Stress Analysis Under Random Loading

James Guo and Norbert Kearney

Robert Bosch LLC, Chassis Systems Control Division

Abstract: *To assess the durability of a structure under random loading, the frequency based random response analysis is performed to obtain the stresses. Since random response analysis ignore the element damping, the results of the random response analysis can be unrealistically twisted if significant local element damping exists. This paper present an Abaqus FEA procedure to more realistically calculate the stress of the structure subject to random loading by submodeling and including element damping, which is a complementary method to Abaqus random response analysis procedure.*

Keywords: FEA, Fatigue, Random vibration, PSD, Mises stresses, frequency, damage, rubber

1. Introduction

The random vibration fatigue can be best handled by the fatigue software such as FE-Safe, The steady state dynamic (SSD) analysis with unit response is first performed to get the FRF (frequency response functions) in terms of stresses for every node in the structure. Then these FRFs are scaled by the input PSDs to get either PSD projections on critical planes or von Mises equivalent PSDs. Whatever the choice, these obtained PSDs are used to evaluate the first four spectral moments to compose the Dirlik's PDF (probability density function) that is integrated to get damage.

Due to limited fatigue analysis capability within ECH-NA (no fatigue software, limited computer power, lack materials properties, etc), and the nature of the fatigue and random vibration (large scatter, statistical nature, etc), detailed fatigue analysis is not practical and did not provide significant advantages. Instead, a fatigue limit guideline based on RMS stresses from random response analysis was used for evaluating the durability of ABS/ESP mounting bracket. This method has been successfully used within Bosch chassis division North America engineering development for bracket durability evaluation.

Although the fatigue guideline based on random response analysis procedure is an efficient method for such application, there are limitations to this method. First, it is not possible to perform submodel analysis using frequency based process, which will limit the mesh size and accuracy of the model; secondly, the random response analysis procedure ignore element damping, estimated modal damping or other types of global damping has to be used, which may twist the results of the random response analysis if significant local element damping exists.

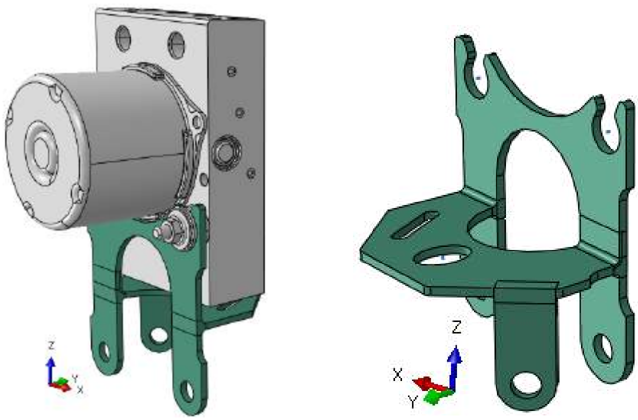
It is the purpose of this research to address above mentioned limitations of the random response analysis procedure, so that more realistic stress analysis can be performed when the random response result is in question or the uncertainty is high. A analysis work flow using various Abaqus procedures and python script were developed. This paper describes such process using a dummy ABS/ESP bracket.

2. Description of the model

Figure 1 shows a dummy ABS/ESP hydraulic brake system including attached bracket and isolators. The goal of the FEA is to estimate the durability of the bracket under random vibrations during lifetime of the vehicle. To assess the durability of the bracket within practically feasible capabilities and time frames, different model simplifications were implemented according to the project needs. For example, the isolator was modeled as 3 dimensional spring and dashpot, the isolator stiffness and damping coefficients were measured by a test device as Figure 2 shows. The measurement is based on resonance method using a sine sweep with constant acceleration.

The measured stiffness and damping can be used directly in transient dynamic analysis or steady state dynamic analysis that account for local damping. For random response analysis, modal damping was estimated from measured isolator damping. The ABS/ESP hydraulic unit was simplified as a block with the same mass properties as ABS/ESP or further simplify as a point mass including rotary inertial effect. The bracket was fixed at the mounting holes and the base motion was applied to the fixed boundaries. This model is constructed to simulate the vibration test that was carried out to check the functional influence during the lifetime of the road load vibration. A road load profile representing lifetime of vehicle road load in PSD form was used to generate input vibration signal, the accelerated vibration load is applied at each axis with total test time equivalent to vehicle lifetime.

The bracket is made of aluminum diecast materials. The analysis was performed with ABAQUS 6.14.2. The global model as well as the submodel are meshed with quadratic HEX-elements.



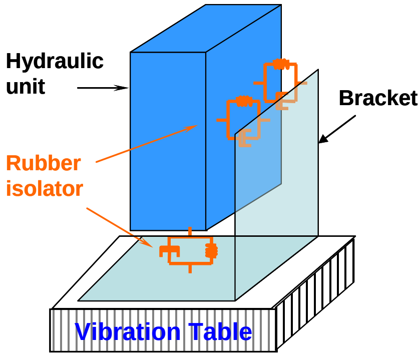
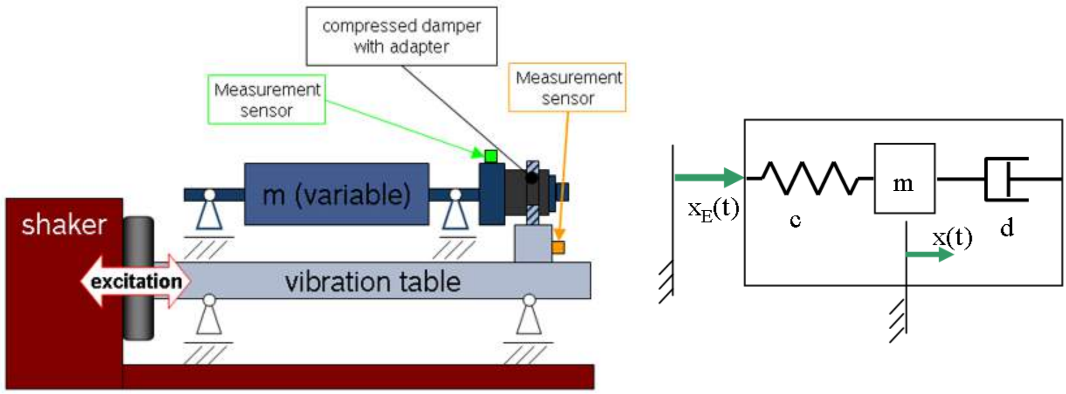


Figure 1: Dummy ESP brake system and model representation



Kelvin-Voigt-Model: $m \cdot \ddot{x} + d \cdot \dot{x} + k \cdot x = d \cdot \dot{x}_E + k \cdot x_E$

$x_E = \hat{x}_E \cdot \sin(\Omega \cdot t)$ harmonic oscillation = **excitation**

$\dot{x}_E = \hat{x}_E \cdot \Omega \cdot \cos(\Omega \cdot t)$ Ω = damped natural frequency

$\Omega = \omega_0 \sqrt{1 - D^2}$ ω_0 = not damped natural frequency

$D = \frac{d}{2 \cdot \sqrt{k \cdot m}}$ Damping-Ratio (according to Lehr

Figure 2 Rubber isolator measurement device and method

3. Description of the stress analysis process

The stress analysis contains the following steps which will be described in detail later:

1. Frequency analysis for global model to determine eigenfrequencies and mode shapes.
2. Random response analysis for global model to identify the highly stress region for submodel (with estimated modal damping).
3. Mode shape extraction to create an equivalent static load case for modal stresses.
4. Submodel analysis with loading from step 3 to determine the modal stresses.
5. Steady State Dynamics (SSD), subspace analysis for global model to determine the generalized displacements and phase angles (local element damping was accounted).
6. Post process above results to determine the stress of the submodel under random loading.

   Then the fatigue life of the part can be evaluated based on more realistic stress from above.

3.1 Frequency Analysis

In the first step the eigenfrequencies and mode shapes of the global model are computed based on Lanczos algorithm. Here, the first 10 eigenfrequencies are calculated, as shown in the ABAQUS output in Table 1.

Table 1. ABAQUS output for frequency analysis

E I G E N V A L U E   O U T P U T

MODE NO	EIGENVALUE	FREQUENCY (RAD/TIME)	FREQUENCY (CYCLES/TIME)	GENERALIZED MASS	COMPOSITE MODAL DAMPING
1	17586.	132.61	21.106	3.64269E-03	0.0000
2	49476.	222.43	35.401	4.22149E-03	0.0000
3	1.38251E+05	371.82	59.177	2.44651E-03	0.0000
4	2.16925E+05	465.75	74.127	6.31115E-03	0.0000
5	3.56316E+05	596.92	95.003	1.76251E-03	0.0000
6	9.62114E+05	980.87	156.11	6.00414E-03	0.0000
7	2.60431E+07	5103.2	812.21	1.81413E-05	0.0000
8	6.79049E+07	8240.4	1311.5	1.06599E-05	0.0000
9	8.65255E+07	9301.9	1480.4	1.93762E-05	0.0000
10	1.42181E+08	11924.	1897.8	1.18341E-05	0.0000

3.2 Random Response Analysis

To identify the highly stressed region in the global model, a random response analysis in ABAQUS standard is performed. The modal damping values for the first 10 eigenmodes are estimated based on the isolator damping. For excitation in y-direction with the acceleration spectrum given on Figure 3, a high stress region is at a corner of the bracket as showed on Figure 4.

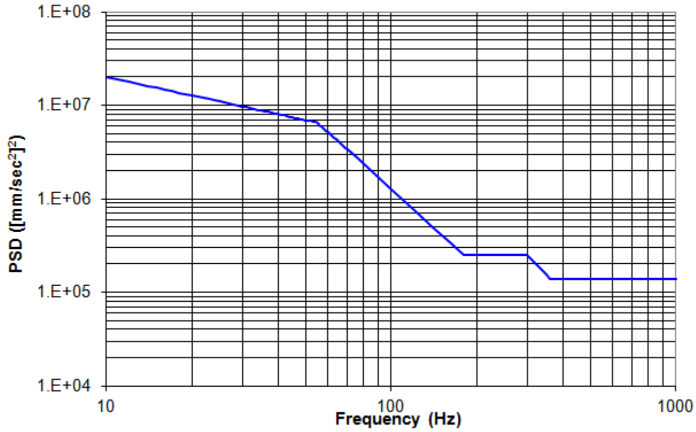


Fig. 3: PSD spectrum

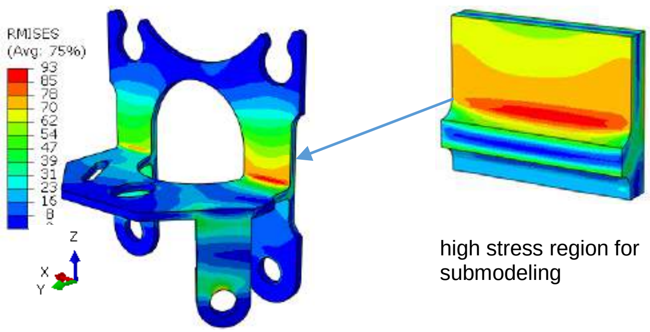


Fig. 4: RMS Mises stresses from random response analysis

3.3 Equivalent static load case extraction

Since ABAQUS does not allow submodel analysis in combination with a frequency step by default, a special submodeling technique for simulations in the frequency domain was developed. By means of the python script, an equivalent static load case for each mode is generated by using the modal displacements from the frequency step as boundary conditions for the static step. Based on these alternative static load cases a submodel analysis can be performed. The extracted mode shapes for the equivalent static load cases are identical to the modal displacements resulting from the frequency step, Figure 5 shows the first three mode shapes.

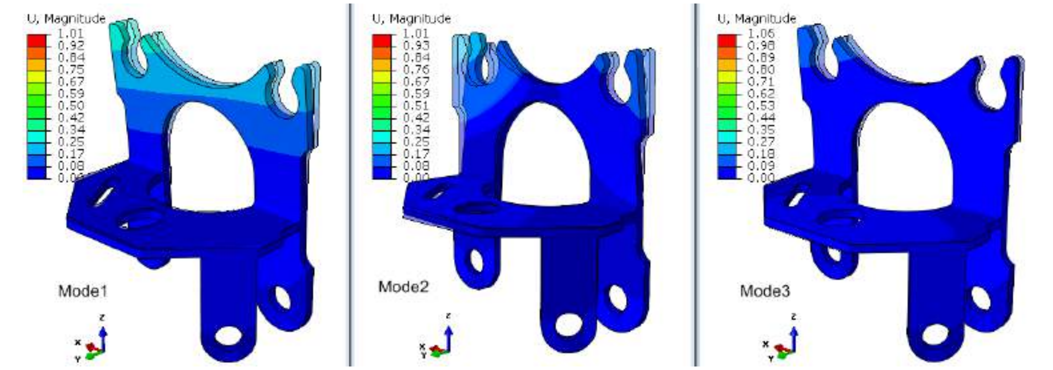


Figure. 5: Mode shapes from equivalent static load case

3.4 Submodel analysis

Based on the equivalent static load cases, the submodel analysis is performed. To reduce the computation time for post processing of submodel results with SSD results later, a smaller region of high stress spots can be defined by a set of nodes and elements in the submodel. The modal stresses resulting from the submodel analysis are shown on Figure 6 for the first three modes.

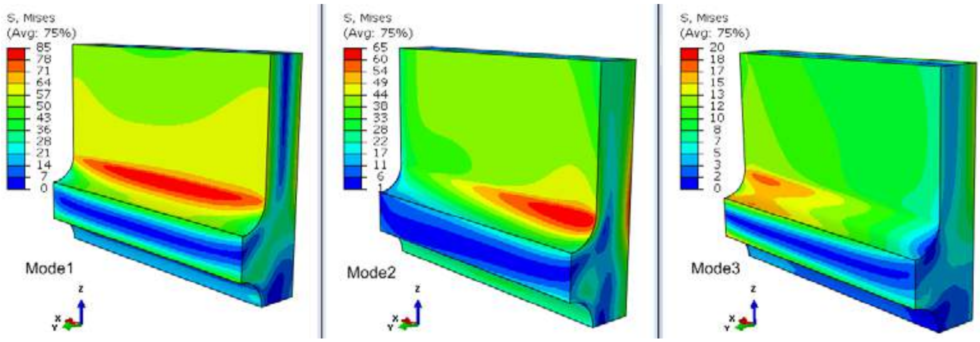


Figure. 6: Submodel analysis results

3.5 Steady State Dynamics (SSD) analysis

Steady State Dynamics (SSD), subspace procedure was performed for the global model with a unit acceleration (a=1 mm/s^2) to derive the generalized displacements and phase angles. SSD subspace procedure take into account of isolator damping, which is ignored at random response analysis. The generalized displacements are shown on Figure 7. The critical modes are indicated by the peaks in the graph of the generalized displacements. It is obvious that for excitation in y-direction the first eigenfrequency play a decisive role.

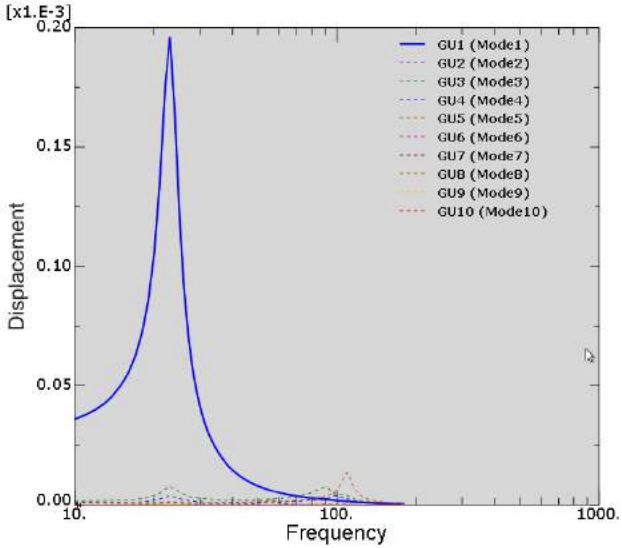


Figure. 7: Generalized displacements

3.6 Post processing based on submodel

The last process is to post process the modal stresses of the submodel (section 3.4) using generalized displacements and phase angles (from section 3.6) and PSD excitation signal (Figure 2). Figure 8 shows the transfer function of Mises stress at hot spot versus frequency from SSD analysis. It is obvious that for excitation in y-direction the maximum stress occurs at the first natural frequency, and some minor peaks can be observed at other frequencies. There are various ways to calculate the RMS stresses of parts subjected to random loading. For example, Segalman’s method and Bonte’s method, Figure 9 shows the RMS values of Bonte stress of the bracket at hot spots.

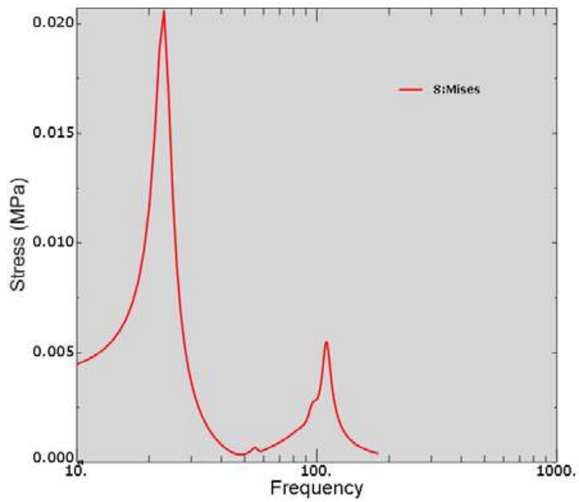


Figure 8 Transfer function of Stress at hot spot

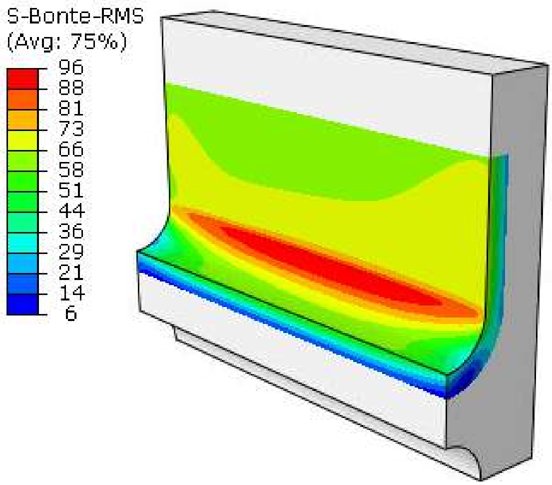


Figure 9 RMS of equivalent PSD-stress (Bonte)

4. Concluding Remarks

It is challenging to predict the fatigue life of a nonlinear system under random vibration. In this paper, we have introduced a stress analysis process to overcome the limitations of random response analysis. This customized method provides more realistic stress for evaluating the fatigue life of a nonlinear system subjected to random loading if significant element damping exist or the random response results are in question and the uncertainty is high, it is a complementary method to Abaqus random response analysis procedure.

5. References

Christian Lalanne, “Random Vibration”, Mechanical Vibration and Shock, Vol. III., Hermes Penton Ltd, pp 33 – 135, 2002.

M.H.A. Bonte,_, A. de Boera, R. Liebregts, “Determining the von Mises stress power spectral density

for frequency domain fatigue analysis including out-of-phase stress components”, *Journal of Sound and Vibration 302 (2007) 379–386.*

Daniel J. Segalman et al, “An Efficient Method for calculating RMS von Mises Stress in a Random Vibration Environment”, Sandia National Lab. Report, 1988.

Abaqus Users Manual, Version 6.10-1, Dassault Systémes Simulia Corp., Providence, RI.

William Thomson, “Theory of Vibration with Applications”. 3rd Edition.