Underhood Flow Analysis of a LCV to Improve Cooling System Performance for better fuel Economy

Ram Mohan Rao Pamoti  
Deputy Manager - CAE  
Mahindra Trucks and Buses Ltd  
Pune - 411019, India  
pamoti.mohan@mahindra.com

Chetan Raval  
Senior Manager - CAE  
Mahindra Trucks and Buses Ltd  
Pune - 411019, India  
raval.chetan@mahindra.com

ABSTRACT

In order to maximize engine cooling system performance of road vehicles one must critically examine the behavior and character of the vehicle underhood air flow. Critical parameters, such as flow hindrances and recirculation, must be located and modified in order to improve the cooling system performance. In this study Underhood flow analysis of MTBL LCV truck with front engine is carried out using Altair CFD solver ACUSOLVE. Flow field of the truck engine compartment is investigated at the most critical working conditions, Max. Torque and Max. Power. Air flow recirculation is observed in the cooling system area. The reasons for recirculation and recirculation zones are studied. Then cooling system sealing is proposed and verified for improvement. The simulation results show that the improvement is in the right direction.

Keywords: Underhood flow analysis, engine compartment, cooling system.

Introduction:

Cooling system for LCV is paramount important. Fan duty cycle and its energy consumption is part of it. The cooling of the Underhood of a heavy truck is very important. As the engine power increases and the space in the Underhood compartment decreases, efficient cooling becomes crucial. To gain knowledge about the cooling air flow of a truck in an early development stage could lower the time needed in the development process significantly. Developing methods to perform trustworthy simulations ought to produce more reliable and faster results than experiments using mock-ups or extremely simplified geometries and flow condition. CFD is a tool for simulating complex geometries and flows and is nowadays a complement to experiments. It offers the opportunity to save time and makes development processes more efficient. Though, in general, results obtained from CFD simulations are to be compared with the experimental results that needs to be carried out. The cooling air drag force could be as much as 8 % of the total drag force acting on the truck [1]. It is crucial to ensure that available ram air reaches to cooling system.

The complete Under hood system must be taken into account in order to optimize the thermodynamic performance. It was not sufficient to only improve a subsystem such as the front grill panel, under simplified conditions.

Process Methodology:

MTBL LCV vehicle is considered for Underhood analysis. The first step of the simulation process consisted of producing a cleaned surface. When the surface had been smoothened and cleaned a computational mesh was produced. Complete meshing is done using HYPERMESH. Acuconsole is used
for setting up the CFD boundary conditions. In other words, solving of governing equations using a discretization scheme and a turbulence model is done Acusolve to obtain the results.

**Geometry cleanup**

CAD geometry cleanup is performed to make smooth and clean surfaces and ensuring that it is a closed volume. Boundary conditions are applied on the surfaces. Completely close heat exchanger geometry (CAC and RAD) and rotational domain around fan to separate it from rest of the domain ensuring that it captures the flow physics.

![LCV truck](image)

**Figure 1: LCV truck**

**Grid development**

The global layout of the domain can be seen in figure 2. The domain represents a wind tunnel with the truck inside. The dimensions are shown in table 1. It is important that the distance from the truck to the boundaries are large, in order to ensure that they do not influence the flow around the truck.

![Schematic picture of the domain](image)

**Figure 2: Schematic picture of the domain**

**Table 1: Dimensions corresponding to figure 2**

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Truck dimensions, L<em>W</em>H</td>
<td>4.5<em>1.65</em>3</td>
</tr>
<tr>
<td>Domain dimensions, X<em>Y</em>Z</td>
<td>60<em>30</em>12</td>
</tr>
</tbody>
</table>
An overview of the surface mesh that is used as an input parameter for the production of the volume mesh is shown in figure 3. As can be seen, the geometry is quite detailed since most parts of the truck were included.

The cooling package consists of a Charge Air Cooler (CAC) and a Radiator (RAD). Radiator is largest and positioned close to the fan, CAC is placed in front of radiator. Shroud is connected to radiator with which fan can effectively suck air through both CAC and RAD. Fan is directly connected to engine crank shaft with certain gear ratio. Grill placed at front side of cabin allows fresh air to engine compartment. The Underhood compartment mesh of the truck can be shown in figure 4. Surface mesh of engine is done by wrapping to capture all features. Average surface mesh size is 15 mm and regions near cooling system are meshed with average element size 5 mm. As the fan speed is high, fan surface is meshed with average element size 1 mm to capture velocity gradients properly. Components away far from cooling system are meshed with average element size 20 mm.
In figure 5 the surface mesh of the porous media is shown together with a more detailed view of the fan blades. Notice that the cells are much finer in the tip region where tip leakage occur. In this region the velocity gradients are expected to be very high. The size of the smallest elements is approximately 0.5 mm in the stream wise direction, 1 mm in the span wise direction. It turns out that the largest $y^+$ values on the surface of the fan blades are situated in the tip region where $y^+ \approx 70$. 

![Underhood compartment](image)
Volume mesh is generated using unstructured tetra element type. The unstructured mesh consists of approximately 25 million cells. Since the main area of interest was the underhood compartment, the majority of the cells were placed there. Approximately 50 million cells are required for underhood analysis but mesh size is reduced to 25 million to make it run on limited computational resource.
The part of the underhood compartment that required the most amount of cells is the region near the fan and the engine. Approximately 13 million cells were placed in this region. Zoomed views of the fan region (rotational domain) is shown in fig 6.

Figure 6 Mesh in the fan region

A mid cut of the global layout of the mesh in the vicinity of the truck is shown in figure 7.

Figure 7 Side view cut in the mid plane of the mesh

Top view of the mesh at plane which cuts through fan hub is shown in fig 8. Obviously, the mesh is much denser in the underhood compartment, since this is the area of interest in this study.
Complete CFD domain consists of four different meshes i.e. the outer static domain, Intercooler, radiator and the rotational domain. Each one is separated with surface mesh to define proper interior boundary condition.

**Setup of the CFD solver**

Underhood simulation is done for Max.Torque and Max.Power vehicle operating conditions where cooling system loads are critical. In this, flow is assumed as steady state and RAN's one equation turbulence (Spalart-Allmaras) is used to model turbulence.

**Boundary condition:**

In the rotational domain, as well as in the static domains, all surface parts of the truck were defined as no-slip walls. The truck cabin, container, roof deflector and wheels were assigned as no-slip surfaces with zero relative velocity. The inlet of the domain was set to a uniform normal velocity which is Vehicle speed, m/s. The ground plane was modeled as a no-slip surface, with a constant translational velocity matching the forward speed of the truck. The side walls and the roof of tunnel were set to slip condition. A pressure outlet condition was applied to the rear face boundary of the model domain. The velocity vector direction was chosen to match $0^\circ$ yaw angle. A constant initial eddy viscosity condition was specified to be $0.00001$ m$^2$/s. The rotational speed of the fan was set to X rpm. An overview of the domain can be seen in figure 9.

**Porous media and MRF**
The cooling packages both Intercooler and radiator are modeled using porous media. The flow was constrained to be directed normal to the inlet of the porous media. The Forchheimer equation describes the pressure drop over the two porous media (CAC and RAD). The input values for the porous media are pressure drop through CAC and RAD for different mass flow rates and its core thickness.

Rotational domain of the fan is modeled as MRF where rotational speed is set as fan speed corresponding to vehicle operating condition.

**D. Numerical methodology**

In this work, the Navier-Stokes equations were solved using AcuSolve, a commercially available flow solver based on the Galerkin/Least-Squares (GLS) finite element method. AcuSolve is a general purpose CFD flow solver that is used in a wide variety of applications and industries. The flow solver provides fast and efficient transient and steady state solutions for standard unstructured element topologies. AcuSolve ensures local conservation for individual elements. Equal-order nodal interpolation is used for all working variables, including pressure and turbulence equations. The resultant system of equations is solved as a fully coupled pressure/velocity matrix system using a preconditioned iterative linear solver. The iterative solver yields robustness and rapid convergence on large unstructured meshes even when high aspect ratio and badly distorted elements are present.

The following forms of the Navier-Stokes equations were solved by AcuSolve to simulate the flow around the Truck:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0 \tag{1}
\]

\[
\frac{\rho}{\phi} \frac{\partial u}{\partial t} + \rho u \cdot \nabla u + \nabla p + Rf = \nabla \tau + \rho b \tag{2}
\]

where \(\rho\) is the density; \(\phi\) is the porosity; \(u = (u_1, u_2, u_3)^T\) is the velocity vector; \(f = (f_1, f_2, f_3)^T\) is the porous media contribution, defined in the orthogonal principal axes; \(R\) is the rotation tensor which rotates \(f\) to the global coordinate axes; \(p\) is the pressure; \(\tau = [\tau_{ij}]\) is the viscous stress tensor; and \(b\) is the specific body-force.

For the steady state solutions presented in this work, a first order time integration approach with infinite time step size was used to iterate the solution to convergence. Steady state convergence was typically reached within 500 time steps.

**Results & Discussions:**

In this section the results obtained from underhood simulation are discussed. Calculations are performed for vehicle Max Power and Max.Torque conditions. Reason for choosing Max.Power and Max.Torque is, in former case heat rejection is more and in the later case vehicle speed is less. Initially simulation is done on base line design. A mid cut of the average velocity distribution in the underhood compartment can be seen in figure 10. The air enters the front grill on the left hand side air then flows through the porous media, interior of the shroud and into the rotational domain. Behind the fan, the flow will experience a path blockage, forcing the air to travel along the surface of the engine. As the air travels past the engine it leaves the underhood compartment. The velocity of the flow is highest in the tip regions of the fan blades.
Velocity distribution near cooling package in X-Y plane is shown in figure 11. It is clear from the picture that good quantity of air flows to Underhood compartment because of rams effect at the grill.

The influence of the front grille and chassis front cross member is obvious when observing the velocity fields in the CAC and RAD planes of the porous media. Figure 12 clearly shows that the ACC creates a distinct footprint in the velocity flow field in CAC and RAD. It almost looks discontinuous. It can also be seen that the flow is accelerated by the fan.
In figure 13 the relative pressure distribution on the surface of the fan can be seen. At the front surface (suction surface) of the blades, the relative pressure is low since the flow is accelerating over the surface of the blades. Due to the curvature of the blade, the flow is forced to change its direction away from the blade on the pressure side of the blade. The pressure is higher on the top half of the back surface because the engine is situated closer to the fan in this region. There is a small area with low pressure as well, situated immediately after the flow has passed the leading edge. The low pressure region keeps the flow attached to the blade. Further downstream the pressure on the surface of the blades increases, forcing the air to flow away from the blade and downstream towards the engine. Having the pressure distribution on the front-end of the engine in mind, it is not surprising that the pressure is higher on the blades in the top half of the fan.

Figure 14 shows that the front-end of the engine is subjected to high pressure compared to the rest of the underhood compartment. Due to the turbulent characteristics of the flow just behind the fan, the curvature of the geometry creates wakes with low-pressure regions in some regions on the front-end as well.
In order to achieve effective cooling it is important to avoid recirculation regions. The overall cooling performance decreases as the mass flow through the underhood compartment decreases. In recirculation regions the temperature may rise and in worst case damage the underhood components. The air that travels through the fan is represented by 3D streamlines in figure 15. Preferred flow paths and recirculation regions may be identified. On both sides of the shroud large recirculation regions are spotted. Note that some parts in the underhood compartment are excluded.
Recirculation of the air exiting the radiator outlet to the intercooler inlet at vehicle 1st gear conditions is of considerable importance. At higher vehicle speeds the recirculation of air is less. The effectiveness of the cooling package is directly proportional to the temperature difference (ΔT) across the heat exchangers (Intercooler and radiator). With an increase in the recirculation of hot air, there is a decrease in ΔT, which in turn would reduce the heat rejected. This would lead to inefficient engine cooling and increase in the radiator top tank temperature. Recirculation Factor (RF) quantifies the amount of air re-circulated. RF is defined as the ratio of the mass of air that recirculates from the radiator outlet to cooling package inlet to the amount of total airflow through the radiator outlet. RF for Max.Torque and Max.power are is 30% and 20% respectively. This re-circulated flow has a significant effect on the performance of engine.

To eliminate recirculation in underhood compartment sealing flap is proposed. Location and design is shown in figure 17.

![Recirculation sealing flap](image1)

**Figure 13 Modified design**

Underhood flow simulation is performed on vehicle with sealing flap. Simulation and mesh parameters are kept similar to base line design. 3D flow stream line plot is represented in figure 18. Recirculation spotted in earlier design is drastically reduced which allows more fresh air to the cooling system thereby improve cooling system performance.

![Figure 14 Stream lines on modified design](image2)
Conclusions:

A front end flow CFD study considering all underhood/underbody components is presented here. Results of this front-end flow study are useful to develop an efficient vehicle cooling package. Results of the simulation are summarized as below

1. Radiator and intercooler mass flow rates (kg/s) are predicted for different vehicle operating conditions. As the restriction at the front of a cooling system is less, radiator and intercooler mass flow rates are slightly exceeding design criteria.

2. 20% and 30% of hot air is re-circulated back to cooling system during Max.Power and Max.Torque vehicle operating conditions respectively.

3. The importance of reducing recirculation in a full vehicle front-end flow is explained and effect of sealing is discussed.

Benefits Summary:

With the help of ACUSOLVE, Leading commercial finite element CFD code we at MTBL product development team quickly take a decision, whether we go for sealing or not, without doing wind tunnel test which is expensive. And it is very useful for us to understand vehicle underhood flow behavior and its characteristics without spending much time as it was in physical test. By eliminating recirculation flow we could able to improve cooling system efficiency which will reduce Fan duty cycle time in turn benefit fuel efficiency.

Challenges:

We have done Underhood analysis we used RAN's one-equation turbulence model. with 2 equation model even though results much accurate but grid size should improve which is not good turnaround to us.

ACKNOWLEDGEMENTS

The authors would like to thank Mr. Chintamani Mhaskar, Manger, MTBL Engine Application team for his valuable support and contributions during this project.

We also would like to thank Mr.Shekar Paranjape, General Manager, MTBL for encouraging us to publish this paper.

REFERENCES: