STRUCTURAL OPTIMIZATION & DURABILITY ANALYSIS OF VW BETTLE CROSS EXHAUST MUFFLER SYSTEM

Vikram Kumbhar[1], Prof D.C.Patil[2]

[1]Post Graduate Student, Department of Mechanical Engineering, KLEMSSCET, Karnataka, India
[3]Professor, Department of Mechanical Engineering, KLEMSSCET, Karnataka, India

---***---

Abstract - *The exhaust system is to route the exhaust gases from the engine and keep the gases away from the passenger compartment, vibration is transferred from engine to the exhaust system and then transfer to the body structure. The flex is used to reduce the vibration transferring from the engine to exhaust system and the hanger isolators is used to Reduce the vibration transferred from exhaust system to body. The CAD modeling is done using CATIA V5 and Finite element modeling is carried out for the automotive exhaust system of Muffler using Altair's pre-processing tool HYPERMESH tool. The analysis is executed by the Optistract tool and the results are observed in the Altair's post-processing tool HYPERVIEW.*

Static and Fatigue Analysis are carried out to the exhaust system components to determine the high stress region at different loading condition one is for self-weight of the exhaust system of Muffler, and another one for the bad road condition observing that that how the structure of exhaust system behaves and also the maximum displacement and the reaction are observed at the hanger locations. Modal analyses are carried out to determine the structural behavior of the exhaust system.

Experimental modal analysis, also known as modal analysis or modal testing, deals with the determination of natural frequencies and mode shapes through vibration testing. The exhaust muffler in an automobile plays an integral role in reducing the sound of the automobile, as well as the ride itself. In order to maintain a desired noise and comfortable ride, the modes of a muffler need to be analyzed. Modal analysis is done both experimentally through FFT analyzer and finite element analysis. The natural frequencies obtained by both the methods agree with each other. This is useful while designing of exhaust muffler to avoid the resonance.

Key Words: Experimental Modal Analysis, Exhaust Muffler, FEM, Modal Analysis, Static Analysis, Fatigue Analysis

1. INTRODUCTION

An automobile exhaust system has several functions. Originally, it was used for silencing the noise caused by high pressure exhaust gases leaving the engine and for transporting these hot and toxic gases away from the driver's compartment. Nowadays, it is also an important and integral part of combustion and emission control. For this to work properly there must be no leakage upstream of the catalytic converter. The durability of that part of the system is therefore crucial.

The sole purpose of an automotive muffler is to reduce engine noise emission. If you have ever heard a car running without a muffler you will have an appreciation for the significant difference in noise level a muffler can make. If vehicles did not have a muffler there would be an unbearable amount of engine exhaust noise in our environment. Noise is defined as unwanted sound.

1.1 Problem Statement

This project will focus on the normal modal analysis, static analysis of an exhaust system and also prediction of fatigue life of muffler mounting brackets w.r.t which optimization of mounting bracket is carried out.

- To carry out normal mode analysis of a VW BETTLE CROSS engine oil pan (with oil and without oil both numerical (Finite Element Method) and experimental approaches.
- To carry out a fatigue failure analysis of a BETTLE Engine oil pan to mounting brackets used to consider the design changes.

1.2 Project Scope and Methodology

The normal and static modal analysis from finite element method has been carried out on a VW BETTLE CROSS engine oil pan. The static analysis of an engine oil pan without oil and with oil by both numerical approach (FEM) and experimental approach gives the mode shapes of a thin walled stamped steel structure. Fatigue analysis is performed to obtain the fatigue life of exhaust system mounting locations.

Following are the methodology adopted in carrying out the project

- Literature survey on fundamental finite element techniques to evaluate normal modal analysis under free-free condition and the engine oil pan structural vibrations due to its thin walled structure.
- From the literature review the following input parameter were collected.

- Boundary conditions to be used to carry out finite element analysis and experimental analysis. Frequency range of interest for engine oil pan excluding the baffle plate.
- Equipment's used for experimental normal modal analysis.
- To calculate the fatigue failure of mounting brackets and predict the fatigue failure limits.

1.3 Project Approach

- The physical component VW BETTLE CROSS engine oil pan was measured for its dimension and CAD model was prepared using CATIA-V5 R17.
- The finite element model for normal modal analysis and static analysis modeled in ALTAIR-HYPERWORKSv10.
- Finite element analysis will be carried out with the application of input parameters and boundary conditions in Optistruct.
- Estimate the fatigue life of muffler mounting brackets using MSC fatigue software.
- Optimization of mounting brackets w.r.t fatigue life criteria.
- Results obtained from the finite element method were validated by conducting experimental analysis.

2. FINITE ELEMENT MODEL BUILDING AND RESULTS

Finite element analysis was used to finding out the Eigen values and Eigen vectors of an engine oil pan. From the literature survey and fundamental finite element techniques, the VW Battle muffler was post processed to understand the structure and dynamic behavior, and to compare finite element analysis results to experimental test data. The task included preparing the geometric model, FE-model, determining the loading conditions and evaluating the mode shapes and natural frequencies of an engine oil pan. The work carried out under this chapter is briefly explained below.

2.1 Creation of Geometric Model

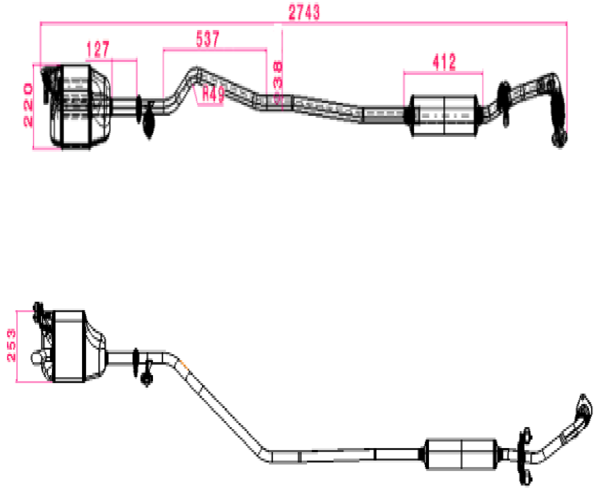

Figure 2.1 2D Drafted CAD model

The physical component of exhaust muffler was bought and measured its all dimension with the help of measuring tape, vernier caliper and a micrometer. These dimensional data was used to build CATIA part model by using CATIA V5 R17.

Figure 2.2 Physical Model of an Exhaust muffler

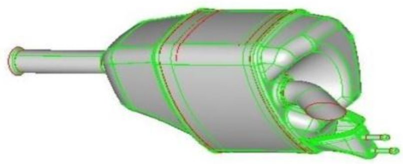

Figure 2.3 CATIA Modal of an Exhaust Muffler

After creation of CAD model in CATIAV5 R17 to carry out further pre-processing process the CAT. part was imported in Hyper mesh during this process some geometric deviation will occur that needs to be corrected (geometric clean up) in Hyper mesh. Engine oil pan component modeled by using linear first-order (3-or 4-noded) shell elements at the part mid surface. Shell elements are quadrilateral and triangular elements.

Table 1 Material Properties of an Engine Oil Pan

Properties	Value	Units
Young's Modulus [E]	2.08e5	N/mm2
Poisson's Ratio [Nu]	.31	-
Density [Rho]	7.85e-09	Tonne/mm3
Mass [M]	4.721e-3	Tonne
Yield Stress	350	N/mm2
Ultimate Strength	450	N/mm2

The Figure 2.4 shows the meshed model of Exhaust Muffler. Meshing is done using Altair Hyperworksv10 pre-processor. There are three options in the "create mesh" sub-panel of Hyper mesh. The switch allows one to choose from the interactive, automatic, and QI optimized selections. The interactive option is used to auto mesh multiple surfaces or elements with user-controlled parameters.

Figure 2.4 Meshed Model of Muffler

2.2 Modal Analysis of Exhaust Muffler Result

Table 2 Frequency Table (Free – Free Modal Analysis)

Mode NO.	Frequency (HZ)
1	2.553E-03
2	1.366E-03
3	6.289E-03
4	9.136E-02
5	1.667E-03
6	7.915E-03
7	2.083E+02
8	3.411E+02
9	6.820E+02
10	1.537E+03
11	1.564E+03
12	1.580E+03

2.3 Static Analysis Results

By the next step the static analysis is performed on the exhaust system as per the loading conditions

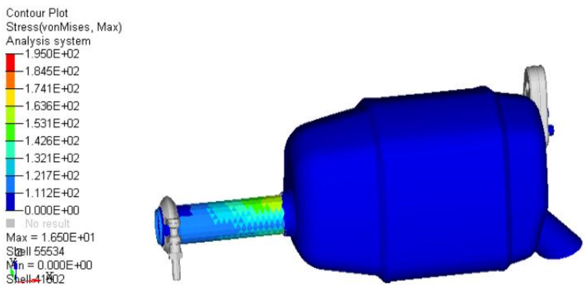

Figure 2.5 Von misses stress plots of muffler

The figure 2.5 shows the stress plots of muffler. In the above figure maximum Von Misses stress of 195.0 Mpa.

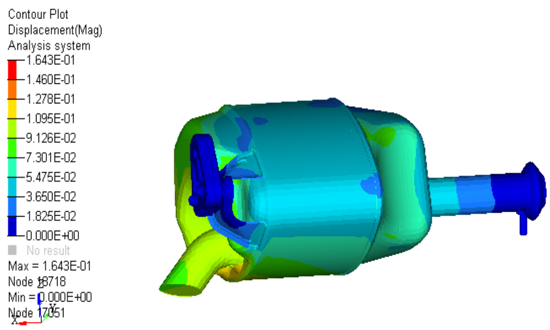

Figure 2.6 Displacement plots of muffler

The figure 2.6 shows the displacement plots of muffler. In the above figure maximum displacement of 0.16 mm is observed which is very negligible

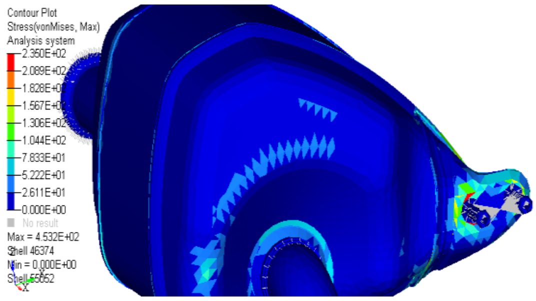

Figure 2.7 Von misses stress plots of muffler

The figure 2.7 shows the stress plots of muffler. In the above figure maximum Von Misses stress of 235.0 Mpa

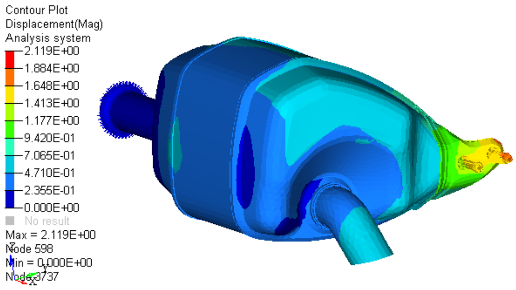

Figure 2.8 Displacement plots of muffler

The figure 2.8 shows the displacement plots of muffler. In the above figure maximum displacement of 2.1 mm is observed which is very negligible at mounting locations.

2.4 Fatigue Test Analysis-Results

Fatigue test was done on the mounting brackets to calculate its fatigue life and predict the fatigue life of mounting brackets of exhaust system

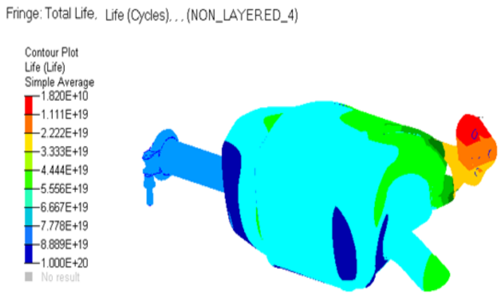

Figure 2.9 Fatigue life prediction of muffler mounting bracket

As seen from above figure 2.9 fatigue life contour plots min. life is 1.82E10 Cycles which is at mounting location& Max. life about 1E20 Cycles which is to be found safe.

3 EXPERIMENTAL VALIDATIONS

The experimental validation is done by using FFT (Fast Fourier Transform) analyzer

3.1 Experimental setup

Figure 3.1 Experimental set up for LMS Impact testing

3.2 Experimental Results

Simple Connections are made from vibration measuring instrument to accelerometer mounted on target position on Muffler that is near to bracket at Muffler sensor sensitivity is set in X Y Z directions

Experimentation was carried to find out stress developed at Muffler bracket as FRA result shows stresses were found more at Muffler bracket so accelerometer was put on Muffler nearer to bracket

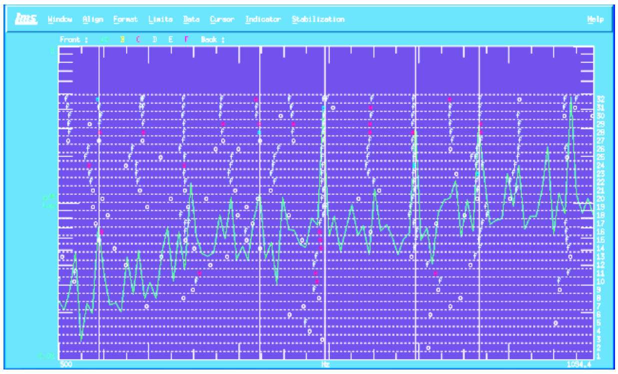

Figure 3.2 LMS Impact Testing Experimental Results

The natural frequency obtained is tabulated in Table 3

Table 3 modal frequency of vibration by FFT analyzer

Mode No	Frequency(HZ)
7	204.52
8	334.20
9	678.41
10	1529.23
11	1562.5
12	1578.56

4 Results and Discussion

For the purpose of optimizing of mounting location, modal analysis was performed. Below table shows the comparison of modal frequencies obtained for two exhaust systems using FEA and Experimental modal analysis approach. Table 4 shows the comparison of Experimental and Simulation Results

Table 4 Comparison of Experimental and Simulation Results

Mode Number	Experimental	Simulation (Free-Free Modal)	Error Percentage
7	204.52 Hz	208.3 Hz	1.85 %
8	334.20 Hz	341.1 Hz	6.91 %
9	678.41 Hz	682.0 Hz	3.59 %
10	1529.23 Hz	1537.0 Hz	7.77 %
11	1562.5 Hz	1564.0 Hz	1.55 %
12	1578.56 Hz	1580.0 Hz	1.44 %

5 CONCLUSION AND RECOMMENDATIONS

In this project work, CAE simulation is designed to address fatigue sensitive Muffler Bracket-Hanger design in the area of the muffler when installed in a vehicle exhaust system. This simulation is used to evaluate the vertical direction fatigue durability of Bracket- hanger on and adjacent to the muffler. Muffler bracket design in this case has a hanger rod which is welded to the bracket.

The muffler bracket is the critical part in the exhaust system as it carries the whole system through hanger rod. The road load and engine vibrations are transferred to the system through the hangers. Simulation of the muffler bracket is carried out to calculate the stresses due to oscillating loads and following points were concluded.

- The average maximum Von-misses stress due to this loading is around 235MPa.
- A durability analysis is performed to predict the life and damage of the muffler bracket design. The S-N Curve is generated on basis of UTS in the Fatigue Process Manager.
- The durability analysis is carried out with the load data from road load as Fz.
- The FPM approach is followed to determine the damage and life.
- Minimum life of the component found to be 1.820E10 cycles which are found to be safe.
- Accelerations observed in experimental and simulation for impact test of muffler was found to correlate with each other.
- Depending on acceleration data it was concluded that the new VW Beetle exhaust system was found to be safe.

5.1 Future recommendations

- We can develop a sophisticated & optimized exhaust system for a up taken VW Beetle vehicle is essential.
- This cad model can be easily converted to a optimized FEM which can be very efficiently used to size the test rig based on fatigue life estimation, strength and deformation requirement.
- A 3D cad model helps to visually the set up with clarity and gives a high confidence of success.
- A fatigue crack normally initiates from the location maximum tensile stress in the structure will also help to predict the fatigue life in mounting region.
- The fatigue calculation is carried out for an estimation of life to crack initiation.
- Once validation by FEA, the cad model can be used generate fabrication drawing for all structural members.

REFERENCES

[1] Bartlett, H.Whalley, R., "Modeling and analysis of variable geometry exhaust gas systems", Applied Mathematical modeling 22, 1998.

[2] S.Rajadurai N.Suresh – R&D sharda motor industries ltd, chengalpattu taluk, Chennai, "systematic fea of passenger car exhaust system using Radios," Hyperorks Technology conference, Chennai 2011

[3] Johan Wall Karlskrona, "Dynamics Study of an Automobile Exhaust System" Department of mechanical enginnering

[4] Sanjay S. Patil and V. V. Katkar, Tata Motors Ltd. "Approach for Dynamic Analysis of Automotive Exhaust System" published on Commercial Vehicle Engineering Congress & Exhibition Rosemont, Illinois October 7-9, 2008.

[5] T. Lauwagie1, J. Strobe, E. Dascotte1, J. Clavier, M. Monteagudo, "Optimization of the Dynamic Response of a Complete Exhaust System", Dynamic Design Solutions, Interleuvenlaan 64, B-3001, Leuven, Belgium.

[6] Amanda Frederick, Sarah Brady "Design project on vibration analysis on automobile muffler" Mechanical Vibrations, April 23, 2003.

[7] Verboven, P., Valgaeren, R., Van Overmeire, M. and P. Guillaume, "Some comments on model analysis applied

to an automotive exhaust system", Proceedings of the International Modal Analysis Conference (IMAC),

[8] M Rajasekhar Reddy & K Madhava Reddy. "DESIGN AND OPTIMIZATION OF EXHAUST MUFFLER IN AUTOMOBILES" International Journal of Automobile Engineering Research and Development

[9] V.P. Patekar "Experimental Study of a Exhaust Pipe Using FEM and FFT Analyzer", International Journal of Innovative Research in Science, Engineering and Technology, Vol. 5