Finite element analysis of diesel exhaust fluid tank holding bracket

Aditi Subhedar¹ and Bhavesh Bohra¹

¹Wainganga College of Engineering and Management, Nagpur, India
E-mail: aditimsubhedar@gmail.com

Abstract. With low running costs, diesel engines have performance, longevity and reliability. In addition to their benefits, they play a very significant role in the degradation of the atmosphere. Hence after treatment technology is adopted to control the emissions caused by these engines to treat the exhaust gas coming from the engine. Selective catalytic reduction is one such method after treatment that is used to treat the NOx (Nitrogen Oxides) content of the engine's exhaust gas. Diesel Exhaust Fluid tank holding bracket supports the (Diesel Exhaust Fluid) tank and its complete assembly. Diesel Exhaust Fluid tank and its assembly is responsible for storage, calculated injection of Aqueous Urea Solution 32. Weight reduction of the Diesel Exhaust Fluid tank holding bracket is a major parameter of concern as it helps us increase the payload capacity of the vehicle and also increase the overall fuel efficiency. This paper presents the steps followed while performing the Finite Element Analysis on the existing Diesel Exhaust Fluid tank holding bracket. Based on the results of the existing dimensions, the new dimensions are designed and analysed.

Keywords. Weight reduction, Finite element analysis (FEA), Diesel Exhaust Fluid and bracket.

1. Introduction
A big problem has been the handling of exhaust gas coming out of the diesel engine. Nitrogen oxides (NOx), carbon monoxide, sulphur oxides and particulate matter are contaminants that are significant causes of concern. Selective Catalytic Reduction is one such technology used to treat nitrogen oxides. Selective Catalytic Reduction is an advanced technology for pollution control that injects liquid reduction fluid known as Diesel Exhaust Fluid or aqueous urea solution (AUS 32) into the diesel engine exhaust stream.

2. Description of diesel exhaust fluid tank holding bracket and its assembly
The Diesel Exhaust Fluid tank holding bracket and its assembly components include the Diesel Exhaust Fluid tank, the air/oil separator, and the dosing module. The dosing module is another assembly consisting of the dosing pump, the level sensor within the Diesel Exhaust Fluid tank and the respective ties between all these components. The aim of the dosing pump is to pump the Diesel Exhaust Fluid from the tank and spray it into the engine exhaust stream into the catalyst. Inside the Diesel Exhaust Fluid tank, the level sensor tracks the level of Diesel Exhaust Fluid present inside the tank. On the bracket, the air / oil separator is installed. It is not specifically involved in the process of the after-
treatment device, but it adds weight to the bracket, which could impact the bracket's holding power. Therefore, we also consider the weight of the air / oil separator.

3. Experimental work
Different methods adopted for weight and cost reduction of the Diesel Exhaust Fluid Tank holding Bracket includes, Material Change, Geometry Change and Thickness Reduction. Depending on its use, the Diesel Exhaust Fluid tank keeping bracket is evaluated in different ways. The vibrations due to various road conditions can move through the hull to the bracket as the tank bracket is mounted to the vehicle's chassis, so Modal and Power Spectral Density analysis has been conducted to determine the natural weld frequency and intensity, respectively. Static structural analysis has been performed to determine the strength of the material to sustain the load. The safety factor of the diesel exhaust fluid bracket could be influenced by the application of the methods above. Therefore, Integrated Design Failure Modes and Effect Analysis (IDFMEA) is performed to define the possible failures and Finite Element Analysis to assess the bracket's structural capacity for different materials of different design and thickness geometry.
Selection of alternative material depends upon following parameters: - The results of the study of various materials help us decide the best material to be used to minimise the overall bracket weight and expense. The selection of material was done considering parameters like, Sustenance of Chassis Vibration, Compatibility with DEF, Ease in Manufacturing, Availability of sheets in desired thickness, Cost of material. The different materials selected for analysis are: -
1. Structural Steel
2. Mild steel
3. Aluminium alloy
4. Stainless steel alloy.
Finite Element Analysis (FEA) is a virtual computerised method simulation process to check if the component maintains the respective load conditions it is given. It is a method of failure analysis which determines a component's probability of failure. Depending on the application and loading conditions, there are various forms of analysis, including structural, vibrational, thermal, etc., which are carried out on the parts to be analysed. For comparison between different materials, material properties, namely density, elastic module, poison ratio and yield power, are considered.
The mechanical properties are given below for the different materials: -

| Sr.no | Material          | Density (Kg/m3) | Elastic modulus (GPa) | Poisson’s ratio | Yield strength (MPa) |
|-------|-------------------|-----------------|-----------------------|----------------|---------------------|
| 1     | Structural steel  | 7800            | 200e+3                | 0.3            | 210                 |
| 2     | Mild steel        | 7700            | 200                   | 0.3            | 247                 |
| 3     | Aluminum alloy    | 2625            | 72                    | 0.33           | 250                 |
| 4     | Stainless steel alloy | 7800 | 200         | 0.27           | 450                 |

4. Methodology
On the Diesel Exhaust Fluid tank keeping bracket, static structural, modal and PSD analysis was performed. For each form of analysis, the following steps were followed.

4.1. Static structural analysis
In order to verify the strength of the material to withstand the loading conditions acting on the Diesel Exhaust Fluid tank holding bracket, static structural analysis is performed. Static structural analysis of the Diesel Exhaust Fluid tank holding bracket is conducted using the following steps:

4.1.1. Calculating the boundary conditions for the bracket
1. Find out the static net load that operates on the bracket.
2. The weight of the various Diesel Exhaust Fluid tank assembly parts, such as the Diesel Exhaust Fluid tank, the dosing module and the air/oil separator, are considered here.
3. Consider the dynamic acceleration operating on the vehicle that is to be evaluated for. We consider a 10g acceleration that works on the car.

4.1.2. Pre-processing
1. Ansys 18.2 software has been used to perform the analysis.
2. Open Ansys 18.2 software.
3. Enter the prerequisite material data for all the four selected materials.
4. Open Design Modeler and import the geometry of the bracket with tank.
5. Find out the mid surfaces of the all the sections of the bracket.
6. Assign respective materials to different parts of the bracket.
7. Suppress all the nuts and bolts, this done to minimize the processing.
8. Give respective contacts to the bracket.
9. Open Mechanical in Ansys, apply a point mass to the geometry of the bracket which constitutes the static load on the bracket.
10. Meshed model was obtained by giving fine mesh settings.
11. Equivalent stress and total Diesel Exhaust Fluid tank deformation were calculated for different materials.

4.2. Modal analysis
To determine the natural frequency of the variable for the desired number of modes, modal analysis is performed. This makes us infer that the part will or will not experience resonance.

4.2.1. Following are the steps involved in the modal analysis of the bracket
1. Import the geometry of the bracket with tank in the Ansys 18.2 software.
2. The bracket is given fixed support at the clamping side of the bracket.
3. Modal solution is solved.
4. Modal analysis is performed on the bracket considering different materials.

5. Stress calculation
The bracket is attached through bolts on one side to the frame of the vehicle and the load is applied on the other side of it. It thus serves as a cantilever beam. The tension will impact the bolts that hold the bracket. Shear stress will also occur in the bolt.

5.1 Calculation of stresses on the bracket
Acceleration for which the bracket is to be designed is 10g

\[
\text{Total force acting on the bracket} = 47.5 \times 10 \times 9.81 = 4659.75 \text{N.}
\]

Since the clamping side is bolted, hence the bolts will experience shear stress due to loading.

\[
\text{Force on single bolt} = \frac{\text{total load}}{\text{No. of bolts}} = \frac{4659.75}{11} = 423.61 \text{ N}
\]

\[
\text{Stress on each bolt} = \frac{\text{force on bolt}}{\text{Cross-sectional Area of bolt}}
\]
= 188.76 MPa

6. Results and discussions
Analysing the results of the analysis: The frequency of the vehicle at various accelerations was measured in terms of road conditions. The overall vehicle acceleration was found to be between 20 Hz-30 Hz. Therefore, when evaluating the effects of the modal analysis, we verify that if indeed the bracket will go under resonance, the first natural frequency of the bracket should be more than 30 Hz.

Table 2. Comparison of Weight for same geometry design and variation in thickness structural steel and Aluminium

| Sr.no | Material     | Thickness of Frame (mm) | Weight of the Bracket (Kg) | Natural Frequency (Hz) |
|-------|--------------|-------------------------|----------------------------|------------------------|
| 1     | Structural Steel | 6                        | 25.069                     | 39.588                 |
|       |               | 5.5                     | 24.176                     | 38.353                 |
|       |               | 5                       | 21.83                      | 37                     |
|       |               | 6                        | 8.38                       | 34.492                 |
| 2     | Aluminium    | 5.5                     | 7.84                       | 33.285                 |
|       |               | 5                       | 6.38                       | 32.054                 |

Figure 1. Comparison of weight of bracket with thickness

We equate the equivalent stress obtained with the yield strength of the material when evaluating the effects of static structural analysis. We conclude that the bracket is structurally secure if the equivalent stress is less than the yield strength of the material. For the current bracket, structural and modal analysis was carried out and it was found that the natural frequency of the bracket was less than 30 Hz for the first time. Therefore, a structural adjustment was made to the current bracket design that would increase the stiffness of the bracket. Structural and Modal analysis is carried out on the optimized design of the bracket with different materials and thickness variations for the acceleration of 10g. And these results are compared to find the optimized design.

7. Conclusion
By adjusting the material and thickness of the frame, the various iterations are carried out on the optimised design of the bracket. By reducing the thickness of the frame by 1 mm and using structural steel material, we have thus obtained a weight reduction of 3 kg.

8. References
[1] Singhal A, Mandloi R, 2013 Failure Analysis of Automotive FWD Flexible Drive Shaft – a review, Inter Journal of Engg Research appl, 3
[2] Naghate S and Patil S, 2012 Modal Analysis of Engine Mounting Bracket Using FEA, Inter Journal of Engg Research and App, 2
[3] Heyes A M 1998 “Automotive component failures”, Eng Fail Anal.
[4] Heisler H 1999 “Vehicle and engine technology”, 2nd ed, London, SAE International
[5] Vogwell, J 1998 “Analysis of a vehicle wheel shaft failure”, Engineering Failure Analysis, 5
[6] ASM metals handbook 1996 “Fatigue and fracture”, Metals Park (OH), 19
[7] www.matweb.com
[8] www.dieselnet.com
[9] ASM metals handbook, 1996, “Fatigue and fracture”, Metals Park (OH), 19
[10] Bayrakceken, H, 2006 “Failure analysis of an automobile differential pinion shaft”, Engineering Failure Analysis
[11] Kim K and Choi I 2003 Design Optimization Analysis, SAE TECHNICAL, 01-1604
[12] Lee D, Jeon, Chang W, Mook C 2003 Optimal Shape Design of an Air-Conditioner Compressor Mounting Bracket in a Passenger Car, SAE TECHNICAL, PAPER SERIES
[13] Yong X and Zonni D, Tuopu Group, Zhong G, Zhao Y and Jiang C 2007 Structural Optimization for Engine Mount Bracket, SAE TECHNICAL, PAPER SERIES