ABSTRACT

Automotive engine cooling system has become a prime focus in the field of growing automotive engineering. The engine cooling system has a major impact on vehicle fuel efficiency and emission norms. In the development of automotive engine cooling system, the design and performance testing of radiator are carried out more often as isolated component without considering the impact of surrounding components. The kind of blockages in the upstream direction of radiator also will not be available at the early design stage of vehicle. These, design and performance tests are done with the assumption that airflow distribution is uniform and cooling air temperature is equivalent to ambient temperature. However, at vehicle level these assumptions are not valid due to the presence of blockages in the Cooling Airflow Path (CAP).

The blockages in the CAP upstream direction of radiator are categorized as non-hotspot and hotspot objects. The non-hotspot obstructions in the upstream direction of radiator like grille, bumper and crash beam affects the cooling airflow passing through them which in turn influences radiator thermal performance. The hotspot obstructions like condenser and Charge Air Cooler (CAC) which are located nearer to radiator increase the air inlet temperature magnitude. This leads to decrease in thermal performance of radiator which affects the engine cooling system performance and increases the air temperature of the underhood compartment. Thus, minimizing the obstructions in the upstream direction of radiator and reducing the heat exchanger thermal interaction will provide better thermal performance of radiator. Hence, the impact of blockage area during the design and performance test of radiator becomes a necessary consideration. This is required to ensure that heat exchanger thermal performance is not getting deteriorated due to the issues related to the surrounding components. In this study, a new methodology is proposed to predict the deterioration in thermal performance of radiator for practical range of blockage areas. Further, working principle of a new control unit is
presented to control the position of heat exchanger which is located in front of the radiator.

In this study a simulation model is developed to explore the impact of blockage on CAP. The CAP is analyzed with different type of blockage and blockage area at different air velocity in order to consider the impact of surrounding components on the air velocity distribution. The different type of blockages considered are horizontal, vertical, side to side and side to center. The Computational Fluid Dynamics (CFD) tool ANSYS FLUENT is used to investigate the airflow distribution over radiator. The airflow distribution is represented in terms of 'uniformity index' and 'non-uniformity'. Further, the simulation results are validated with the experimental results for a specific case 'without blockage condition'. In addition, the Analysis Of Mean (ANOM) study is also conducted to rank the factors influencing airflow distribution. The considered factors are blockage, fan speed and ram air speed. It was observed that blockage ranks at number one followed by fan speed and ram air speed. In the simulation study while comparing the airflow distribution among the different type of blockages, it is found that horizontal and vertical type blockage shows better performance compared to others.

Further the study highlights the influence of airflow distribution on the thermal performance of radiator. The thermal analysis is carried out using the existing empirical relation available in literature. This empirical relation evaluates the heat rejection rate under non-uniform airflow distribution, with the known value of heat rejection rate under uniform flow distribution. From this study it was observed that increasing the 'non-uniformity ' from 0 to 1 decreased the thermal performance of radiator by 30% with respect to uniform airflow distribution. However, till the non-uniformity of 0.5, the reduction in thermal performance is 10% only. The radiator thermal performance obtained under non-uniform airflow distribution is compared with uniform flow distribution in terms of 'deterioration ratio'. The obtained results are validated with experimental results.

Moving from simplified CAP level to vehicle level, CFD simulation and analytical study have been carried out to investigate the impact of Air
Conditioning (AC) condenser position on thermal performance of radiator. The availability of cooling air flow and Top Tank Temperature (TTT) are investigated by considering the two position for condenser. Further, a new approach for underhood thermal management is outlined with a control mechanism to position the condenser unit in the underhood environment.

The proposed new control unit decides the condenser position to be in a configuration 'one behind other' (obo) or 'one below other' (obelo). From the study it is recommended that control unit can be operated with 'obo' configuration at vehicle speed less than 30kmph and beyond that 'obelo' can be chosen. This control approach along with the Active Grille Shutter (AGS) system will provide better engine cooling system performance and fuel economy.

In conclusion this work analyses the impact of blockage area on performance of radiator and recommendations are suggested to lay out the CAP in the upstream direction of radiator with minimal blockages and reduced heat exchanger interaction effects.