International Journal of Advanced Technology in Engineering and Science
Vol. No.09, Issue No. 01, January 2021
www.ijates.com
ijates
ISSN 2348 - 7550

Thermal Heat Analysis of Engine Compartment for Ventilation: A Review

Mr. Suraj K. Chaukade[1]

Mechanical Engg. Deptt. GHRCE, Amravati

Surajchoukade33@gmail.com

ABSTRACT:

Now a Days, thermal protection is a vast issue in the development automobile sector. Due to compact engine compartments and highly increased in engine power demand, Heat loosed from engine cooling system, exhaust manifolds just alternator and the other components. Generated heat from packed engine compartment can be exhausted by providing ventilation. This paper discuss the problem occurred due to under hood air flow carries heat away at radiator after the air flows through numerous hood components, coolant flow circuit and air flow circuit meet each other which exchanges the heat at radiator. Also provide some solutions as differential survey and special ventilation arrangement for said problem.

Keywords:- *APU, RH, ECS, CFC, AFC, RH, Engine, Heat Transfer, Air Circulation, etc.*

INTRODUCTION:-

Thermal protection is progressively more in automobiles revolution, due to compact engine compartments and increased engine power demand. Basically Engine Heat exhausted from the ECS(Engine Cooling System), Exhaust Manifolds, the Alternator and the components those are in the Engine Cab. The under hood air flow carries the heat away from radiator and flows through the numerous hood components and CFC(Coolant Flow Circuit) and AFC(Air Flow Circuit) together exchanges the heat at radiator. Working environment of engine compartments are different and it depends on the application. That exhausted heat can be removed by the use of proper ventilation.

In this paper investigation and comparison of engine compartment heat for heat ventilation is done. Different environmental conditions present in Vehicles are like temperature, altitude, pressure and humidity are crucial factors and it affects the cooling systems. In India, Running Temperature for vehicle are from -20°C to 50°C and the RH values are from 60% to 94% while altitude low down in coastal areas to highs in Himalayan Areas. Due to this environmental conditions, It becomes more critical task to maintain some constrains limit of ACS for special purpose vehicle and simulate the system to operate in actual environmental conditions. At low temperature environment, there are chances of freezing of coolant and at high temperature environment leads to boiling of coolant which ultimately lowers the performance of cooling system. The pressure at high altitude is low and accordingly fan needs some extra power to overcome these pressure losses. For efficient working of Defence Vehicles in military bases, it becomes necessary to maintain proper air ventilation through engine compartment so there should not be occurred any hotspot in engine compartment, which damages the under hood components of the engine compartment. Normally vehicle cooling system is used to ensure that engines are maintained at its most efficient practical operating temperature.

The Engine Housings are used to hold up the complete APU, hydraulic drive system, batteries, hydraulic tank and other automotive subsystems. In Engine Housing tremendous heat is generated and housing should ensure proper air intake and exhaust. Engines of heavy duty vehicle need to expelled all this heat and throughout this energy to environment to avoid hot spot in engine compartment. Many more Engine hood compartments consists of number of equipments but here only heat dissipating equipments are considered for analysis and other equipments are considered as adiabatic conditions.

This work will helps to minimize the iteration timing and meshing up too. Heat dissipating equipments are Engine, Radiator, Alternator and Pump. But this work focused on a brief background of engine compartment and air circulation in engine compartment. The working environment in engine compartment is important to understand basic fundamentals of this subject.

LITERATURE SURVEY

To optimize various parameters of the energy conversion operations which involved in power generation by using thermal power plant, it is needed to review various researches involved in energy analysis and efficiency evaluation of power plants. Energy supply as per demand is narrowing down day by day and growing in demand of power made for power plant scientific interest. But most of the power plants designed by energetic performance criteria based on the first law of thermodynamics. The real energy losses in the power plants cannot be justified by the first law of thermodynamics because it is hard to differentiate between quality and quantity of energy based on the several activity and power plant experience. Some key observation has made and presented in this paper.

An experimental study of Aero- Thermal Phenomena in vehicle under hood compartment investigated by measuring temperature, convective heat flux and radiative heat flux. Measurements were carried out on passenger vehicles in wind tunnel S4 of Saint-Cyr-France. Under hood space was instrumented by 120 surface and air thermocouples and 20 flux meters. Measurements were performed for three thermal functioning conditions. In engine running period, front wheels were positioned on test facility with power- absorption-controlled rollers. In observation, constant-speed driving phase and for the different thermal functioning points, typical exponential trends for all component temperature variations, air zones and engine parameters were done. The exterior air that enters in the under hood compartment cooled convection heated up components. This was demonstrated by the cold box, a critical component from a thermal point of view since it contains computers. In new approach of design improvements had been made by designing stating and mobile deflectors in order to protect certain components from hot air circulation[1].

Optimization of hood cooling module done by controlling the cooling module positioning according to engine energy requirements. The velocity and temperature measurements are carried out on a simplified vehicle body designed based on real vehicle front block. Particle Image Velocimetry

(PIV), Laser Doppler Velocimetry (LDV) and thermocouples Methods were used for velocity and temperature measurements. Dynamic results obtained with the help of CFD which were validated with experimental results. Rotational speeds 1400 rpm and

2800 rpm of the fan were studied in this investigation. Experiments were performed on actual Peugeot 207 vehicle. For each speed, 8 configurations of the engine blockage (8 spaces between the cooling module and the engine block) were studied. 6, 8, 10, 12, 14, 16, 18 and

20 cm spaces were taken for the experiments. Thermal powers evacuated from radiator increased by 37%, when distance between engine block and cooling module was increased by the fan at low speed and at high speed, which increases power by 20% and Downstream temperature decreased by 9% which results in distance between engine block and cooling module increased. When engine block moved backward by 2cm from cooling module, it was observed that, thermal power evacuated by the radiator increased by 18% when the fan operates at low speed and 7% when it was operated at high speed. And decrease in the air temperature downstream of the radiator of about 5% at low speed and 2% at high speed. Whereas the temperature of air at the outlet of radiator was decreased by 5% at low speed and 2% at high speed. Distance between engine block and cooling module is important parameter while designing engine compartment[2].

Sometimes numerical modeling of engine cooling system under hood air flow, heat transfer at water jacket, heat transfer at radiator and coolant after- boiling phenomenon. There are two main sources of energy which contributes to cooling air which flows through under hood ram air and the radiator fan. For vehicle at high speeds, driving force for cooling air flow is ram air and at low speeds, driving force for cooling air is radiator fan. Ram air is flow driving force which resulted from favorable static pressure gradient between vehicle frontal opening inlets and underbody. Free stream air approaches to frontal opening, Air Velocity or Dynamic Pressure is reduces while static pressure increases, to maintain total pressure (Bernoulli Law). Static Pressure is highest at stagnant point before frontal opening (fluid velocity is zero). On the other hand, Air Acceleration results in Venturi Effect (lower cross sectional area, higher velocity) which creates low static pressure area at underbody. Meanwhile, electrical fan, acting as momentum source, incurs static pressure jump and total pressure jump for air which flows through the fan. Fans work on cooling air by giving air by both static and dynamic energy[3].

Containerized cogeneration sets (CCSs) are an efficient answer for remote developing regions which do not have alternative energy sources and for those applications requiring mobility and quick installation of energy plants. This is type of engine compartment. A Computational Fluid Dynamics (CFD) model has been developed such that it allows simulation of velocity parameters, temperature and pressure for calculating the heat flows in a CCS with reciprocating diesel engine with an alternator power of 903 kW. CFD model has been used to analyze possible alternatives for improving ventilation system. The presence of surfaces registering high surface-temperatures and no thermal insulation means within a container-housed set where is a great deal of heat to be dissipated[4].

Here cooling modules of an automobile had been analyzed numerically. Simulation data validated with the help of experimental data. Simulation had been made by checking front end air pattern. Test and Simulation were done in accordance with changing layout of fan, radiator and fan power. In experimental setup of fifteen vane anemometers were mounted to frontal face of core of radiator. Thermocouples were used to measures inlet and outlet temperature of coolant. Here estimate of exact heat rejection of radiator was calculated. Two radiators with different in fin density were used to analyzed engine cooling modules. In observation, performance of fan is generally not affected by shroud type on airflow stand. Non-uniform Airflow due to the shroud type makes the performance of heat exchanger worse[5].

The boundary conditions are that in entrance is velocity inlet, and pressure outlet at exit. Remaining four surfaces of virtual wind tunnel is set with wall and wall is a standard wall, no-slip boundary wall. Under corresponding working conditions, with principles of relative velocity, entrance conditions of virtual wind tunnel using different uniform air flow velocity as speed of inlet boundary. There is air reflow phenomenon between top of the power compartment and top of the radiator module. Due to presence of air reflow, resulting in part of hot air flowing through radiator module to re-flowing into radiator modules. This part of the recirculation of hot air causes the radiator heat exchanger performance degradation. The design process should improves design of reduce reflux[6].

Here research simulate and analyzed flow characteristics in engine compartment of a light aircraft with the help of FLUENT. Simulation of light air craft compartment had done with help of FLUENT. Air inlet, duct, guide vane and air outlets are designed to improve flow conditions according to drawbacks of prototype model. Improved model is added air inlets at high pressure region in front of the cowling which is expected to increase the cooling airflow to the engine cylinder. Next improvement had done by adding air ducts at high pressure region in front of cowling. Which results air velocity leads directly to engine through recirculation zones and causes rise in air velocity around engine cylinder. The enhanced model designed to guide vanes at proper positions of cowling to lead cooling airflow to second half of engine cylinder[7].

Here concept of designing and prototyping of off-highway heavy-duty trucks are simulated with CFD. Parameters and components like speed of cooling fans, core size of heat exchangers, temperature, pressure drop, thickness of insulation and design options of heat shield are considered. Four cases had chosen to investigate complete airflow field and solid surface temperature. All cases are considered for investigation as Cold flow without the cooling fan, thermally coupled flow with installation of cooling fan at one speed and thermally coupled flow with installation of cooling fan at a different speed. First task is to determine most critical components that influence airflow rate. It was observed that radiator is most critical components that affect pressure drop. As per Author, apply baffles and/or rubber seals between gaps of fan guard and heat exchanger are prevents hot air re-circulation in to the hood. In second case, performance of under-hood cold flow by checking fan curve and operating point compared to concept design. Performance of cold fluid checked with help of fan performance curve and observed that, small changes in pressure drop there was large air flow rate was reduced. Further Cases focused on flow-coupled heat transfer involved in engine, power train, hydraulic system, after-treatment equipments, and cooling package. CFD thermal analysis provides a comprehensive surface temperature distribution to validate and verify material thickness, surface property/insulation condition and heat shield locations by comparison to experimental data, which are obtained from thermocouples. As per observation, After-treatment equipments are at high risk due to extremely high temperature of exhaust gases. High temperature poses a risk to neighboring instruments and components. Maximum temperature was observed on clamps (non- insulated parts) and in neighboring region which is due to strong conduction through metal skin. For further improvisation, working condition insulation layer was assigned to minimize heat flow through conduction insulation was provided on maximum temperature region[8]. **Methodology:-**

To complete the entire project the following methodology will be selected.

- CAD Modeling of Four-wheeler engine section:

CAD model of complete engine section is to be done by using CATIA V5R19 software. The model prepared is must be cavity model to find the CFD analysis or flow behavior inside the engine section. The accurate dimensions are taken from

the reverse engineering process. TATA ManzaQuadrajet 90 car is considered for the reference of engine section.

- IGES Conversion of CAD Model: In this step developed CAD model is imported into .igs format (Neutral File Format called IGES format). Entire geometrical and dimensional information is carried out in other software through this type of formatted file. To perform CFD Analysis of Engine section, this type of file is used as a input object in CAE software.

- CFD Analysis of Engine Section: CFD Analysis of engine section is carried out with the help of ANSYS 2020 Workbench software where ANSYS Fluent is the environment where the actual analysis takes place. All boundary conditions are to be settled well. Like input velocity, pressure, temperature, hydraulic diameter etc.

- Result Generation from CFD Analysis:

At least for minimum 4 cases of different velocities will be checked to find the effect of engine cooling method implemented on the engine section. Considered engine is water and oil cooled. Also, the cooling fan is installed on the engine. Hence the combined effect of cooling is to be obtained to perform proper engine section analysis.

- Conclusion on the basis of CFD results:

By studying all the generated results through CFD Analysis conclusion will be drawn. Better results should be obtained in higher velocity as it carries the maximum heat generated through the engine.

Advantages:-

- Cooling technique efficiency can be checked and scope of improvement of cooling.
- Implementation of CFD Analysis will reduce the cost of experimentation.
- Project leads the entire engine section which is not considered in most of the cases.
- Study of CFD Analysis will provide the extra dimension to the research work.

DISADVANTAGES:-

- Preparation of CAD model for entire engine section is very critical task.
- Generation of CFD results and its validation needs more focus on input parameters.
- Simulation of exact flow condition is not possible because of human error and software limitation.

CONCLUSION & FUTURE SCOPE:-

This work generate reports in development of novel technique for purpose of understanding the airflow behavior and temperature distribution in the engine compartment. This analytical method will explain the analysis of temperature and air circulation of engine compartment. Here air temperature and air circulation will measured with the help of CFD design software. Different equations are used in numerical modelling according to working to get unknown parameters like velocity, temperature, heat transfer coefficient. In future this work will help in Thermal Heat Analysis of Engine Compartment for Ventilation

REFRENCES:-

[1] Mahmoud Khaled "Towards The Control Of Car Under Hood Thermal Conditions", Applied Thermal Engineering 31 (2011) 902-910

[2] Mahmoud Khaledet 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] S.C. Pang, M.A. Kalam "A Review On Air flow And Coolant flow Circuit In Vehicles' Cooling System", International Journal of Heat and Mass Transfer 55 (2012) 6295–6306

[4] Marc A J.M. Sala "Optimizing Ventilation-System Design For A Container-Housed Engine", Applied Energy 83 (2006) 1125–1138

[5] 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

[6] Wang "The Numerical Analysis Of Engineering Vehicle Power Compartment Air Flow", Social and Behavioral Sciences 96 (2013) 2480 – 2486.

[7] Hsujeng Liu et al "A Study On The Engine Compartment Airflow Of A Light Aircraft Using Computational Fluid Dynamics" (2012) J1013ICETI.

[8] Shaolin Mao "Off-Highway Heavy-Duty Truck Under-Hood Thermal Analysis", Applied Thermal Engineering 30 (2010) 1726 -1733.

[9] V. A. Romanov, N. A. Khozeniuk "Experience Of The Diesel Engine Cooling System Simulation" Procedio Engineering ISO(2016) Page 490-496.

[10] Dinesh Kumar Soni, Rajesh Gupta "Comparison Of Performance And Emission Characteristic Of Diesel & Diesel Wates Bland Under Varying Injection Timings" science and technology 2015,page 49-59.

[11] Dr. Abdul Siddique, Shaikh Abdul Azeez, Raffi Mohd., "Simulation And CFD Analysis Of Various Combustion Chamber Geometry Of C.I. Engine Using CFX" Volume 5, Issue 8,2016,Page

33-39. [12] K. Abay, U. Colak, L. Yoksek "Computational Fluid Dynamics Analysis Of Flow & Combustion Of A Diesel Engine" Thermal Engineering Vol. 4, No- 2, Issue 7, 2018, Page 1878-1895.

[13] K.L.Shrinivasulu, D.Srikanth, K.Rafi, B. Ramanjaneeyulu "Optimization Of Air Filter In An Automobile Diesel Engine By Using CFD Analysis" : Mechanical & Civil Engineering, Vol.13, Issue 2, 2016, Page 78-89.

[14] Tarek M. Belal, EI Sayed M. Marzouk, Mohsor M. Osman "Investigating Diesel Engine Performance & Emission Using CFD " Energy & Power Engineering, 2013, Page 171-180.

[15] A. M. Indvodia, N. J. Chotai, B. M. Ramoni "Investigation Of Different Combustion Chamber Geometry Of Diesel Engine Using CFD Modeling Of IN- Cylinder Flow For Improving The Performance Of Engine"

[16] S. K. Gugulothu, K. H. C. Reddy "CFD Simulation Of In-Cylinder Low on Different Piston Bowl Geometries in a DI Diesel Engine" Applied Fluid Mechanics, Vol. 9, No.3, 2016, Page 1147-1155.