IJISET - International Journal of Innovative Science, Engineering & Technology, Vol. 2 Issue 7, July 2015.
www.ijiset.com
ISSN 2348 – 7968

Heat Load and Air Circulation Analysis of Diesel Engine Compartment

Sulekha Meher[1], Sudhakar Umale[2]

[1, 2] Department of Mechanical Engineering, Sardar Patel college of Engineering,
Mumbai, Mahrashtra, India

Abstract

The temperature distribution in the engine compartment is depending on air ventilation through compartment. It is very serious job to keep some constraints temperature limit for special purpose vehicle. A Computational Fluid Dynamics (CFD) model has been developed that allows for simulation of the parameters of velocity, temperature and for air circulation in an engine compartment with a reciprocating turbocharge diesel engine with power 110 kW. To reduce air recirculation from radiator outlet, air guider is incorporated between radiator and compartment wall. Hot air recirculation is minimized to greater extend by using air guider (radiator canopy) behind the radiator. With radiator guider temperature of air entering in the radiator is reduced by 14 %. The temperature distribution in the compartment by adding air guider is reduced by 3 to 10 % in some regions compares to without air guider. Predictions from this model have been compared with the experimental data and observed that actual temperature at the outlet is 3 to 12 % higher than CFDesign results.

Keywords: *Air circulation, louvers, air guider, hotspot, engine compartment and CFD*
.

1. Introduction

With the Recent trends, automotive market concentrates on high-performance engines and climate-control systems. However, automobile design must respect geometrical restrictions related to style constraints. More and more components are employed in the underhood region. The underhood compartment is thus becoming more and more cramped, causing complex airflows and difficult air paths that give raise to complex aerothermal phenomena [1]. If thermal equilibrium is not maintained, the temperature of the engine is found to be high or low. Thus it leads to drop in engine power, economy and reliability [2], [3], [4].

Engine compartment is an area where all engine assemblies are installed. Design of engine compartment is mainly depends on application of main system design. Large amount of heat is generated from several subsystems. The first heat source is from the internal combustion engine [1], [7].

Thus it is very important to design the system which can operate on constraints limits. It is crucial task to analyze the system on actual working conditions. It is disclosed from the recent research that two methods used to analyze engine compartment i. e. experimental method and numerical method. In experimental method, different values of temperature, velocity and flow rates at different locations are obtained with the help of experiments perform on the actual model or simplified experimental setup with same working conditions. Experimental methods like Particle Image Velocimetry (PIV), Laser Doppler Velocimetry (LDV) and thermocouples are used for velocity and temperature measurements. Visualization of air flow pattern was checked by smoke wire method [1]-[7].

With the rapid development of the technology of computing power and numerical calculations, computational fluid dynamics (CFD) simulation analysis as a design research tools, and applications in the research of the engineering vehicles.[3],[4],[6].

2. Physical model

The model has been prepared with the help of Solid Edge ST6 software in the plant. Figure 1 shows the engine compartment divided into three sub-compartments which are separated by solid walls. Middle compartment is occupied with all power generating unit like engine body, gear box, alternator for battery changing, pumps, radiator and fan assembly. The radiator fan is used to enhance the forced convection heat transfer in the middle compartment. Radiator and engine (heat source) are placed in the middle sub compartment therefore it is decided to focus on middle sub compartment. It is necessary to maintain appropriate air recirculation and pressure. The air guider / radiator canopy is designed to obtain necessary conditions.

IJISET - International Journal of Innovative Science, Engineering & Technology, Vol. 2 Issue 7, July 2015.
www.ijiset.com
ISSN 2348 – 7968

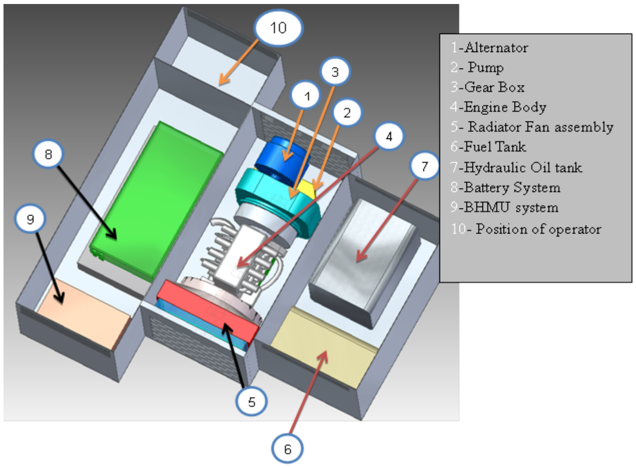

Figure 1 Engine compartment with air guider and all sub compartments

The middle sub compartment model as shown in figure 2 is used to simulate on CFDesign. This would help us to minimize mesh grid numbers which reduces convergence and simulation time. The engine compartment is modeled with the help of Solid Edge St6. The resultant model has dimensions of 1710×790× 850 mm.

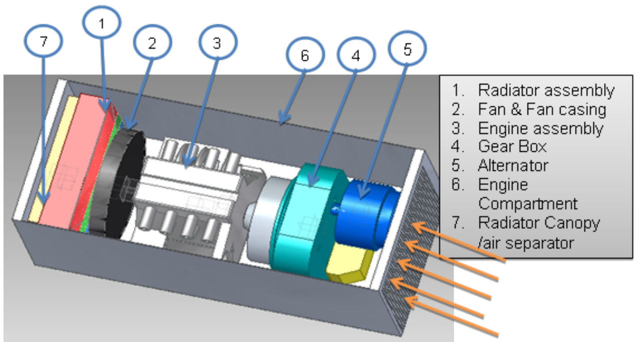

Figure 2 CAD model of engine compartment with louvers and air guider

3. Mathematical model

The incompressible, steady-state, turbulent three dimensional flow is considered for analysis. k-ε turbulent model is used for solving this problem.

3.1 Nomenclature

Cp	Constant pressure specific heat J/Kg.K
gx, gy, gz	Gravitational acceleration in x, y, z directions m/s^2
h	Enthalpy J/Kg.K
k	Thermal conductivity W/m.K
p	Pressure N/m^2
q_v	Volumetric heat source W/m^3
T	Temperature K
t	Time sec
u	Velocity component in x-direction m/s
v	Velocity component in y-direction m/s
w	Velocity component in z-direction m/s
μ	Viscosity N.s/m^2
ρ	Density kg/m^3

3.2 The governing equations

The Navier-Stokes or momentum equations and the First Law of Thermodynamics or energy equation are used while solving with help numerical method [9]. The steady state, incompressible, and turbulent phenomenon are considered while solving systems equations. The governing equations can be written as [9]:

1. Continuity Equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z} \quad (1)$$

2. X-Momentum Equation:

$$\rho\frac{\partial u}{\partial t}+\rho u\frac{\partial u}{\partial x}+\rho u\frac{\partial v}{\partial y}+\rho u\frac{\partial w}{\partial z}=\rho g_x-\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left[2\mu\frac{\partial u}{\partial x}\right]+\frac{\partial}{\partial y}\left[\mu\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)\right]+\frac{\partial}{\partial z}\left[\mu\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)\right]+S_\omega+S_{DR} \quad (2)$$

3. Y-Momentum Equation:

$$\rho\frac{\partial v}{\partial t}+\rho u\frac{\partial v}{\partial x}+\rho u\frac{\partial v}{\partial y}+\rho u\frac{\partial v}{\partial z}=\rho g_y-\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)\right]+\frac{\partial}{\partial y}\left[2\mu\frac{\partial v}{\partial y}\right]+\frac{\partial}{\partial z}\left[\mu\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)\right]+S_\omega+S_{DR} \quad (3)$$

4. Z-Momentum Equation:

$$\rho\frac{\partial w}{\partial t}+\rho u\frac{\partial w}{\partial x}+\rho u\frac{\partial w}{\partial y}+\rho u\frac{\partial w}{\partial z}=\rho g_z-\frac{\partial p}{\partial z}+\frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)\right]+\frac{\partial}{\partial y}\left[\mu\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)\right]+\frac{\partial}{\partial z}\left[2\mu\frac{\partial w}{\partial z}\right]+S_\omega+S_{DR} \quad (4)$$

5. The energy equation is written in terms of static temperature:

$$\rho C_p\frac{\partial T}{\partial t}+\rho C_p u\frac{\partial T}{\partial x}+\rho C_p v\frac{\partial T}{\partial y}+\rho C_p w\frac{\partial T}{\partial z}=\frac{\partial}{\partial x}\left[k\left(\frac{\partial T}{\partial x}\right)\right]+\frac{\partial}{\partial y}k\left[\frac{\partial T}{\partial y}\right]+\frac{\partial}{\partial z}\left[k\left(\frac{\partial T}{\partial z}\right)\right]+q_v \quad (5)$$

ISSN 2348 – 7968

3.3 Boundary Conditions

The boundary conditions are considered in the entrance velocity, and pressure at the exit. The flow rate at fan is assigned as 2.98 m3/s and 2700 rpm speed. The surface temperatures of engine block, alternator and pump are assigned as 115, 60 and 50°C respectively. The volumetric heat generation of radiator cores assigned as oil cooler, water cooler and charge air cooler 17, 47 and 24 KW/m3 respectively [8].

3.4 Meshing

Before analysis of CFdesign the geometry of engine compartment is broken up into small, manageable pieces called as elements. The meshing view of compartment is shown in figure 3. The corner of each element is a node, and it is at each node that a calculation is performed. These elements and nodes comprise the mesh. In three dimensional models, each element is a tetrahedral: a four sided, triangular-faced element [9]. The total numbers of cells for above model are 3,496,786.

4. Results and Discussion

Simulation analysis was made for engine compartment gun system. This paper is focused on air circulation and temperature distribution in the engine compartment. Then based on air flow distribution, the characteristic of air flow through the radiator module are researched. The temperature distribution in the compartment is analyzed. Figure 4 shows air velocity distribution of engine compartment.

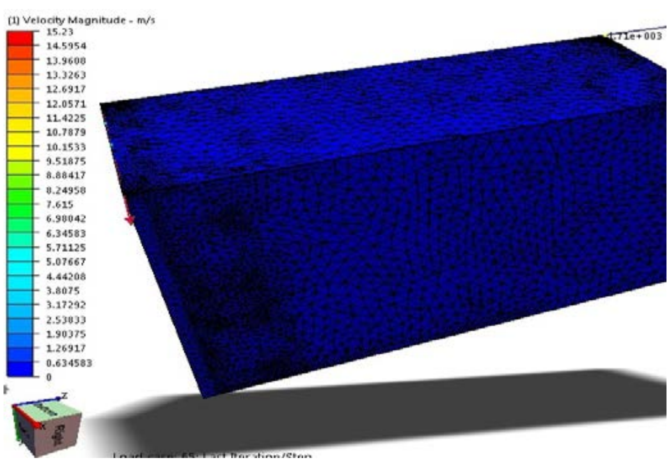

Figure 3. Meshing view of engine compartment

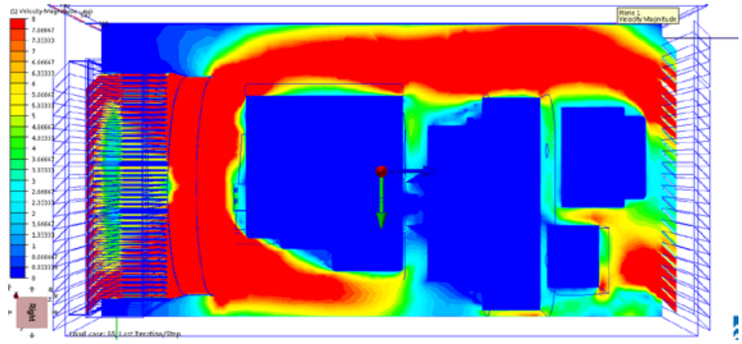

Figure 4 Velocity distribution in the compartment

It is observed that uniform air velocity at the inlet and outlet of the compartment. Air is entered into the engine compartment through inlet louvers and exit through outlet louvers. Maximum velocity is found as 10.373 m/s in the compartment.

It is seen from figure 5 that air comes from inlet louvers and enters in fan followed by radiator, air separator and exit louver. There is no air recirculation after the radiator. There are no eddies observed at top and bottom of the radiator.

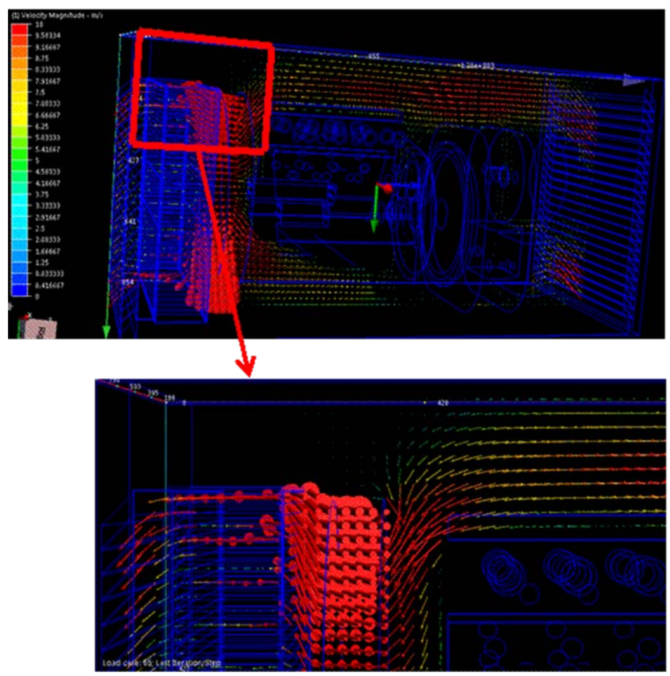

Figure 5 Velocity vector diagram

The engine compartment analysis is very difficult hence same compartment divided in three imaginary sections for simplifications as shown in figure 6. Temperature distribution graphs are plotted for these three sections.

ISSN 2348 – 7968

These regions and graphs are explained with help of figures 6 and 7.

Figure 7 shows the temperature distribution in engine compartment for three sections. From figure it is clear that temperature distribution in compartment is different at different locations. The high temperature zone is observed in middle section. When air comes in contact with engine its temperature is increased. Air temperature slightly increases in fan region.

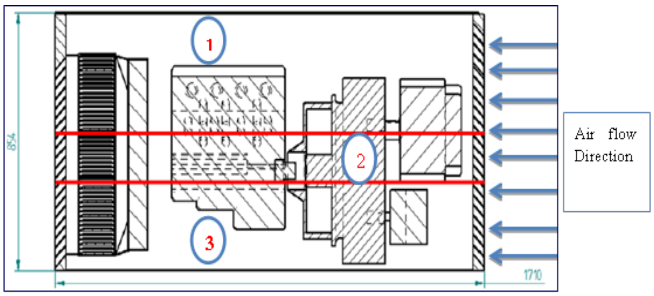

Figure 6 Imaginary sections of compartment

Air enters in the radiator at different temperature respect to sections of the compartment. It is observed that high temperature air enters in radiator through middle section.

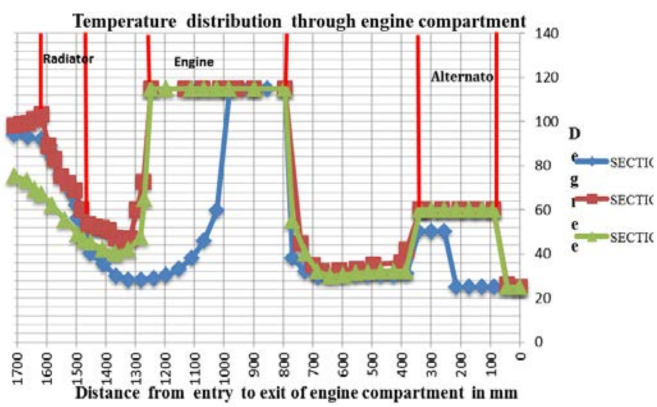

Figure 7 Temperature distribution for three sections

High temperature air comes out from middle section. In first and second section air comes in contact with alternator surfaces. In third section air comes in contact with pump surfaces. It is found that temperature of air entered in the radiator at 52.6, 57.3 and 48°C for top, middle and bottom sections respectively.

Figure 8 shows the temperature distribution of the air at the outlet louvers. Three temperature zones are observed in the figure 8. The upper louver is considered as zero marking while preparing figure 8. Temperature of the air increases as distance from the upper louver increases then it reduces to bottom louvers. Average temperatures for hydraulic oil cool region, charge air cooler and water cooler are 64.087 °C, 105.8507 °C and 95.47 °C respectively. It is observed from the graph, actual temperature is found 3 to 12 % higher than CFDesign results.

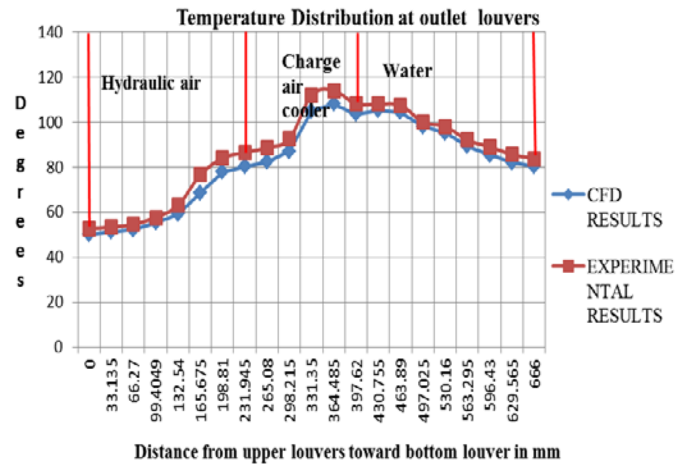

Figure 7 Temperature of air leaving the compartment

5. Conclusions

The following points were concluded:

1. With the improved design (air guider implementation) air recirculation gets reduced extensively.
2. For improved design temperature of air entering in the radiator is reduced by 14 %.
3. The temperature distribution for improved design was reduced by 3 to 10 % in some regions compared to without improved design.
4. The temperature of the air at the entry of radiator is different for different section of the compartment.
5. The actual temperature is 3 to 12 % higher than CFDesign results.

Acknowledgments

The authors would like to thank Mr. Onkar Gandrolla, Senior Engineer Mechanical Design Department, TATA Power Co. Ltd. Strategic Engineering Division for providing valuable guidance and support for this research work.

ISSN 2348 – 7968

References

[1] Mahmoud Khaled "Towards the control of car under hood thermal conditions", Applied Thermal Engineering, 31, 2011, 902-910.
[2] Mahmoud Khaled et al "Fan air flow analysis and heat transfer enhancement of vehicle under hood cooling system – Towards a new control approach for fuel consumption reduction. Applied Energy,91, 2012, 439–450.
[3] Wang et al "The Numerical Analysis of Engineering Vehicle Power Compartment Air Flow", Social and Behavioural Sciences, 96, 2013, 2480 – 2486.
[4] Hak Jun Kim et al "A numerical analysis for the cooling module related to automobile air conditioning system." Applied Thermal Engineering,28,2008,1896–1905
[5] S.C. Pang, M.A. Kalam et al "A review on air flow and coolant flow circuit in vehicles cooling system", International Journal of Heat and Mass Transfer, 55, 2012 6295–6306.
[6] J.M. Sala et al "Optimising ventilation-system design for a container-housed engine", Applied Energy, 83, 2006, 1125–1138.
[7] Thomas Binner "Aspects of Underhood Thermal Analyses" VTMS 6 SAE Conference, 2003.
[8] "Gun system development" TATA SED MANUAL 2014.
[9] H. K Versteeg and W Malalasekara "An introduction to computational fluids Dynamics, the finite volume method" Longman Scientific and Technical publication, England,1995.

First Author
Sulekha C. Meher
B.E in Mechanical Engineering, 2011; M.Tech in Thermal Engineering, 2015; M.Tech student Sardar Patel College of Engineering; first rank in thermal engineering department; interest in heat exchanger analysis.

Second Author
Sudhakar Umale
PhD in material science, 2015; Associate Professor, Sardar Patel College of Engineering, Mumbai.