

Under-Hood Air Flow Evaluation of Pedestrian-Friendly Front-End Style Using CFD Simulation

Kambiz Jahani and Sajjad Beigmoradi
CAE Engineer, R&D Center of SAIPA

ABSTRACT

The efficiency of the vehicle cooling system strongly depends on the air flow through the radiator core. The flow through the radiator core in turn depends on other panels that are in the vicinity of the radiator. In this study, the effect of geometrical change at vehicle front-end including the whole bonnet, grille and bumper area is investigated by means of Computational Fluid Dynamics (CFD). Numerical modeling is carried out by means of CAE tools. Simulations are performed for maximum power and maximum torque conditions, monitoring the mass flow rate through the radiator core and velocity contribution over the radiator face. To the velocity field of the airflow, the heat exchangers are represented as porous media and fan module is modeled utilizing Multiple Reference Frame (MRF) approach. The validity of the developed simulation capability is tested by successful comparison with the available experimental data for the base model at the given operating conditions. On studying the model with complete new front-end style, local modifications are applied incorporating adding airguide, flap and anti-recycler in order to enhance the flow distribution in the vicinity of radiator and increase the mass flow rate passing through it. Modifications lead to more than 10% improvement in mass flow rate through radiator core in Pedestrian-Friendly Front-End Style.

CITATION: Jahani, K. and Beigmoradi, S., "Under-Hood Air Flow Evaluation of Pedestrian-Friendly Front-End Style Using CFD Simulation," *SAE Int. J. Passeng. Cars - Mech. Syst.* 7(2):2014, doi:10.4271/2014-01-0762.

INTRODUCTION

The airflow through the front end heat exchangers is critical for the cooling performance of a vehicle. Car manufacturers are taking this into consideration when designing a vehicle. Since many contradicting factors are defining the front end design, a clear understanding of the airflow behavior and its influence on the cooling performance is required. One practical way is to employ three dimensional CFD as a cost-effective tool for capturing the dynamic pattern of cooling air flow at vehicles' underhood. There has been some work in using CFD models for optimization of the front end cooling module. The work by Yang et al. [1] has used CFD based models to evaluate condenser-fan-radiator module (CFRM) against the conventional condenser-radiator-fan module (CRFM) at a fixed vehicle velocity. Their work although later extended to vehicle idle condition [2]. The other works include the study by Lee and Hong [3] that used a validated CFD model to modify the front end opening for optimizing the engine cooling performance. For other applications, a cooling fan and heat shield were designed and the distance between the fan and heat exchanger was optimized for a heavy-duty truck using CFD by Mao et al [4]. The aim of this work is to confirm CFD as a valuable tool for improving front end air flow in a face lift concept. This study describes a developed methodology to solve the airflow through the engine compartment and predict

the performance of the front end heat exchangers by means of CFD when the vehicle is cruising particularly at V_{max} along with other operating condition (see Table 2). The model is validated by comparing the predicted flow velocities across the radiator with experimental values. The validated CFD model is used to employ some strategies in order to increase the mass flow rate of air passing through the radiator core in a face lift concept of the base model. This concept possesses the main features of a pedestrian-friendly front end style. The way to achieve such interest is to optimize in the shape and geometry of front end openings or utilizing aerodynamics devices such as air-guide, air-dam, anti-recycler and etc to obtain a maximum air flow. Such use of validated CFD models has led to significant cost savings by reducing prototype testing and has also led to reduction of design cycle time.

THEORY OVERVIEW

In this study, the under-hood air flow driven by the cooling fan is treated as a steady-state, single-phase turbulent flow. The eddy viscosity is obtained with two-equation *realizable* $k - \epsilon$ turbulence model. The pressure drop through the heat exchangers (Radiator and Condenser) caused due to the flow resistance offered by them is simulated using a porous media approach. The viscous and inertial components of the pressure drop is determined by a curve fit of the measured pressure

drop-air velocity data for the heat exchanger of interest. Porous media are modeled by the addition of a source term to standard fluid flow equations [5].

$$S_{ij} = - \left(\sum_{j=1}^3 D_{ij} \mu v_j + \sum_{j=1}^3 C_{ij} \frac{1}{2} \rho |v| v_j \right) \quad (1)$$

The source term is composed of two parts; Darcy (the first term on the right hand side of equation) and the inertial loss term (the second term on the right hand side equation); where S_{ij} is the source term for the i_{th} (x, y, or z) momentum equation, $|v|$ is the magnitude of the velocity and D_{ij} and C_{ij} are prescribe matrices. This momentum sink contributes to the pressure gradient in the porous cell, creating pressure drop that is proportional to the fluid velocity (or velocity squared) in the cell. For recover case simple homogenous porous media the pressure drop caused due to the flow resistance can be estimated by equation (2). This method was first proposed by Patankar and Spalding [6].

$$\Delta p = - \left(\frac{\mu}{\alpha} v_i + C_2 \frac{1}{2} \rho |v| v_i \right) \quad (2)$$

The fan was modeled as a moving reference frame. The steady state approximation MRF, allows individual cell zones to rotate or translate with different speeds. This is achieved by dividing the domain into separate zones where the flow is solved in stationary or rotating coordinate systems. Therefore, the fan cylinder and fan blade comprise a separate region. To transform the fluid velocities from stationary to rotating frames, the following relations are used [7].

$$\vec{u}_r = \vec{u} - \vec{v}_r \quad (3)$$

$$\vec{v}_r = \vec{\omega} \times \vec{r} \quad (4)$$

where $\vec{u}_r$ is the velocity relative to the rotating frame, $\vec{u}$ is the absolute velocity and $\vec{v}_r$ the whirl velocity (due to the moving frame). $\vec{\omega}$ is the angular velocity and $\vec{r}$ is the position vector to the rotating frame. Solving the equations of motion in the rotating reference frame results in additional acceleration terms in the momentum equation [7].

$$\frac{\partial \rho}{\partial t} + \vec{\nabla} \cdot (\rho \vec{u}_r) = 0 \quad (5)$$

$$\begin{aligned} \frac{\partial}{\partial t} (\rho \vec{u}) + \vec{\nabla} \cdot (\rho \vec{u}_r \vec{u}) + \rho (\vec{\omega} \times \vec{u}) = \\ - \vec{\nabla} P + \vec{\nabla} \cdot [\vec{\nabla} \vec{v} + \vec{\nabla} \vec{v}^T] + \rho \vec{g} + \vec{F} + \vec{S} \end{aligned} \quad (6)$$

The Coriolis and centripetal accelerations are included in the momentum equation with the term $(\vec{\omega} \times \vec{u})$.

CASE STUDY

In this study, CFD simulation is performed to estimate and predict flow field through engine compartment of a B-segment car shown in Figure 1. The model is validated by comparing the numerical results with experimental data. Afterward, the CFD validated procedure is followed to simulate same concern in a face lift concept of the base car which can be considered as a pedestrian-friendly front end style.



Figure 1. Base model (left) and its pedestrian-friendly front end face lift (right).

According to ECE-78-2009 regulation for the lower legform, impact is imposed to the bumper at 40km/h velocity parallel to the longitudinal axis of the vehicle on at least three points where injuries or shape changes may result. The lower leg acceleration, knee shearing displacement and knee bending angle are measured. Some general guidelines exist that may be helpful for good leg protection [8]. These reported suggestions, which are approximated estimations, are valid only if they could cause good energy absorption in impact positions. Some of the main guidelines for leg protection are as follow:

1. Increase bumper overhang to implement deformation elements in front of bumper beam.
 - Implementing optimized low-density pedestrian protection foam in front of a stiff bumper cross-member to absorb the impact energy and provide sufficient deformation space to avoid the impactor hitting the stiff bumper cross-member.
2. Move forward lower bumper fascia area to control lower leg kinematics.
 - Installing optimized lower bumper stiffener firmly mounted to the chassis and bumper fascia, to control the leg kinematics by reducing knee bending.

Same guiding principles mentioned right before are applied for the new concept (See Figure 2).