GRILLE DESIGN FOR PASSENGER CAR TO IMPROVE AERODYNAMIC AND COOLING PERFORMANCE USING CFD TECHNIQUE

J. M. KIM, K. M. KIM, S. J. HA and M. S. KIM*

Department of Aerospace and Mechanical Engineering, Korea Aerospace University, Gyeonggi 10540, Korea

(Received 20 May 2015; Revised 3 January 2016; Accepted 29 May 2016)

ABSTRACT—Grille opening shape for small passenger car is designed numerically by using parametric study. Key geometric parameters to design a grille opening configuration are represented by vertical height, horizontal width, size, linear deformation, position, and blockage. Numerical study investigates the effects of those key parameters on the aerodynamic drag and the grille inlet flow rate, which are very important to the aerodynamic performance as well as the powertrain cooling performance of the car. Flow simulations are performed at the velocity of 110 km/h inflow condition. The outflow boundary condition is implemented by pressure outlet condition of atmospheric pressure. Moving wall condition of 110 km/h is set on the ground.

KEY WORDS: Small passenger car, Grille opening, Numerical analysis, Aerodynamic drag, Cooling performance, CRFM

1. INTRODUCTION

The automotive manufacturing companies have devoted many efforts to improve the fuel efficiency because many countries have not only reinforced the emission regulations to reduce environmental air pollution due to toxic exhaust gas, but also coped with the depletion of fossil fuels (Ministry of Environment, 2014). Endeavor to improve the fuel efficiency starts up from the beginning of conceptual vehicle design.

Many experimental investigations have been performed to reduce the vehicle aerodynamic drag (Conan et al., 2011; Grilleron and Kourta, 2010; Sung and Chang, 1988; Tortosa and Karbon, 2011). Numerical simulations using computational fluid dynamics become a major approaching tool to analyze the aerodynamic drag nowadays (Koike et al., 2004; Lee and Ahn, 2012; Levin and Rigdal, 2011; Song et al., 2010).

FEAF (Front End Air Flow) affects the aerodynamic drag as well as cooling performance a lot. Therefore, one of the ways to reduce the aerodynamic drag and to enhance the cooling performance is to manage the FEAF, which flows into the engine room and passes CRFM (Condenser, Radiator, and Fan Module) (Baeder et al., 2012; Ding et al., 2006; Hsieh and Jang, 2007; Kim et al., 2010, 2015; Kim and Kim, 2008; Kubokura et al., 2014; Lee et al., 2010; Pfeifer, 2014; Wittmeier and Kuthada, 2015; Xu et al., 2013; Yoo et al., 2010).

The grille shape influences FEAF a lot so that the study of grille shape must be performed to improve FEAF. Han et al. (2013) verified very well-known commercial flow solver FLUENT (2011) by comparison study between numerical simulation results and experimental results for the Ahmed Body, and current research uses FLUENT to obtain the research goals.

The aim of the current research is to build up the design guideline for grille opening shape. The parametric study on grille opening geometric shape enables to analyze the variation of air flow rate through grille opening area and to estimate the total aerodynamic drag of vehicle according to the grille shape change.

NOMENCLATURE

\( A \) : frontal projection area of car, \( \text{m}^2 \)
\( C_d \) : drag coefficient
\( h \) : vertical height of upper grille upper
\( H \) : vertical height of upper grille lower
\( V_\infty \) : free stream air velocity, \( \text{m/s} \)
\( w \) : horizontal width of upper grille upper
\( W \) : horizontal width of upper grille lower
\( \Delta C_d \) : drag coefficient difference between the base model and parametric simulation model
\( \Delta m_{\text{Grille}} \) : grill mass flow rate difference between the base model and parametric simulation model
\( \Delta m_{\text{Radiator}} \) : radiator mass flow rate difference between the base model and parametric simulation model
\( \mu \) : free stream air dynamic viscosity, \( \text{kg/(m·s)} \)
\( \rho \) : free stream air density, \( \text{kg/m}^3 \)

*Corresponding author e-mail: mskim@kau.ac.kr
2. NUMERICAL ANALYSIS

2.1. Modeling of Small Passenger Vehicle
A generic model shown in Figure 1 is selected as a numerical simulation model to accomplish the parametric study for the grille opening shape.

The grille blockage of the original generic model is totally removed to have parametric study and the front fascia surface around grille is flatten to reduce the number of surface mesh points as shown in Figure 2. All parts inside the engine room are kept as they are.

2.2. Grid Generation and Boundary Conditions
Figure 3 shows a computational domain of current numerical study, whose size is 6 times the length of the generic model, 5.7 times the width of the generic model, and 3.3 times the height of the generic model. One imaginary box is prepared inside the computational domain to have a dense grid zone around a car.

Twelve boundary layers are laid in order to figure out the flow separation phenomena precisely using \( y^+ = 5.0 \) as shown in Figure 4. The total numbers of surface mesh and volume mesh are about 1,000,000 and 15,500,000, respectively.

Inflow boundary condition is uniform flow condition of 110 km/h driving velocity and outflow boundary condition is pressure outlet condition of atmosphere pressure. The Realizable \( \kappa - \varepsilon \) turbulence model is hired to predict turbulent flow field (Goldberg et al., 2006; Singh et al., 2011). Finally, moving wall condition of 110 km/h is set on the ground to prevent the formation of boundary layer along the ground.

CRFM is located at the backside of grille inlet as shown in Figure 5 (a) and is biased to the right side a little bit from the intermediate position of grille inlet as shown in Figure 5 (b). Condenser and radiator shown in Figure 5 (c) are treated as porous wall. The pressure drop due to viscous