NOVATEUR PUBLICATIONS
INTERNATIONAL JOURNAL OF INNOVATIONS IN ENGINEERING RESEARCH AND TECHNOLOGY [IJIERT]
ISSN: 2394-3696
VOLUME 2, ISSUE 6, JUNE-2015

VIBRATION ANALYSIS OF AN AUTOMOTIVE SILENCER

K. R. Gadre
PG Student, Department Mechanical Engg.,
Sinhgad College of Engineering, Pune

T. A. Jadhav
Associate Professor, Department Mechanical Engg,
Sinhgad College of Engineering, Pune

ABSTRACT

Automotive silencer plays an important role in dynamic performance of the exhaust system. As silencer is located at the tail end of the exhaust system with minimum support, it is subjected to intense vibrations. These vibrations cause localized stresses in silencer. The vibrations and stresses are analyzed by finite element method. Modal analysis is performed to evaluate dynamic characteristics of the silencer. It is observed that natural frequencies are away from excitation frequency. Further, dynamic frequency response analysis is performed with 3G acceleration in all directions varying with frequency. It is noticed that the stresses are concentrated at a joint between inlet pipe and flange and these stresses are above the material yield stress limit in X and Y directions. Hence, in order to reduce the induced stresses, the structural modification in silencer is proposed and subsequent analysis is carried out. The obtained stresses are below the yield stress limit in X as well as Y direction. The FEM results are verified by experimental results.

INTRODUCTION

Automotive silencer is a significant part of the exhaust system. In an automotive engine, pressure waves are generated when the exhaust valve repeatedly opens and emit high-pressure gas into the exhaust system. These pressure pulses generate exhaust sound. As the engine RPM increases, the pressure fluctuations increases and the sound emitted is of higher frequencies. The silencer has to allow the passage for exhaust gases while restricting the transmission of sound. This induces vibrations in the exhaust system. Since, the silencer is at the tail end with limited scope for supporting along its length, the influence of vibrations is most prominent over this component. Also, silencer has to withstand the stresses induced due to fluctuating load. Therefore, it is necessary to analyze vibration characteristics of silencer in order to improve the effective life span and performance of the silencer.

Finite element method and experimental modal analysis have been commonly used for vibration related problems of the exhaust system. Amanda F. and Sarah B. [1] conducted an impact test on a muffler system to determine the resonant frequencies and suggest changes in the system design. In order to minimize the effects of these resonant frequencies, suggested design improvements were to add damping to the system. Wall J.[2] performed modelling, simulation and experimental investigation of a typical exhaust system. Investigations showed the great reduction of vibration transmission to the exhaust system when a bellow type joint was used as compared with a stiff joint. Rajadurai S. and Suresh N.[3] performed a static analysis of the exhaust system with 1G gravity load. Dynamic frequency response analysis was performed with enforced acceleration as 3G load in all direction varying with frequency. The Von-mises stresses were compared with the material yield stress limit for failure. Researchers also used an integrated approach to optimize silencer design using CAE and CFD techniques[4]. Using the integrated approach, it was possible to optimize the design and meet the two conflicting requirements, acoustic and engine performance and reduce the design cycle time. Jie W. and Yue D.[5] performed modal analysis to study the vibrations of muffler to distinguish the working frequency from natural frequency and avoid resonance. The natural frequencies and mode shapes were considered during the design of the muffler to avoid the resonance occurred in the exhaust system.

FINITE ELEMENT ANALYSIS

Finite Element Method is a numerical method used for obtaining the approximate solution of engineering problems. In this method, the complex region or body defining a continuum is discretized into simple geometric shapes. When the loads and boundary conditions are applied, a set of linear or nonlinear equations is usually obtained. The solution of these equations gives an approximate solution of the problem.

In this work, modal analysis of the silencer is performed with an FEA methodology to find out natural modes of vibration. Dynamic frequency response analysis is also performed to find out the localized stresses induced in silencer.

MODAL ANALYSIS

Modal analysis is a method to describe a structure in terms of its dynamic characteristics, which are frequency, damping and mode shapes. The natural modes of vibration are inherent to a dynamic system and are determined completely by its physical properties and their spatial distributions.

3D model of the silencer is created by using CAD modeling software CATIA. Fig.1 shows a front view of silencer mentioning different parts. Material properties used for silencer material are, Young's modulus of elasticity E= 2x 10^5MPa, Poisons Ratio = 0.3, material density = 7850 Kg/m^3. Mid-surfaces are extracted using mid-surface option in HyperMesh. As the thickness of the silencer is small as compared to other dimensions of the silencer, shell elements are used. TRIA3 element is stiffer than QUAD4 elements, so QUAD4 element selected for meshing. The total number of elements observed in silencer model is 6117 and nodes are 6060. Meshed model checked for Jacobian and Aspect Ratio. Constraints are applied at the inlet tube of the silencer at clamping location. All Degrees of Freedom (DOF) U_x, U_y, U_z =0 and R_x, R_y, R_z =0. The EIGRL card image selected for modal analysis. The excitation from an automobile engine is usually in the frequency interval of 30-200 Hz. Therefore, frequency range considered for analysis is from 0 to 250 Hz.

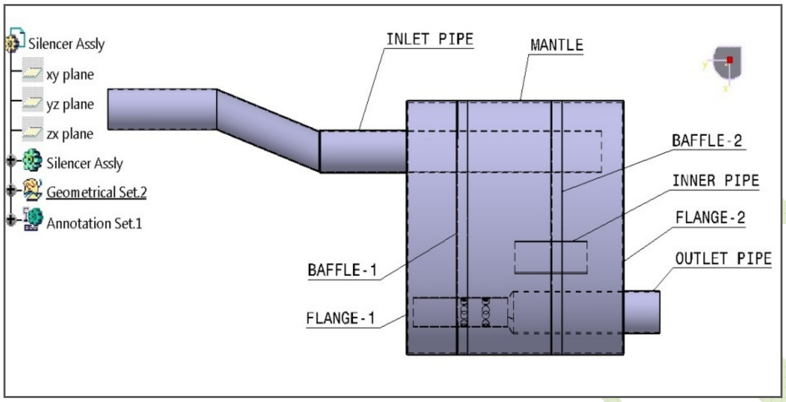


Fig.1: Front view of silencer

.

Table 1: Natural frequencies of silencer

Mode Order	1	2	3	4	5	6
Frequency (Hz)	96.14	120	164.35	224.02	243.49	247.43

In Nastran, SOL 103 card is used for normal mode analysis. After the analysis is completed, post processing is done in HyperView. First six natural frequencies are extracted by using a Block Lanczos algorithm. The first six natural frequencies obtained are given in Table 2. Fig.2 shows the first mode of vibration which is bending about Z axis with natural frequency 96.14Hz.

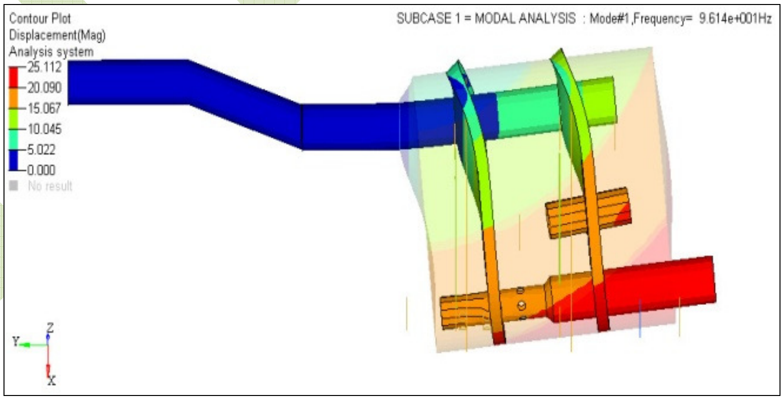


Fig.2: Mode 1- 96.14 Hz

DYNAMIC ANALYSIS

For Frequency Response Analysis (FRA), constraints are applied at the inlet tube of the silencer. The frequency considered for the analysis is first natural frequency, which is 96.14Hz. RLOAD2 card defines frequency dependent dynamic load. Dynamic load applied at constraint point for x, y and z directions in different load collectors. DLOAD defines the dynamic load given in RLOAD2 card. Load collector for frequency with FREQi card and first natural frequency is created. Acceleration data assumed as 3G acceleration and applied at first natural frequency. HyperMesh is used for deck preparation and Nastran is used as a solver.

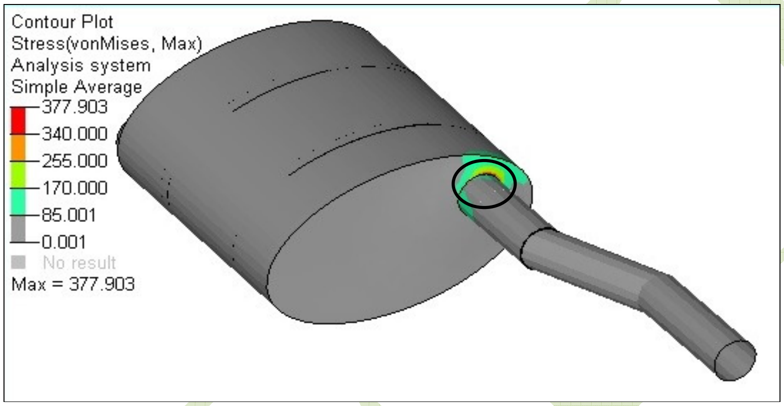


Fig.3: Stress plot for excitation in X direction

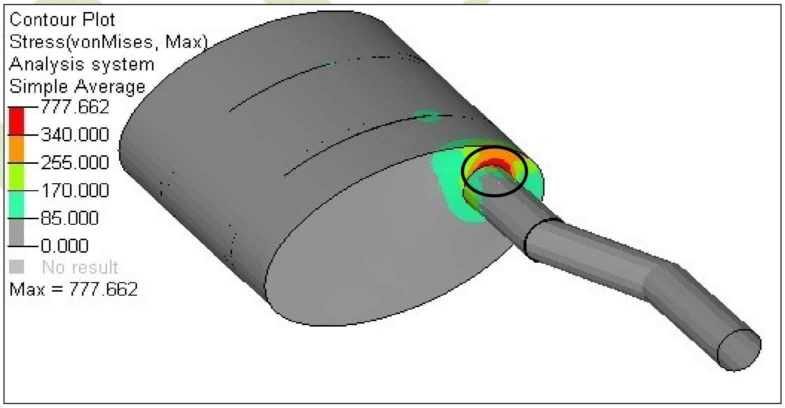


Fig. 4: Stress plot of FRA for excitation in Y direction

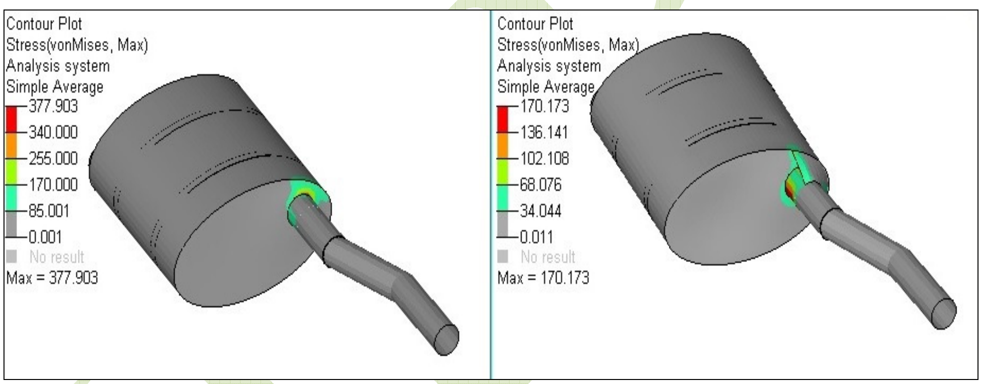
The Von-mises stresses obtained by FRA compared with the material yield stress for failure. The tensile yield strength of the material of the silencer is 340 MPa. Stress plot of FRA for excitation in X direction with the first natural frequency and 3G acceleration is shown in Fig.3. Maximum von-mises stress being 377.9 MPa and at the joint of inlet pipe and flange-1. Fig.4 shows the stress plot of FRA for excitation in Y direction with a first natural frequency and 3G The maximum von-mises stress is 777.66 MPa and is at the junction of the inlet pipe and flange-1. For excitation in Z direction, maximum stress observed is 157.07MPa and is

ISSN: 2394-3696
VOLUME 2, ISSUE 6, JUNE-2015

below material yield stress. From the analysis, it is observed that the maximum von-mises stress is above material yield stress for X as well as Y direction. It is also observed that the stress is concentrated at a joint between inlet pipe and flange-1. This area is considered for further improvement in design.

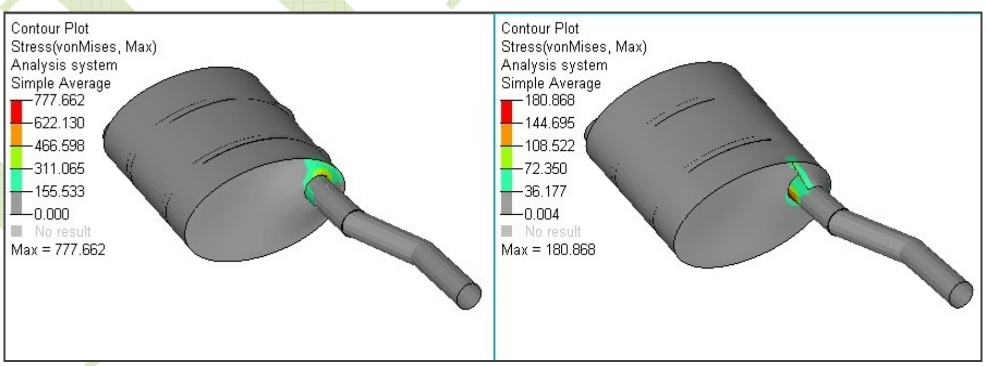
SILENCER MODIFICATION

From FRA of the silencer, it has been seen that the maximum stress on silencer is at junction of inlet pipe and flange-1. The maximum stress is above the yield stress limit. Hence, it is necessary to modify the geometry of the silencer. New bracket modeled at a joint between inlet pipe and flange-1. The thickness of bracket finalized as 1.25mm. Thickness of both the baffles changed from 1mm to 0.8mm and hole of diameter 24mm added on baffle-1. The above changes are incorporated into the model of silencer. With the above changes in geometry of silencer, modal analysis is performed on modified silencer geometry. It is observed that, the first natural frequency of modified silencer is 117.19Hz. Frequency is increased from 96.14 Hz to 117.19 Hz, when compared with existing silencer.



Existing Silencer Modified Silencer

Fig.5: Comparison of stress plot for FRA in X direction



Existing Silencer Modified Silencer

Fig. 6: Comparison of FRA stress plot for FRA in Y direction

Frequency response analysis performed for the modified geometry of the silencer. The procedure for FRA was same as that applied to original silencer design. For modified silencer geometry, maximum Von-mises stress for excitation in X direction is 170.13MPa as against 377.9MPa for original silencer and is shown in Fig.5. For excitation in Y direction, Von-mises stress for modified silencer is 180.86MPa and shown in Fig. 6. From the above analysis, it is observed that the maximum Von-mises stress for modified silencer geometry is below yield stress for excitation in X as well as Y direction.

EXPERIMENTAL ANALYSIS

Experimental results are required for supplementing the analysis by providing certain basic data or parameters that cannot be predicted precisely by analytical or numerical methods. Experimental modal analysis is a technique used to derive the modal model of a linear time-invariant vibratory system. A typical measurement setup for experimental modal analysis consists of three constituent parts. The first part is responsible for generating the excitation force and applying it to the test structure, the second part is to measure and acquire the response data and the third part provides the signal processing capacity to derive FRF data from the measured force and response data.

For experimental modal analysis of silencer impact hammer is used for giving excitation to the silencer. The temperature of silencer measured, to assure it is within operating temperature range of accelerometer. An accelerometer used has a magnetic base and is located at the end of the silencer. The FFT spectrum analyzer is used to analyze the data.

With the use of the above equipments experimental test setup is prepared. The impact is given at the end of silencer by using impact hammer. Analyzer receives analog voltage signals from the accelerometer. The analyzed signals used to find natural frequencies and mode shapes in graphical form with the use of software DEWSOFT installed in the computer. The graph of acceleration verses frequency for impact test is shown in Fig.7. From the graph, the first peak is observed at 104.98Hz with acceleration 0.2165 m/s^2. Hence, from the graph first natural frequency of the silencer is 104.98Hz. The response from silencer is also measured when the engine is at idling condition. From the response at idling condition, excitation frequency is observed at 51 Hz.

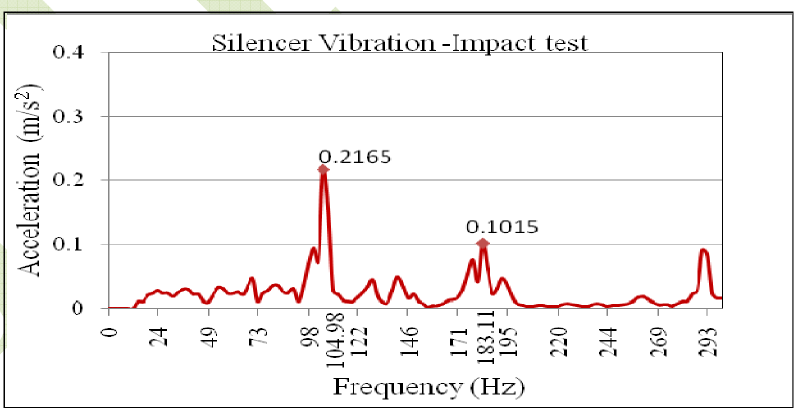


Fig.7: Acceleration Vs. Frequency for impact test

CONCLUSIONS

The silencer is subjected to intense vibrations which are responsible for fluctuating loads and hence localized stresses. By using FEM, modal analysis is performed to find out dynamic characteristics and FRA is performed to find out localized stresses induced in silencer. From modal analysis, it has been observed that, the first natural frequency is at 96.14 Hz. As this frequency is above excitation frequency, existing silencer is safe from resonance. Experimental modal analysis is performed for existing silencer using FFT analyzer. Fundamental natural frequency observed at 104.98 Hz, which is close to simulation results.

Dynamic frequency response analysis performed with 3G acceleration varying with first natural frequency in all directions. The Von-mises stresses observed are 377.90 MPa and 777.66 MPa in X and Y direction respectively. These stresses are above the yield stress limit. It is also noted that, the stresses are concentrated at a joint between inlet pipe and flange. Hence, the geometry of the silencer is modified. Frequency response analysis performed for modified silencer geometry showed reduction of stresses compared to original silencer and these stresses are well below the yield stress limit. Modal analysis performed for modified silencer revealed that the fundamental natural frequency is above excitation frequency. Hence, modified silencer is also safe from resonance.

REFERENCES

[1] Amanda F., Sarah B. *'Vibration analysis on an automobile muffler', EGR 315: Mechanical Vibrations, 2003, pp.1-16*.

[2] Wall J., *'Dynamic study on an automobile exhaust system' Karlskrona 2003' Sweden, 2003, pp. 1-30.*

[3] Rajadurai S., Suresh N., *'Systematic FEA study of passenger car exhaust system using radioss', HTC 2011, pp. 1-13*.

[4] Pujari N., Mahajan S., Mohite *'Optimization of silencer - an integrated approach of acoustic performances & backpressure', International Journal of Emerging Science and Engineering (IJESE), ISSN: 2319-6378, Volume-2, Issue-2, 2013, pp. 21-25.*

[5] Jie W. and Yue D., *'The modal analysis of automotive exhaust muffler based on pro-e and ansys', 3rd International Conference on Advanced Computer Theory and Engineering(ICACTE), 2010, pp. 251-253.*