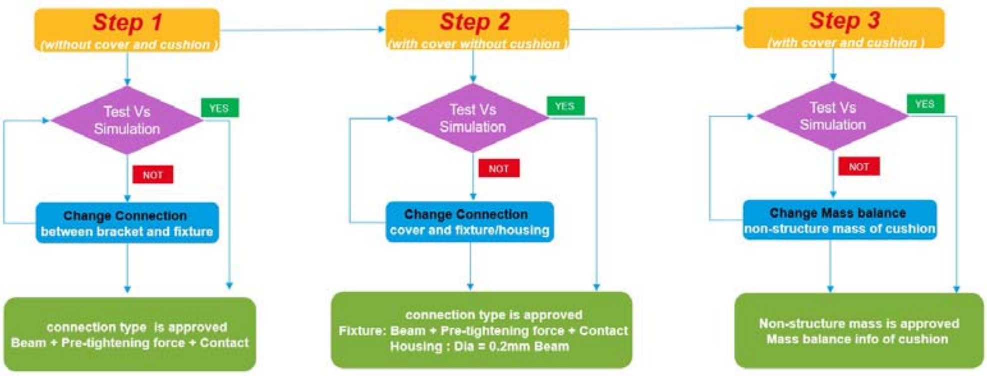


Figure 3 – Work Flow of Calibration Process

In step 1, the connection between bracket and fixture is calibrated first. Different connection types such as nodal rigid body, beam and different constrained area are implemented to mimic the bolt connection between bracket and fixture. At last, the Beam + Pre-tightening force + Contact is approved, which simulation results are close to the hammer test results. The comparison of modal results for the first order frequency is shown in the first row of table 1.

	Test result /Hz	Simulation Result /Hz	Error /%
Step 1	66.63	65.2	-2.1
Step 2	82.67	84.02	1.6
Step 3	70.13	67.58	-3.6

Table 1 – Comparison of modal results

In step 2 and step 3, the connection between cover, fixture, housing, and non-structure mass both are calibrated. The bolt connections between cover and fixture use the Beam + Pre-tightening force + Contact type. The connection between cover and housing use beam element. The mass of cushion uses non-structure mass to balance the distribution of cushion. The comparison of modal results for the first order frequency are shown in the second and third rows of table 1.

In each step of table 1, the simulation results have a good consistency with the test results. The FE model of PAB module can be used in further calculation, which can get a reasonable result.

3 Random Vibration Fatigue Analysis

Random vibration fatigue calculation shall be performed in frequency domain, because of the large structural models and high number of possible load combinations if fatigue life is solved in time domain. The keyword used in LS-DYNA is *FREQUENCY_DOMAIN_RANDOM_VIBRATION_FATIGUE. There are four steps included in the simulation, bolt fastening, modal analysis, random vibration response analysis, and fatigue calculation. These simulation procedures are performed in one code run without being stopped in LS-DYNA.

For step 1, bolt fastening process is performed by implicit analysis in LS-DYAN. Bolt pre-fastening force and contact are considered in the simulation process and fastening force is assumed to be significant effect on fatigue life.

For step 2, the intermittent eigenvalue computation method (*CONTROL_IMPLICIT_EIGENVALUE with NEIG < 0) can be used to solve the non-linear vibration behavior in LS-DYNA, because load and contact can't be considered in linear eigenvalue calculation. This is a transient simulation during which loads are applied, with eigenvalues computed periodically during the simulation [3].

In step 3, random vibration response is calculated, and PSD stresses of components are saved in d3psd. Random loading is provided via the PSD curve according to figure 4, which follows ISO 12097 standard. The PSD curve describes excitation acceleration levels in the frequency domain. Each PAB module is subjected to the specified vibration load in each of the three main axes, so the response in three directions need to be calculated.

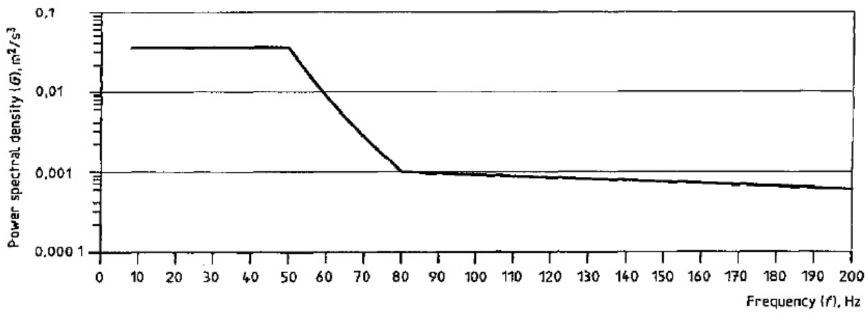


Figure 4 – Random Vibration Input Load

In step 4, the fatigue calculation based on probability distribution and Miner's rule of cumulative damage ratio is performed in LS-DYNA. Dirlik's method is used to calculate the PDF (Probability Density Function) of stresses. The number of stress cycles can be calculated for a given time of exposure [2]. The exposure time for the PAB module is 24 hours along each of the three main axes. The material's S-N curve can be defined by *MAT_ADD_FATIGUE, and the S-N curve used for the material of PAB bracket is shown as below (Figure 5).

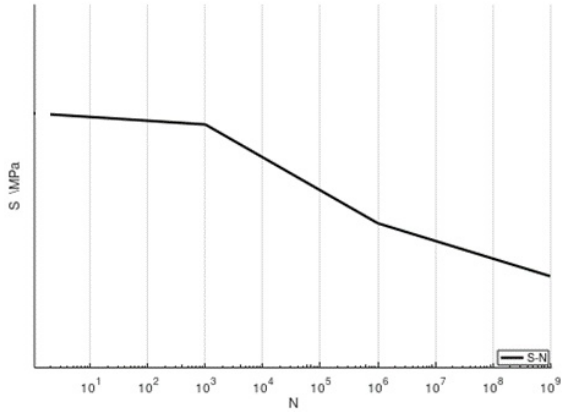


Figure 5 – S-N Curve

4 Results Comparison

There are 4 PAB modules tested on shaker-table for random vibration durability behavior, shown in figure 6. Each PAB module is mounted in a rigid fixture which represents a rigid in-vehicle environment. X axis is defined as longitudinal, Y as lateral and Z as vertical by in-vehicle position in a random vibration test. Test vibration schedule in each direction for 24 hours.

(a) X (b) Y (c) Z

Figure 6 – Vibration Fatigue Test of PAB Modules on Shaker-Table

Random vibration test for PAB modules under base acceleration PSD excitation. The acceleration is specified in three main axes. The PSD curve is given as figure 4 shown for the range of 1-200 Hz. The excitation PSD curve measured on shaker-table to monitor the exciting status is shown in figure 7.

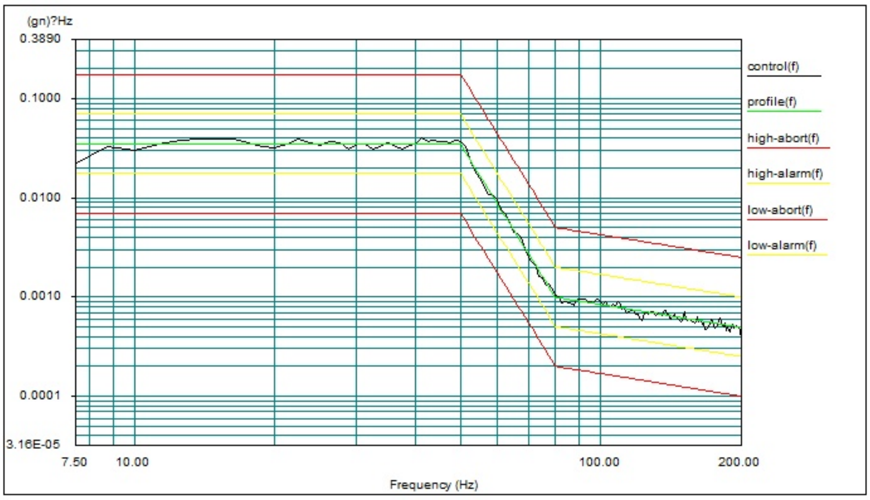
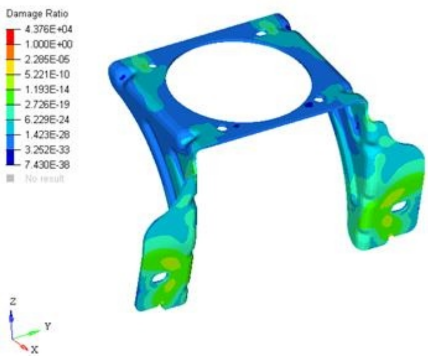


Figure 7 – Load Curve Monitored on Shaker-Table

There is no visible damage in PAB modules after vibration fatigue shaker-table test, one of the PAB module's test results are shown in figure 8.

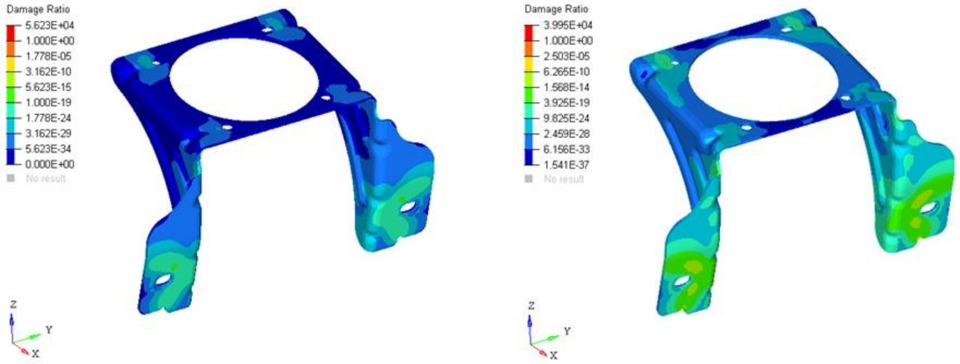
Figure 8 - PAB Modules after Test

The simulation results are shown in figure 9, which shows damage ratio results of PAB bracket. The damage ratio less than 1 in all directions, which represents there are no damage in PAB bracket.



(a) X direction

(b) Y direction



(b) Z direction (d) X+Y+Z

Figure 9 – Damage Ratio Simulation Results

Compared the test results and simulation results, the bracket of PAB module is acceptable, and the simulation method used to calculate the random vibration fatigue in LS-DYNA has a good prediction accuracy.

5 Conclusion

In this paper, random vibration fatigue analysis is performed using LS-DYNA, meanwhile bolt pre-tightening force and contact are considered in simulation. Simulation results have a good accuracy compared with impact modal and shaker-table test, which represents the simulation method can be widely used in fatigue prediction of PAB bracket to reduce cost of shaker-table test.

References

[1] Huang Y, Cui Z: "Recent updates in LS-DYNA frequency domain solvers", Proceedings of the10th European LS-DYNA Conference 2015, Wurzburg, Germany.

[2] Arnaud Ringeval, Huang Y: "Random Vibration Fatigue Analysis with LS-DYNA", Proceedings of the 12th International LS-DYNA Users Conference, 2012.

[3] LS-DYNA keyword user's manual version R11, Livermore Software Technology Corporation, 2018.