

The Cooling Airflow of Heavy Trucks – a Parametric Study

Thomas Hallqvist
Scania CV AB

Copyright © 2008 SAE International

ABSTRACT

In order to maximize the cooling performance of road vehicles one must be familiar with and understand the behavior and character of the underhood flow system. Critical features, such as bottle necks, must be isolated and modified in order to improve the performance. The underhood flow system can be thought of as a network of subsystems, among which no one can be neglected since they are all, in some way, linked. This paper describes the general character of the cooling airflow on Scania heavy trucks. The variation of installation parameters and studies of different design concepts reveals the sensitivity and complexity of the cooling airflow. The results indicate, among other things, that the depth and also the shape of the fan shroud are of great importance. From the results also the influence from flow uniformity on the pressure loss and cooling capacity is obtained.

INTRODUCTION

The demands on efficient engine cooling on heavy, high performance, trucks are hard to meet. These demands will most likely become even tougher to fulfill in the near future. This is so since the engine power successively increases and the available space in the underhood tends to decrease. The required heat rejection does not only increase due to the higher engine power, but also due to exhaust after treatment, air conditioning and auxiliary units. Parallel to the underhood flow optimization, meaning that the needed amount of cooling air flow can be achieved more efficiently, efforts must be made in order to optimize the thermodynamical performance of the cooling system. This since the cooling airflow has a negative effect on the total drag. The cooling air drag can be as large as 8 % of the total air resistance [1]. One way of improving this is to minimize the amount of cooling air flow at high speeds by closing the front air intakes.

Experimental studies on cooling system performance have been conducted since the beginning of the 20th century [2]. More recently, Taylor and Chu [3] studied the influence from installation parameters on the cooling performance on trucks. They concluded that the fan

should be immersed by 60-70 % in the fan shroud. They also concluded that the space between the fan and radiator should be maximized and that the space between the fan and the engine block was of limited importance. Furthermore, they stated that the type of fan shroud used is not of importance. This statement however, seems to be based solely upon the two considered shrouds.

Similar experimental studies were performed by Magnusson [4]. This work where carried out at the former fan test rig at Scania CV AB in Södertälje, Sweden. This study showed that the shape of the fan ring was of importance for the amount of flow generated by the fan. Without the engine dummy installed a sharp edged fan ring gave the highest flow rate. With the engine dummy installed a fan ring with a smooth trailing edge was the most efficient one. Furthermore, with this fan ring the flow rate was higher with the dummy than without. The smooth trailing edge can be interpreted as a fan being less immersed into the fan ring. The influence from fan-to-radiator spacing and fan-to-engine spacing was also studied, with similar findings as in the work by Taylor and Chu. Contrary to [3], Magnusson's studies showed that the design of the fan shroud influenced the underhood flow.

During the years several theoretical models for the underhood flow system have been developed. The most common ones are one-dimensional approximations. These models can be a great aid in the design of vehicle engine cooling systems. Furthermore, the turn-around time for this kind of analysis is short why comprehensive studies can be conducted at low cost. Theoretical analysis is, also, an efficient way to achieve basic understanding of the thermodynamical system, e.g. identifying important dimensionless parameters, fundamental relationships [5,6,7].

Fully three-dimensional numerical studies, of differing complexity, of the underhood flow system have been conducted by, among others, [8,9,10,11,12]. In the work by [10] the fan performance curves and system resistance curves were computed numerically. In order to increase the accuracy when determine the system resistance curve a new method was developed by [11]. This method focus on systems where the flow is between the inlet grille and the cooling package.

Strongly simplified CFD analysis of the cooling performance was conducted by [9]. Even though only the front part of the vehicle was considered, they obtained good agreement on velocity distributions and temperatures with experimental data. This shows that a simplified computational model with short turn-around time can successfully be used. This methodology is particularly suitable in the early design phase.

In the work by [8] the feasibility of using a one-dimensional network analysis tool was studied by means of CFD simulations. The results showed that the 1D tool was successful in estimating the cooling performance. However, caution should be taken in situations where the velocity distribution is highly non-uniform.

More extensive work on the influence from flow uniformity and temperature uniformity have been performed by Chiou [13,14,15]. The result indicates that the uniformity is of importance and should always be taken into account. For some typical installation the importance may even be significant.

In this work the focus is on studying the sensitivity from various installation parameters. When the bottle necks have been identified, it is more straight forward to make effective design changes. The method used is 3D numerical simulation of a complete truck.

COMPUTATIONAL TECHNIQUE & ACCURACY

The simulations presented within this paper are isothermal runs performed in a large virtual windtunnel, in which the streamwise direction represents the x-direction and the vertical represents the z-direction. Within the tunnel a complete truck (Figure 3) is positioned. The location of the truck and the size of the tunnel assure that the results are not affected by the boundaries. The inlet of the windtunnel is applied with a constant velocity field and the outlet is equipped with a pressure condition. The floor of the tunnel is specified as moving ground, with a speed equal to the inlet velocity. The wheels of the truck are defined as rotating walls. The fan is modeled using the MRF¹ methodology. The finest elements are concentrated to the fan region and measure approximately 0.001 m. In order to determine the sensitivity of the mesh size, grid independency studies have been performed. Particularly with respect to the resolution of the fan and adjacent geometries. This study was done by doubling the smallest element size. The influence from this coarsening on the overall massflow was less than one percent and negligible with respect to the absolute static pressure. This verifies that the present mesh is sufficient. The used solution method is the Lattice Boltzmann Method.² This method implies,

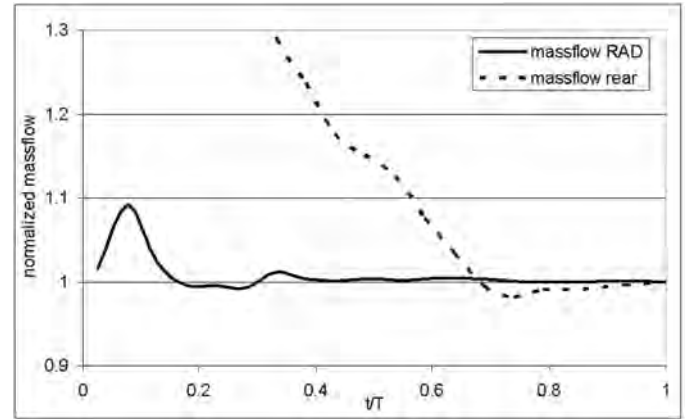


Figure 1. Convergence of massflow through the radiator and through the evaluation surface behind the engine.

among other things, that the simulations become transient. For further details on this methodology see e.g. [16]. The timestep, which is given by the spatial resolution and the velocity scale, is here approximately equal to 4 μ s. Each simulation has been run to convergence of massflow through the radiator. Also criteria on massflow behind and underneath the engine have been considered. The massflow through the radiator typically converge within ± 1 % after approximately $t=1/6$ of the total runtime T , as depicted in Figure 1. The massflow through the evaluation surface behind the engine converges to the same value after approximately $t=4/5T$. As seen, the massflow through the evaluation surface experiences a dramatic change. This is due to the initialization of the velocity field. Better convergence is achieved if the initial velocity field is fed by an earlier, similar, run. The sampling of data was initialized as the velocity field reached a stable state, i.e. after $t=0.8T$. The total sampling time is enough to obtain statistically converged data, within an accuracy of at least ± 0.5 % for the evaluated macroscopic properties. Regions of main interest, such as the cooling package, experience even higher accuracy, within ± 0.25 %.

For the simulations, which were performed at our computational center, 128 cpu's were used. The total simulation time for one isothermal case was around 22 wallclock hours. The typical model size was about 40×10^6 voxels (20×10^6 fine equivalent) and 20×10^6 surfels (10×10^6 fine equivalent).

Heat exchangers and the mesh screens in the front grille panel have been modeled using the porous medium approximation. To increase the efficiency of the simulation some cases have been run without explicit treatment of the screens in the front. Instead the resistance of the screens has been applied to the radiator. Tests have shown that this simplification does not influence the massflow through the radiator and the underhood flow character. However, the distribution of the static pressure between the front grille and the radiator becomes wrong. Furthermore, as depicted in Figure 2, the flow distribution through the radiator

¹ Multiple Reference Frames.

² The commercial software PowerFLOW from EXA CORP has been employed.

becomes slightly different. For the simplified model the flow distribution is a bit smoother.

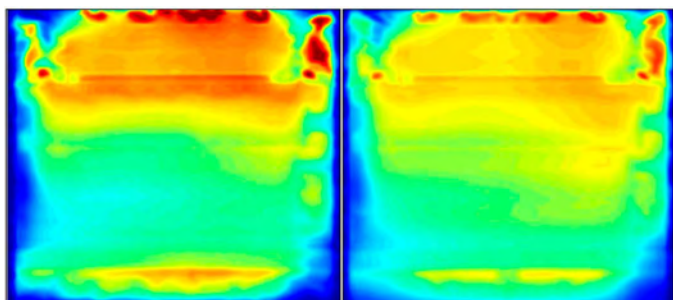


Figure 2. Normalized flow distribution through the radiator. Default model to the left and simplified model to the right. (Blue color represents the lowest value and red the highest.)

The computational results have, when feasible, been compared with experimental results. Available from physical testing are the cooling capacity, local temperatures and micro-probe measurements of the air flow through the radiator for some specific truck configurations.



Figure 3. Picture of the studied reference truck, a two axis tractor.

The agreement with micro-probe measurements are within 6 % (Case NF). The thermal runs that have been conducted (not represented in this paper) verify, both qualitatively and quantitatively, that the present computational method and setup provides data within reasonable absolute accuracy. The cooling capacity is predicted within 5-8 % and specific fluid temperatures are predicted within 6-12 %. Absolute values are impossible to determine exactly, both computationally and experimentally. However, relative values are captured within greater accuracy.³ No measure of this accuracy can be stated here since experimental data is

³ Experience from external aerodynamical simulations.

lacking. This is of course due to the complexity involved in modifying a real truck.

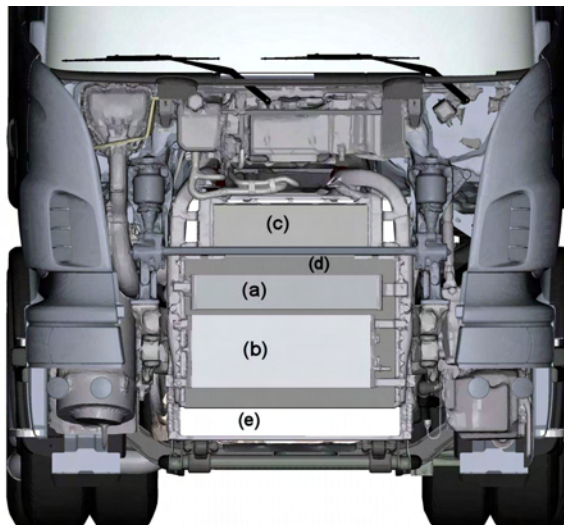


Figure 4. Picture showing the cooling package installation.

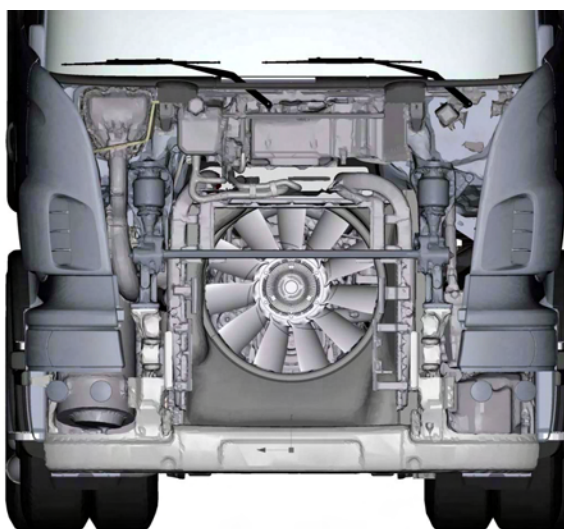


Figure 5. Picture showing the fan configuration.

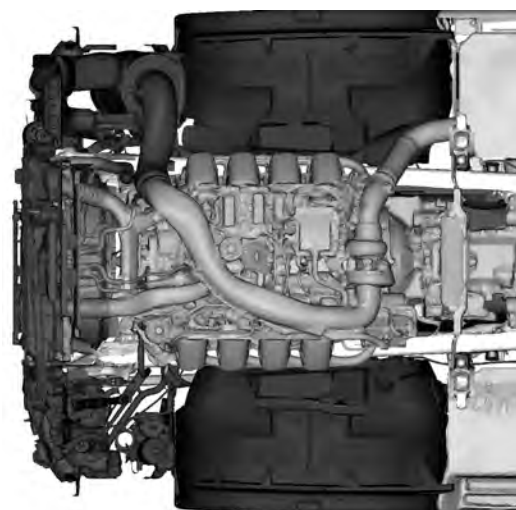


Figure 6. Picture of the underhood environment.

iD	name	Description	ref.	screens	engine
1	Case REF	Basic reference case.	-	X	V8
2	Case REF*	As Case REF but without explicit modeling of the mesh screens.	1	-	V8
3	Case FS	Improved fan shroud, featuring a slender shape.	2	-	V8
4	Case NF	New front with approximately 50% increased total screen area.	1	X	V8
5	Case NE	No lower encapsulation.	2	-	V8
6	Case CF	Fan centered to the middle of the radiator.	5	-	V8
7	Case dx20+	Engine and fan moved 20mm in the x-direction and improved fan shroud.	3	-	V8
8	Case dx60+	Engine and fan moved 60mm in the x-direction and improved fan shroud.	3	-	V8
9	Case dx100+	Engine and fan moved 100mm in the x-direction and improved fan shroud.	3	-	V8
10	Case w20+	Cooling modules widened by 20%, improved fan shroud and new front.	-	X	V8
11	Case w20+dx30+	As Case w20+ but with engine and fan moved 30mm in the x-direction.	10	X	V8
12	Case w20+dx60+	As Case w20+ but with engine and fan moved 60mm in the x-direction.	10	X	V8
13	Case w20+dx90+	As Case w20+ but with engine and fan moved 90mm in the x-direction.	10	X	V8
14	Case w20+D20+	As Case w20+ but with 20% larger fan diameter (900 mm).	10	X	V8
15	Case dx*100+	Engine moved 100mm in the x-direction.	2	-	V8
16	Case dx*100+EF	As Case dx*100+ but with modified engine front.	2	-	V8
17	Case S6	Basic case for the inline (straight) six engine configuration.	2	-	S6
18	Case S6dx60+	As Case S6 but with engine and fan moved 60mm in the x-direction.	17	-	S6
19	Case 50%	As Case FS but with 50% projection of the fan into the fan shroud (fan ring)	3	-	V8
20	Case 33%	33% projection of the fan into the shroud. (Default projection is 67%.)	3	-	V8
21	Case S6 FS	As Case S6 but with a more streamlined fan shroud.	17	-	S6

Table 1. Overview of computational cases. Column number four ("ref.") shows which iD the reference has. "screens" implies if explicit modeling has been applied (X) or not (-).

TRUCK CONFIGURATION

The considered truck type is a two axle tractor, shown by means of the surface mesh in Figure 3. The velocity of the truck is fixed and equals 30 km/h, representing low speed hill climbing. The mesh screens are represented by the black regions in the front grille panel. There are two additional mesh screens in the bumper. Behind the front grille panel the cooling package is located, as shown in Figure 4. The cooling package consists of: (a) an oil cooler, (b) a condenser, (c) an EGR cooler, (d) a charge air cooler (CAC) and (e) a radiator. The radiator is the biggest one and covers, as seen, the complete cooling package. In the studies where the cooling package is widened, all included heat exchangers undergo the same scaling in the y-direction. In Figure 5 the fan shroud and fan installation for the reference truck is shown. The complete fan cover consists actually of two separate parts. One mounted on the cooling package (the fan shroud) and one mounted on the engine (the fan ring). These two are connected by a rubber sealing. In the computation, though, these two parts are treated as one.

The default fan diameter is 750 mm. The fan speed is the same for all simulations and equals 1700 rpm. This speed originates from the actual driving conditions at hill climbing, at which engine cooling is a great challenge.

The layout of the underhood is depicted in Figure 6. This picture shows, as also all others pictures do, the surface mesh and not CAD data. It is clear that the number of components and the complexity of the underhood system are significant. For comparison, two different engine types have been considered. One V8 configuration (shown in the picture) and one inline six configuration (S6). The blockage effect for these two engine types differs strongly, particularly in the upper regions of the engines. Except from the engine layout

the fan is situated at a lower position, further to front, for the S6 configuration. For the basic V8 truck the lower part of the fan is right above the crossmember, whereas for S6 the lower part of the fan is in front of the crossmember (see Figure 11).

COMPUTATIONAL CASES

The computational cases presented in this paper are described in Table 1. The work flow within this study follows a logical pattern. The first case is on the present truck configuration. Thereafter, only one installation parameter is changed and evaluated at each time. However, for some cases more than one installation parameter is modified (relative to the basic reference case) and, hence, a different reference case is used in the evaluation (see table).

FLOW CHARACTER

The cooling airflow is of high complexity, why the most detailed physical phenomena is impossible to take into consideration, both experimentally and computationally. In the computation no effort has been made to fully resolve boundary layers, instead wall functions have been used for no-slip boundaries. This methodology is sufficient for these kinds of flow cases. The main contribution from, for example, the engine is a large scale blockage. For which small scale details are rather irrelevant.

A snapshot of the cooling airflow is shown in Figure 7, by means of streamlines colored by the static pressure. The organized flow right in front of the truck features a static pressure close to the ambient pressure (colored orange). As the flow passes through the mesh screens the static pressure decreases (colored green) and the

flow becomes more unstructured. As the heat exchangers are modeled as porous media, the unsteady flow becomes of a more organized character. This is due to that only flow in the x-direction is possible within the porous media. However, even if the resolved disturbances are dampened, turbulence is still accounted for via the explicit turbulence model.

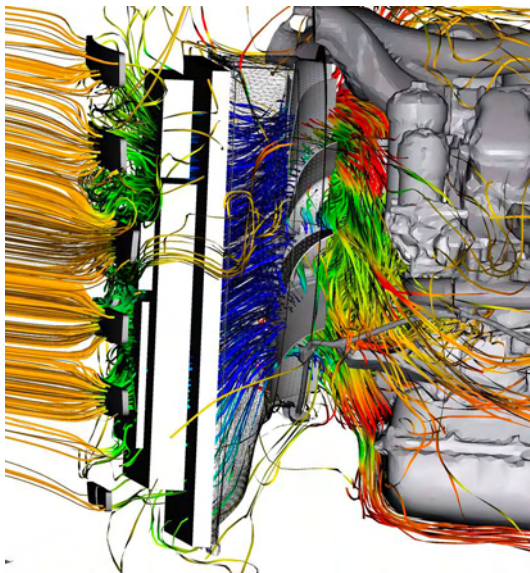


Figure 7. Streamlines colored by static pressure (*Case REF*). The showed parts are the front mesh screens, the five separate heat exchangers, the fan shroud, the fan and the engine. (Blue color represents the lowest value and red the highest.)

Behind the cooling package the static pressure has decreased even more (colored blue), due to losses in the heat exchangers. This decrease is augmented through out the fan shroud and a minimum in pressure is obtained just in front of the fan. The pumping effect of the fan results in pressure build-up. This increase corresponds to the total static pressure loss throughout the underhood, i.e. the pre-fan pressure drop plus the post-fan pressure loss. Behind the fan the flow becomes highly unstructured and rotational. Recall that the present computational model approximates the fan via the MRF technique. This is reflected in the computational results, as the fan is fixed in space and rotates only mathematically. This explains why the streamlines in the fan region may look strange and, also, why the flow at some locations is not symmetric in the right sense (e.g. see Figure 10).

In the underhood the flow is not restrained by guide vanes or encapsulations. Instead the cooling airflow features high flexibility and is allowed to go in a number of directions. The flow distribution depends strongly on the local pressure losses. As depicted in Figure 8 the flow features three main directions, i.e. above the engine and through the rear encapsulation, downwards towards the front left corner and downwards towards the front right corner. This type of situation, where the underhood flow is not controlled, can of course have negative

consequences, such as high thermal loads. On the other hand the high degree of freedom may be an advantage from a restriction perspective. A well defined optimized engine tunnel may still be advantageous.

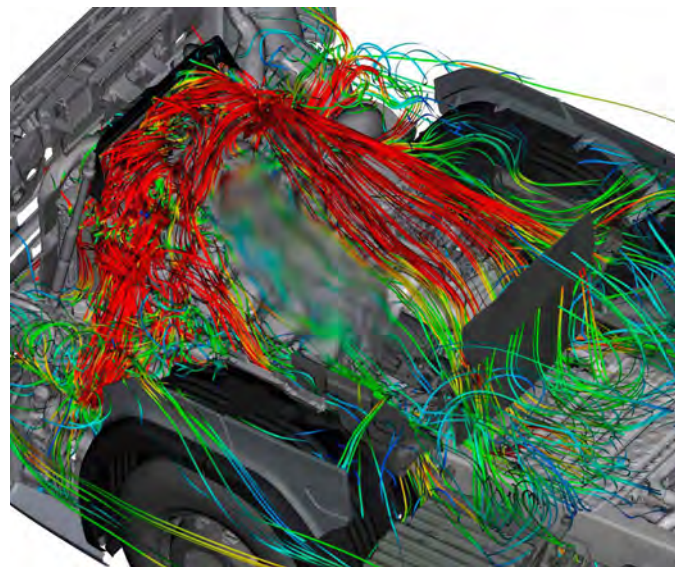


Figure 8. Underhood flow character, illustrated by streamlines colored by velocity magnitude for *Case REF*. (Blue color represents the lowest value and red the highest.)

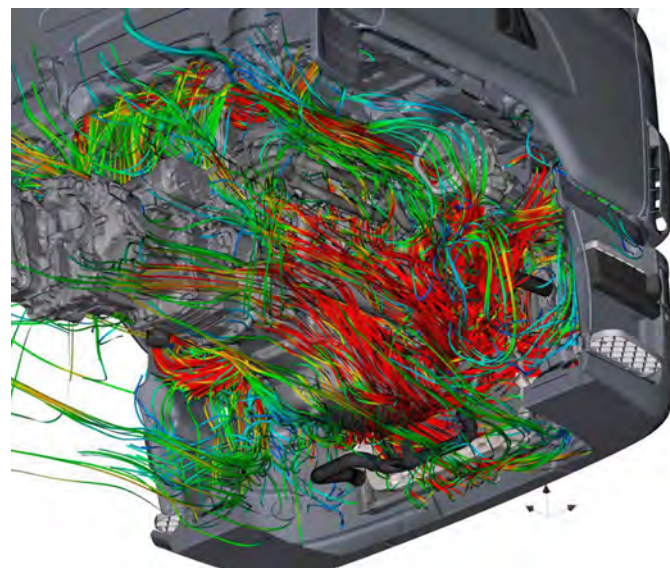


Figure 9. Flow character from below. Streamlines colored by velocity magnitude for *Case REF*. (Blue color represents the lowest value and red the highest.)

Looking from below (Figure 9) the airflow underneath the cab, above the engine, is seen. It is also possible to distinguish the flow within the lower encapsulation and also on the sides of the engine. Apart from the rear encapsulation, the cooling airflow is mainly evacuated via the wheelhouses and the lower encapsulation. Furthermore, a certain amount of the cooling airflow is not evacuated and undergoes recirculation. This is of significant importance for the cooling capacity. This issue has not been addressed within this paper.