AIRFLOW ANALYSIS OF ENGINE COOLING GRILLE USING COMPUTATIONAL FLUID DYNAMICS

[1]NIRAMOL HUTACHARERN, [2]ARTIT RIDLUAN

Research and Develop, Defence Technology Institute (Public Organization), Thailand
E-mail: [1]niramol.h@dti.or.th, [2]artit.r@dti.or.th

Abstract - Engine-bay-cooling system is an important part of the vehicle engine, maintaining the operating temperature. One of the factor determines heat-transfer efficiency of the system is the inflow air velocity which is controlled by the grille shape. A specific military vehicle designed for amphibious use intakes cooling air from the top of the vehicle via high performance exhaust fan. The grille and engine bay are modelled and analyzed by computational fluid dynamics to determine the suitable grille shape. The simulation has shown that vehicle speed has insignificant effect on the amount of air intake, and the grille blade setting at 45-degree gives the highest flow velocity into the engine bay.

Index Terms - Engine bay cooling system, cooling grille, CFD

I. INTRODUCTION

Cooling system is tremendously important for vehicles as it keeps the engine running at operating temperature.Air is considered as a function of cooling. Therefore, cooling system of most commercial vehicle located at the front bumperto directly intake air. The intake air is normally controlled by a grille which is carefully designed to direct the air into the engine chamber because the shape of the grille has much effect in the aerodynamic and cooling performance of the vehicle [1]. Unlike a commercial vehicle, military vehicle has its cooling air intake on top of its body to allow amphibious use as demonstrated in fig.1. The air is no longer entering the engine bay directly, and needed high performance fan to increase the amount of airflow into the engine bay. To enhance the cooling performance airflow through the engine bay must be managed [2]. As theamount of airflow through the engine bay varies with the inlet grille shape i.e. the settings of the grille blade, and the increase in velocity of the airflow through the engine has much impact on the overall heat transfer coefficient[3],the grille must be designed to allow the maximum air velocity into the engine bay.



Fig.1 Military Vehicle

II. THE MODEL

The engine chamber is as illustrated in fig.2. The cooling air enters the chamber through the inlet grille on top of the chamber, pass through the cooling pack, and exit through the high-performance-exhaust fans. The flow into the engine bay is driven by the velocity of the vehicle and two fans. However, as the simulation is aimed to study only the airflow through the grille, the cooling system is simplified. The grille slope with respect to the horizontal plan is 10 degrees. The grille dimension is shown in fig.3which affect to characteristic of airflow.

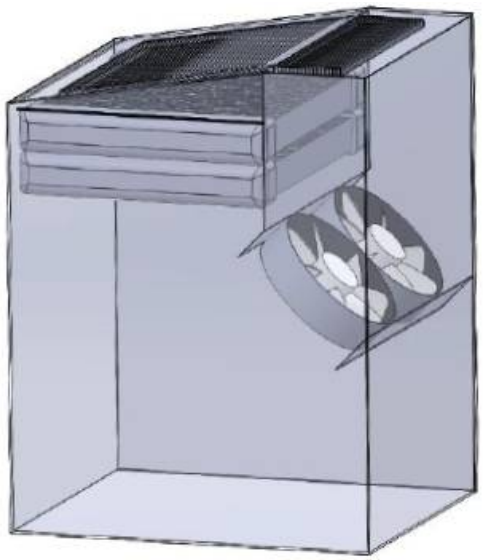


Fig. 2 Engine-bay-cooling chamber

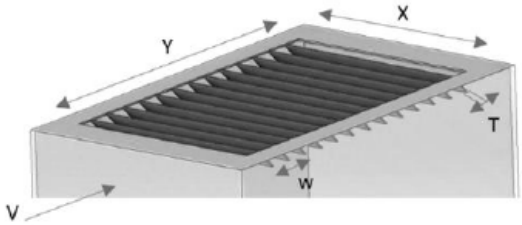


Fig. 3 Inlet grille-blade dimension

Where
T = The blade thickness
W = a distance between two blaedes
X = ventilator width

Y= ventilator length
V = airflow velocity

III. METHOD

The analysis uses the eddy motion that is described by the Large Eddy Simulation (LES)to stimulate the flow.

A. Large Eddy Simulation Model (LES)

Filtered Continuity Equation:

$$\frac{\partial}{\partial x_i}(\rho \tilde{u}_i) = 0$$

Momentum Equation:

$$\frac{\partial \tilde{u}_i}{\partial t} + \frac{\partial}{\partial x_j}\left(\tilde{u}_j \tilde{u}_j\right) = -\frac{1\partial \tilde{p}_i}{\rho \partial x_i} + v\left(\frac{\partial^2 \tilde{u}_i}{\partial x_i \partial x_j}\right) - \frac{\partial \tau_{ij}}{\partial x_i}$$

$\tau_{ij}$is called subgrid-scale (SGS)stress term

For LES simulation, the small scale is modeled. Therefore, the SGS is describing the small-scaled-turbulence mechanism the cascade and dissipate the energy from the large one .Here, the Wall-Adapting Local Eddy (WAL (Emodel is selected since this model provide good properties both near to and far from the wall and both for laminar and turbulent flows .The detail of LES and SGS is further studied.

B. CFD Analysis

The CFD analysis examine the airflow through the inlet grille at the driving velocity of 30 and 80 km/hr.At each velocity, the grille blade angle is set at 15, 30, 45, 60, 75 and 90 degrees with respect to the horizontal plane as shown in fig.4. The outlet airflow is 10 kg/s at fan location, and the computation domain is 3 times larger than the model.

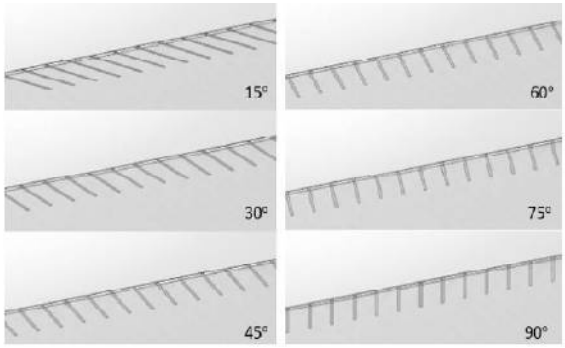


Fig. 4Grille blade angle at 15, 30, 45, 60, 75 and 90 degrees

C. Reynolds number

The Reynolds numberis a dimensionless value that measures the ratio of inertial forces to viscous forces and describes the degree of laminar or turbulent flow. Systems that operate at the same Reynolds number will have the same flow characteristics even if the fluid, speed and characteristic lengths vary.[4]

The Reynolds number is calculated from:

$$\mathrm{Re} = \frac{\rho v l}{\mu} = \frac{v l}{\nu}$$

Where

V = Velocity of the fluid
l = The characteritics length, the chord width of an airfoil ρ = The density of the fluid
μ = The dynamic viscosity of the fluid
ν = The kinematic viscosity of the fluid

IV. RESULTS

The total of twelve simulations have been done under the same condition for vehicle speed of 30 and 80 km/hr. The air velocity through the grille and turbulence intensity were recorded. Fig. 5 and 6 present the air velocity and turbulence intensity, respectively

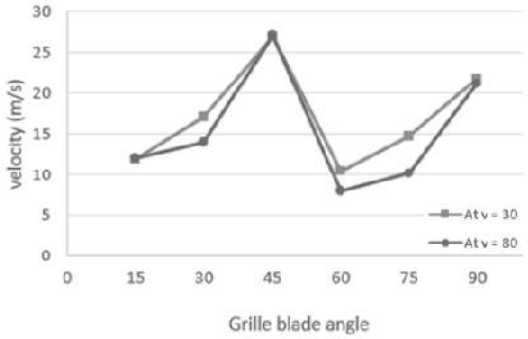


Fig 5 Air velocity at various grille blade angle

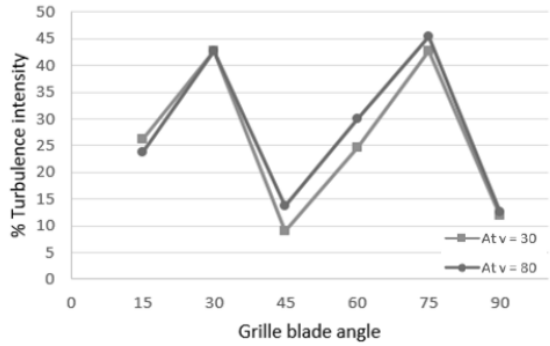


Fig 6 Turbulence intensity at various grille blade angle

The result from simulations at 30 and 80 km/hr share the same trend. This can be explained by noticing that Reynold's Number of both simulation is in the close range. Moreover, the air velocity depends on the fan performance much more than vehicle speed in cruise level. The setting angle of 45 degrees gives the highest air velocity and the least turbulence intensity.

Fig7illustrates the flow trajectory that the airflow smoothly into the engine bay. However, the left portion of the engine bay is rarely cooled by the intake air.

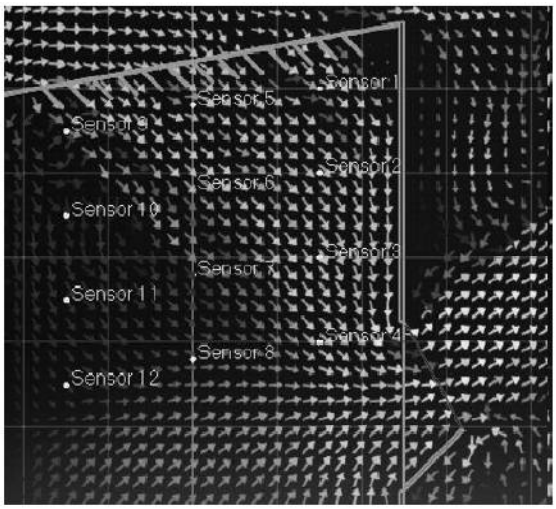


Fig.7 Flow trajectory – 45 degrees

On the other hand, the lowest air velocity occurs at 60 degrees even though the turbulence intensity is not highest amongst the simulations. The slow velocity occurs because there are two vortices interfering the incoming airflow as shown in fig.8

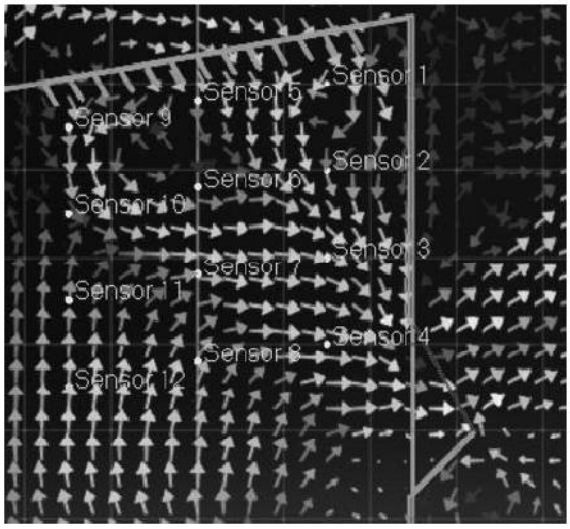


Fig. 8 Flow trajectory – 60 degrees

In addition, the 90 degrees simulation also gives comparable air velocity to 45 degrees simulation. It is only lower by 18%. However, the flow trajectory (figure 9) suggests the intake air has more effect on overall area of the engine bay. Further study is needed to clarify whether the overall heat transfer performance between 45 and 90 degrees grille-blade-setting angles is better.

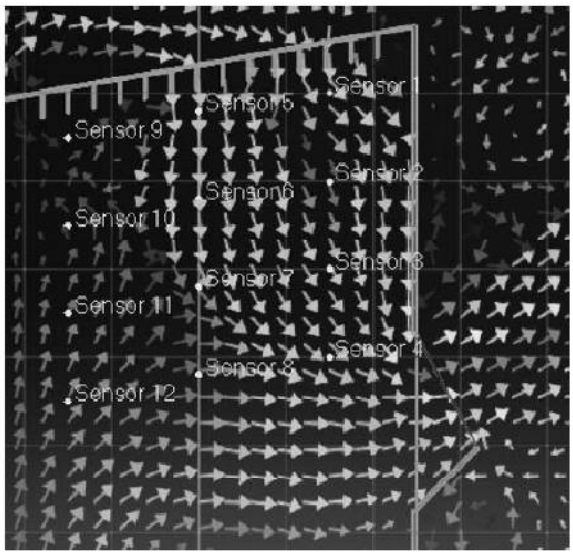


Figure 9 Flow trajectory – 90 degrees

CONCLUSION

The result has shown that grille blade setting angle has significant effect on the velocity of air flowing through the engine bay. The setting angle of 45 and 90 degrees are interested due to air velocities and flow trajectory. The 45 degrees simulation shows the highest velocity and the 90 degrees simulation is 18% less. However, the flow trajectory of 90 degrees case passes through larger volume of the engine bay. More studies on the heat transfer is needed to justify whether which angle should be used to manufacture.

REFERENCES

[1] S. Baskar and R. Rajaraman, "Airflow Management in Automotive Engine Cooling System," International Journal of Thermal Technologies, vol. 5, no. 1, 2015.

[2] M. Naraki, S. Peyghambarzadeh, S. Hashemabadi and Y. Vermahmoudi, "Parametric study of overall heat transfer coefficient of CuO/water nanofluids in a car radiator," International Journal of Thermal Sciences, vol. 66, pp. 82-90, 2013.

[3] L. Liu, C. Jing and X. Hu, "Numerical Simulation of the Influence of Intake Grille Shape on the Aerodynamic Performance of a Passenger Car," International Power, Electronics and Materials Engineering Conference (IPEMEC), 2015.

[4] R. M. C. John J.Bertim, Aerodynamics for Engineers, Pearson Prentice-Hall, 2009.

★★★