International Journal of
Advanced Technology and Innovative Research
www.ijatir.org
ISSN 2348–2370
Vol.09,Issue.09,
August-2017,
Pages:1439-1443

Thermal Analysis of Automobile Oil Sump

VALMETI SUDHEER[1], B. SRIDHAR BABU[2]
[1]Professor, CMR Engineering College, Hyderabad, TS, India.
[2]Associate Professor, CMR Engineering College, Hyderabad, TS, India.

Abstract: The aim of the project is to model oil pan, designing a casting tool and generate CNC program for the same and reduction of weight will be done at unnecessary areas. Initially data will be collected to design mold tool and for the conditions of analysis. In next stage a model will be generated using PRO-E 5.0 for further study. The performance of an IC engine has an ionic bond with the performance of lubricating oil system. Oil sump and Oil Suction Pipe is one such assembly. The oil sump acts as partially filled reservoir of lubricating oil. The oil pump sucks the oil through the oil strainer which is immersed inside the oil sump. Design of oil sump involves designing for capacity, layout, and engine tilt angles[3]. Due to these forces, the oil is pushed towards oil sump sides depending on the resultant of the above forces. This resulting oil movement may have serious implications on the engine performance, if the oil strainer is not immersed in oil and the pump sucks air into the lubricating oil system. The resultant forces acting on the lubricating oil system are difficult to calculate using classical design approach since it involves sloshing type fluid structure interactions. Since the problem is highly non linear and considering the importance of this design aspect, the verification has been attempted using Fluid-structure interaction capability.

Keywords: Oil Sump, Tilting, Fluid Structure Interaction.

I. INTRODUCTION

In an internal combustion engine, the oil sump acts primarily as a container which stores lubricating oil and supplies it to the components via suction system immersed inside it. A general layout of sump and suction – strainer system in a heavy duty diesel engine is shown in Figure. The amount of oil (capacity) in the sump is arrived from empirical relations involving the total oil flow rate required for rotating parts in the engine and the number of times the oil needs to be re circulated through the system. In general heavy duty diesel engines require large sump capacity in order to cater to extended oil change interval periods. In order to have such high capacities, sumps have deep well design. The positioning of this deep well (Front well / mid well/ rear well) depend son the vehicle layout. Due to standard vehicle configuration (Layout of anti-roll bars, front cross members, front axles, engine positioning, etc) mostly buses require rear well sump and trucks require front well sumps. During engine operation, oil pump draws oil from the pan and circulate it through the engine, after the oil has passed through the engine, it is allowed to return to the oil pan. It's a spot for the oil to collect if the engine isn't running.

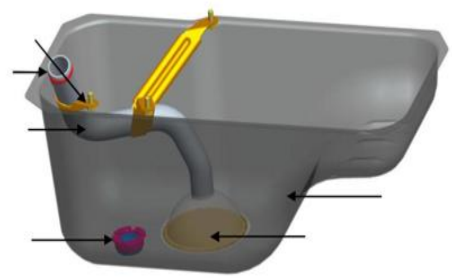


Fig 1. Oil sump model.

It also is where the oil cools as air passes over the surface of the pan. It's a place for impurities in the oil to settle and it has a drain to allow for removal of old oil. Use of a sump requires the engine to be mounted slightly higher to make space for it. Often though, oil in the sump can surge during hard cornering starving the oil pump. For these reasons racing and piston aircraft engines are "dry sumped" using scavenge pumps and a swirl tank to separate oil from air which is also sucked up by the pumps. Oil pans are detachable mechanisms made out of thin steel and bolted to the bottom of the crankcase. To maximize its function, it is molded into a deeper section and mounted at the bottom of the crankcase to serve as an oil reservoir[2]. When an oil pan is removed, some components revealed usually include the crankshaft, oil pickup, and the bottom end of the dipstick. Some oil pans will also contain one or more magnets that are designed to capture small pieces of metal before they can plug the oil filter or damage the engine.

II. SYSTEM DESCRIPTION

The system analyzed was a rear well oil sump for the new design of heavy duty 4 cylinder diesel engine developed in Ashok Leyland. The cross section of the engine is shown in Figure to understand the layout of sump and suction-strainer system.

The details of the system analyzed are given below:

1. Engine: 4 cylinder common rail engine
2. 2. Engine oil : SAE 15W40
3. 3. Oil sump capacity : 16L

4. 4. Max oil level : 18L
5. 5. Min oil level : 12L
6. 6. Installation angle :1.5°/3°- longitudinal, flywheel down.

Other loading conditions for a heavy duty truck as per internal company standards (ALS.010.05, 02.05.1999) are given below:

1. Max Deceleration (braking) : 0.6g
2. Cornering (acceleration) : 0.3g

where “g” is acceleration due to gravity 9.81 m/s2.

The tilt angle targets for the system under study are discussed in detail in the following section.

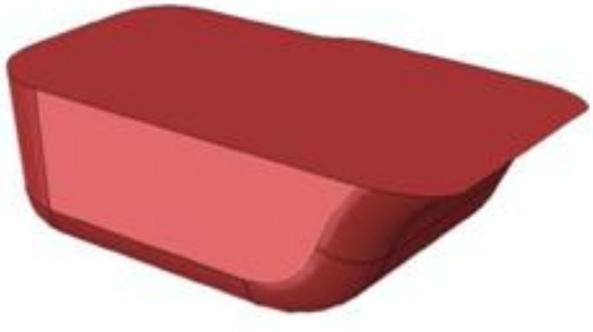
A. Design of Oil Pan:

Oil pan model was designed by using 3D modeling software (PRO-E 5.0). Oil pan has been designed for operating pressures loading and vibrations. The 3D model of actual oil pan is shown

Fig 2. Solid model of Oil pan

III. Load and boundary condition

For problems, initial condition has to be defined to specify the initial volume occupied by different fluids in the 3D space defined for the CAD Model. To ease the effort of analyst to create such complex model, Ansys has come up with an excellent tool, called “Volume Fraction tool”. A pre-requisite to use this is that the PRO-E(3D model) has to be hex meshed. To use this both the simple cuboid model and the complex initial fluid state model (in this case the solid oil model as described in section model description) has to be selected as shown



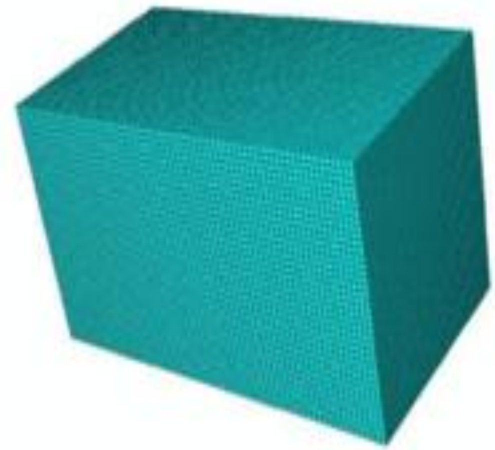


Fig 3. Oil Pan Meshing.

Sump has been meshed with second order tetrahedral elements, C3D10M. The cuboid model has been meshed with EC3D8R elements.

IV. FEA MODEL OF OIL PAN

The Oil pan model was meshed with shell 63 element type. A total number of 24029 elements and 24223 nodes were created. The meshed model is shown in the below figure. Static analysis is used to determine the displacements, stresses, strains and forces in structures or components caused by loads that do not induce significant inertia and damping effects. The Objective of this analysis is to check the High stressed locations and deflections on the Oil pan for the applied loads. Modal analysis is used to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. Modal analysis was carried out on the prestressed oil pan to determine the natural frequencies and mode shapes of a structure in the frequency range of 0 -1200 Hz.

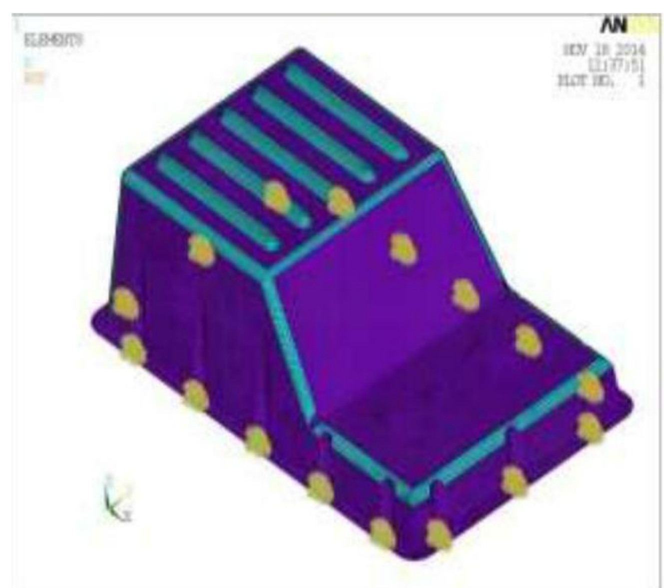


Fig 4. Boundary conditions for static analysis.

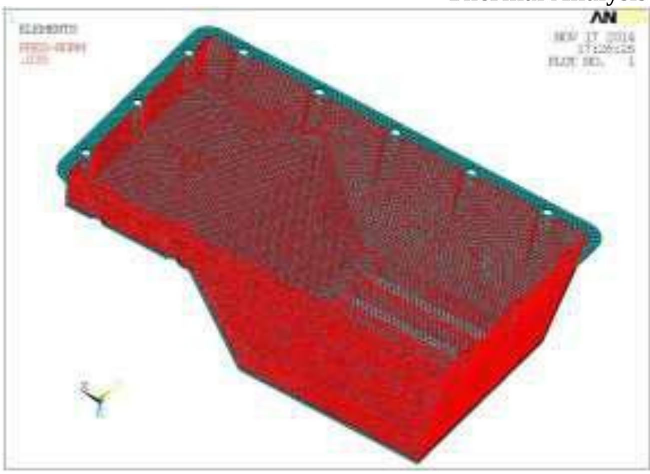


Fig 5. Applied pressure Loads for static analysis

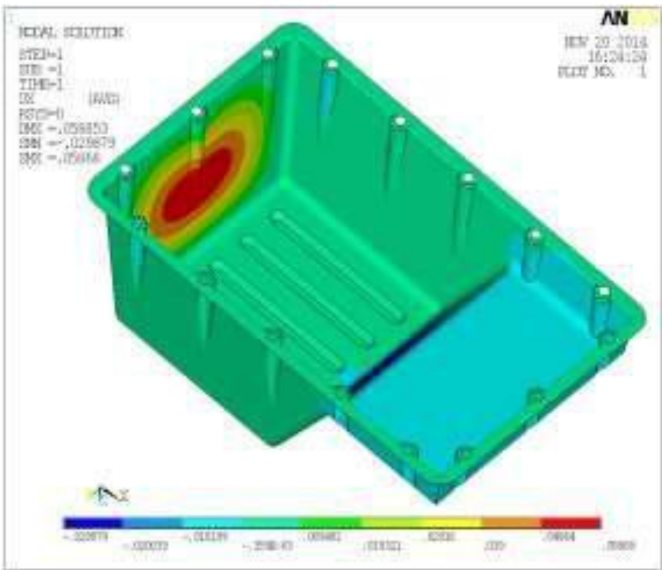


Fig 6. Deflection in X-dir for static analysis.

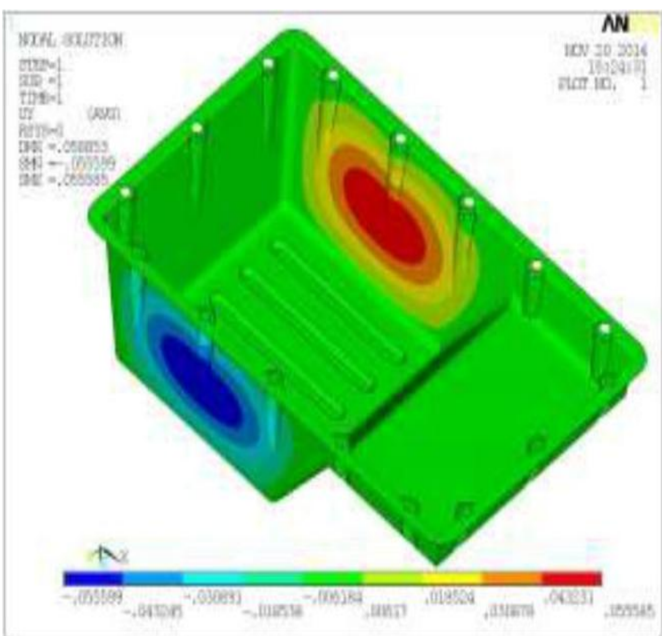


Fig 7. Deflection in Y-dir for static analysis.

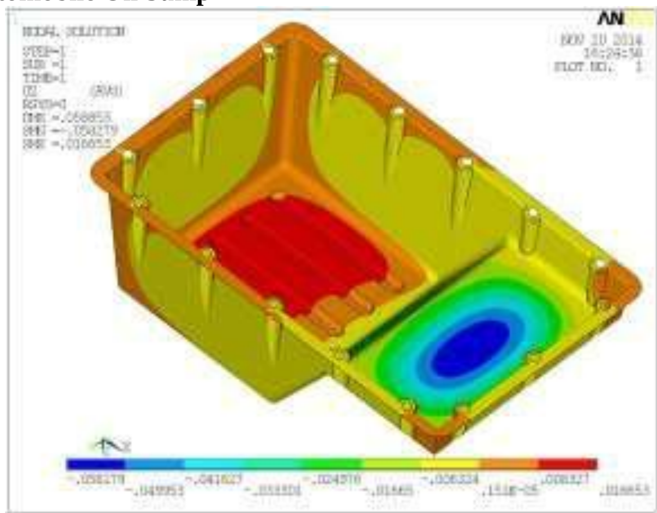


Fig 8. Deflection in Z-dir for static analysis.

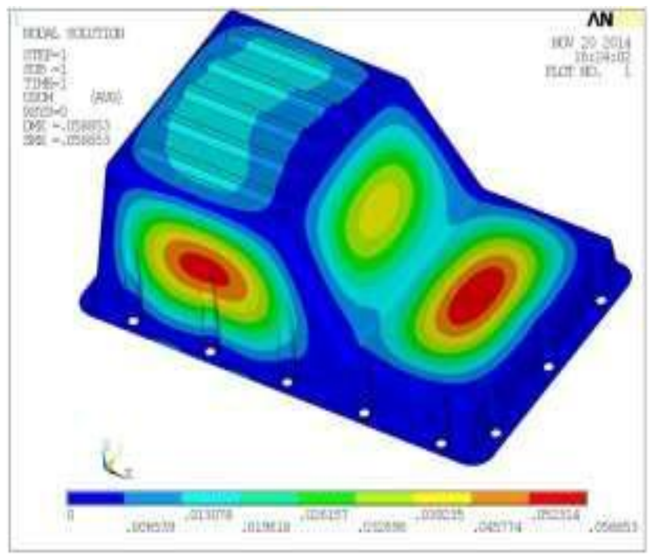


Fig 9. Total Deflection for static analysis.

From the analysis results the maximum deflection of 0.06mm is observed on the oil pan in pre stress condition. This deformed model will be used to do the modal analysis to obtain the natural frequencies and mode shapes on the prestressed oil pan.

V. VonMises Stresses

The Maximum Von Mises stress of oil pan observed 56MPa in pre stress analysis. From the analysis, the maximum VonMises [5] stress of 56 Mpa is observed on the oil pan. The maximum stress is observed on the bolting locations of the oil pan. The yield strength of the material is 418 Mpa.

VI. SUMMARY OF STATIC ANALYSIS

From the analysis results the maximum deflection of 0.06mm and VonMises stress of 56 Mpa is observed on the oil pan in pre stress condition. This prestressed oil pan will be used to do the modal analysis to obtain the natural frequencies and mode shapes on the prestressed oil pan.

VII. MODAL ANALYSIS

Modal analysis was carried out on Oil pan to determine the natural frequencies and mode shapes of a structure in the frequency range of 0 -1400 Hz. From the modal analysis, a total of 4 natural frequencies are observed in the frequency range of 0-1400 Hz[4]. The total weight of the Oil pan considered for the analysis is 6.0kg. The mode numbers and the corresponding frequency values are shown in the below table. The mode shapes for all the frequencies are plotted below.

Table.1 shows the Frequencies in the range of 0-1400Hz.

MODE. NO	FREQUENCY (Hz)
1	859
2	1054
3	1294
4	1337

The mode shapes for the above frequencies are plotted below:

Results –Mode1 @859 Hz

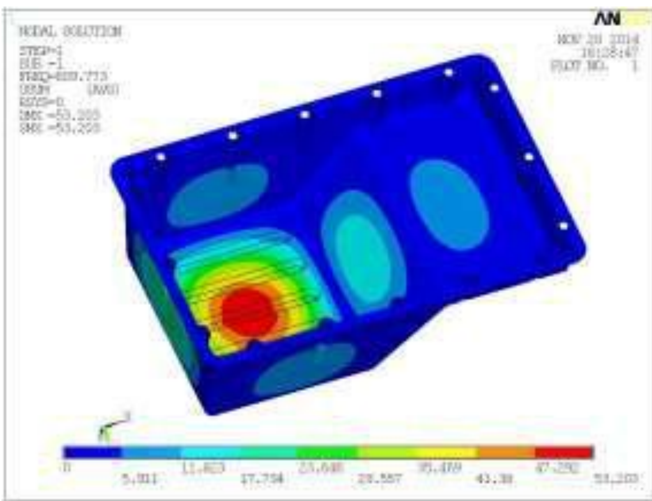


Fig 10. Results –Mode2 @ 1054 Hz.

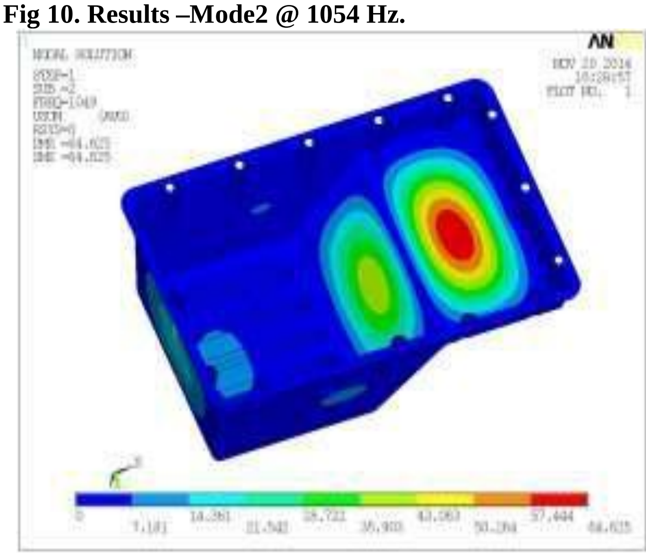


Fig 11. Mode3 @1294Hz.

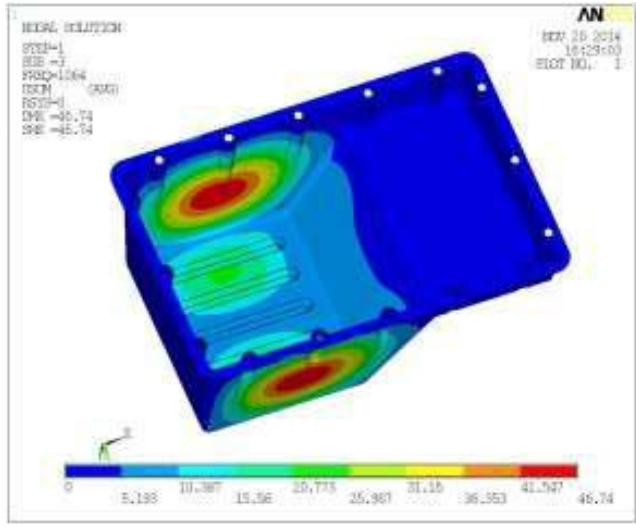


Fig 12. Results –Mode4 @ 1137 Hz.

From the above modal analysis results of the prestressed oil pan, following observations are made: Four natural frequencies exist in the range of 0-1400 Hz. Deflections are observed on the bottom side for a frequency of 859 Hz. Deflections are observed on the front bottom side for a frequency of 1054 Hz.Deflections are observed on the side walls for a frequency of 1294 Hz and 1337 Hz. To check the magnitude values of deflections and stresses at the above mentioned frequencies due to the operating loads, harmonic analysis is carried out on the prestressed oil pan From the above harmonic analysis, the stresses and deflections at the nearest natural frequencies are recorded and tabulated below.

Table.2

S.no	FREQUENCY(Hz)	DEFLECTIONS (mm)	VON MISES STRESS (MPa)
1	900	0.54	129
2	1100	0.59	169
3	1200	0.99	386

From the above results it is observed that the stresses at the nearest natural frequencies 900Hz, 1100Hz, and 1200Hz are 129MPa, 169MPa and 386MPa respectively. The yield strength of the material used for modified oil pan is 406MPa. According to the VonMises Stress Theory, the VonMises stress of modified oil pan at frequencies 900Hz, 1100Hz, and 1200Hz are having stresses less than the yield strength of the material. Hence the design oil pan is safe for the above operating loading conditions.

VIII. CONCLUSION

In this project a truck oil pan was designed and analyzed for vibration reduction. Finite element analysis was done to model the structural behavior of oil pan. Both static and dynamic loads were considered for the analysis. In this project the effects of pre-stress forces on modal parameters of oil pan and harmonic response analysis of pre-stressed oil pan using ANSYS has been performed. First pre-stress modal analysis was performed using sump. Later design changes were implemented to increase the stiffness of the oil pan. From the FE simulation results of the modified oil pan, it is concluded that the modified oil pan is safe for the mentioned operating loads

IX. REFERENCES

[1] Vibration-based damage assessment for controller reconfiguration application to an oilpan by bart peeters, stoyan kanev,

[2] Optimization of an oilpan (robust design optimization) by andreas veiz, johannes will,

[3] Design and development of sheet metal 'draw' component using cae technology by y.n. dhulugade, p. n. gore.

[4] Design of active noise and vibration control for car oil pans using numerical simulations by s. ringwelski, t. luft, u. gabbert.

[5]Pre-stressed modal analysis of a turbine spacer by m. attia, j. hou.