The mutual influence of the A/C condenser and radiator cooling system

A Petov, S Bannikov and V Petrova
Kurgan State University, 40, Tomina Street, Kurgan, 640000, Russia
E-mail: alex_p2@mail.ru

Abstract. Car air conditioning systems are installed on many cars. The system leads to direct energy costs. In addition, the system elements have an additional influence on the efficiency of the engine cooling system, inasmuch as both systems use the same air flow for cooling. The mechanism of the air flow passing through heat exchangers has been investigated. The study showed that part of the air flow does not pass through the core of the condenser, but flows around it along the external contour. The degree of flow-around depends on several factors.

1. Introduction

One of the main environmental problems of the present time is air pollution. In large cities, motor vehicles are the main source of air pollution. In this regard, the automotive industry engineers are faced with the task of reducing fuel consumption and emissions of harmful substances; this requires a search for such opportunities. One of the areas where the potential is still carefully investigated is aerodynamics. The aerodynamics of a passenger car depends on many factors, one of which is the passage of cooling air through its internal space [1-7].

The drag coefficient of a passenger car depends on the amount of cooling air passing through its engine compartment. In order to more effectively use this air, it is necessary to determine the factors influencing this process. Optimization of the air conditioning and cooling system of a passenger car is possible only after studying the processes of cooling air passing through the air path of the cooling system. Solving this problem with the help of physical modeling in a wind tunnel or on a specialized aerodynamic stand would be a costly task, and the testing is time-consuming. Alternatively, automobile manufacturers use computational fluid dynamics (CFD) simulation, which helps to reduce the research time [8]. Through the use of CFD, the costs are reduced and the accuracy and reliability of the results are increased. The work shows that in order to reduce the aerodynamic drag of a car, it is necessary to reduce the flow of air through the engine compartment, but this optimization is limited by the efficiency of the cooling system.

The research [9] deals with the issues of optimization of the front part of the air path of a passenger car. The studies were performed using numerical CFD simulations. The CFD model was built for various driving conditions. The model contained a complete and detailed layout of the geometry under the hood including a condenser and a radiator. It is noted that the location of certain elements affects the air flow passing through the condenser and radiator. The velocity field along the frontal area of the heat exchangers is carefully analyzed. The task was to locate the stagnant zones of the air flow on the frontal area of the heat exchangers, to find out the causes of this problem and to identify the ways to eliminate
it. Uneven distribution of the air flow in heat exchangers indirectly reduces their effective area. The frontal zone with limited air flow does not actually participate in heat radiation.

The work [10] deals with the question of the joint influence of a capacitor and a radiator on each other. Studies show that the heat released at the condenser adversely affects the temperature of the radiator coolant, while the inlet temperature of the coolant when the air conditioner is on increases by 7° C. Studies were performed for various conditions of engine load and vehicle speed. The calculated values performed using CFD were compared with the results of experimental studies. CFD predictions and experimental coolant temperature data at the entrance to the radiator are well correlated and range from 2 to 3 °C.

The analysis of the presented works shows that the optimal organization of the air flow passing through the heat exchangers of the cooling and air conditioning system of a passenger car reduces the air consumption, and at the same time the aerodynamic drag of a car decreases [1-7]. There is a lot of potential in this direction. On the other hand, in order to reduce the flow of cooling air and improve the efficiency of the cooling and air conditioning system, it is necessary to know the mechanism of the formation of the air flow passing through heat exchangers [8-10]. The analysis of the work carried out renders it possible to trust the results of CFD calculations with full confidence. In practice, it is very difficult to obtain similar results with the help of full-scale modeling due to the limited technical means for measuring air velocity.

The purpose of the work is to study the mechanism of the mutual influence of the condenser of the car’s air conditioning system and the engine cooling system radiator on the flow of cooling air passing through them.

2. Materials and methods
Studies were performed using CFD. The condenser, radiator and fan installation were presented in the form of three-dimensional models with full details (Figure 1). Aerodynamic characteristics of the condenser and radiator were obtained experimentally.

![Figure 1. Model of cooling unit.](image)

Gas-dynamic modeling was realized using the model of an “aerodynamic chamber” simulating the engine compartment of a car. The chamber was divided into two parts by a partition, on which the cooling unit was placed. The front part located a preradiator chamber with two air intake openings. Behind the partition there was an engine compartment with a simplified “engine model” and the outlet was in the bottom.
Airflow modeling was carried out by imitating the oncoming flow to the vehicle during its movement. It was noticed that the air flow passing through the condenser is less than that through the radiator by 11%, while the condenser is installed in front of the radiator (Figure 1). In fact, this is easy to explain, if the entire cooling air passes through the radiator, since the radiator is fixed on the partition of the “engine compartment” and there is no more room for it to pass. The condenser is installed “freely”, so part of the air flows around the condenser on its outer side (Figure 2). The figure shows the trajectory of the air flow around the condenser. The mechanism of this phenomenon needs to be considered in more detail, it is necessary to find out how much air passes through the condenser core, how much of it passes by, what it depends on and how it affects the passage of air through the radiator; for this, a set of additional studies was performed.

Figure 2. The trajectory of the air flow through the cooling unit (in the horizontal section plane, view of the left side).

In order to investigate the mechanism of mutual influence of the condenser and the radiator on the air flow, the model of the “aerodynamic chamber” was simplified. In the preradiator chamber, the air was taken over the entire cross section of the chamber, thus the initial deformation of the flow was excluded. The fan system and the “engine” were removed for the same reason. The air flow was simulated using the incoming flow. In order to be able to compare different options, the air flow in all cases was set at 1.8 m³/s (which corresponds to a high vehicle speed).

3. Result and discussion
Figure 3 shows that the speed at the edges of the condenser is less than in the middle part. The core of the condenser as well as of the radiator has a uniform distribution across the front of the drag. Therefore, due to the dynamic pressure on the central part of the condenser core the air flow is forced to go in a straight direction, i.e. it immediately enters the core of the radiator. Along the edges of the condenser, the flow follows the path of least resistance, i.e. most of the flow streams around the condenser and enters the radiator along a curved path. When the path is bent, the flow also experiences resistance, so part of it still passes through the core of the condenser. The depth of the cross flow penetration between the heat exchangers and the amount of air around the condenser are limited by the flow resistance along this channel.

As noted above [10], the air flow at the condenser outlet heats the air entering the radiator, in addition, the condenser affects the air flow entering the radiator. Figure 4 shows that the plot of air velocities along the frontal surface of the radiator has an inverse shape compared to a condenser. Here, on the
contrary, the air velocity in a circumferential direction is higher than in the central part. This is additional air that bypasses the condenser.

**Figure 3.** The plot of the velocity distribution on the frontal surface of the capacitor.

**Figure 4.** The plot of the velocity distribution on the frontal surface of the radiator.

CFD technology allows you to trace the entire process of air flow passing through heat exchangers. The conditions of propagation of air flowing around the condenser were analyzed by the velocity of the air in the transverse direction between the core of the condenser and the radiator. The air flows around the condenser perimeter-wise, i.e. wraps around it in horizontal and vertical planes. The conditions of the air diffusion in all directions are different.

In the horizontal plane, the condenser is not symmetrical, on the right side of the condenser the receiver-dryer is structurally located. It occupies the entire right side of the condenser and is an additional barrier to the passage of air. Therefore, the flow around the capacitor on the left was considered as a control variant. Figure 5 shows the results of research. The control of air flow was carried out in the transverse plane along the surface of the core of the condenser and radiator. The average velocity $v$ along this plane was determined. The assessment was carried out for three options for installing the air conditioner relative to the radiator; in the basic version the distance between them was 16 mm, then the distance was increased to 22 and 28 mm. As seen, the penetration depth $l$ of the air flow between the heat exchangers is in the range of about 70 mm from the edge of the radiator. Moreover, the farther from the edge, the lower the speed of the transverse flow, i.e. part of the air gradually escapes through the core of the radiator. The flow rate makes it easy to estimate the amount of air streaming around the condenser on the side and the “depth”, to which it penetrates. As seen from the figure, the greater the distance between the heat exchangers, the greater the amount of air bypassing the condenser. These results explain why the air velocity decreases at the edges of the condenser (Figure 3), and grows at the edges of the radiator (Figure 4).
In the vertical plane, the condenser with respect to the radiator is set slightly symmetrical, the height of the condenser core is 20 mm less than the radiator core and is shifted down from the axis by 7 mm. The upper part of the heat exchangers was selected as the control variant for the analysis. The flow pattern in this case is similar to the one that was observed in the horizontal plane (Figure 6). The difference lies in the fact that the upper edge of the condenser is 13 mm lower than the radiator, therefore the flow rate in this case is greater and the “depth” of penetration from the edge of the radiator is greater.

In all three cases, the air flow passing through the radiator was the same, but the flow passing through the condenser was significantly different, with the gap between the condenser and radiator being equal to 16 mm the flow was 13.8% less, with the gap of 22 mm – it was less by 17.4%, with the gap of 28 mm – it was less by 21.4%.

Similar patterns were observed in the case when the cooling and air conditioning systems fan was used for cooling.

The research helps to solve another important problem. In analytical calculations, a condenser and a radiator are considered as series resistances, which seems logical because they are arranged sequentially one after the other. In such calculations, it turns out that the total resistance of the heat exchangers is overestimated, since they are not separated from each other at external boundaries and part of the flow goes past the condenser. In such cases, the air flow passing through the condenser can be taken into account using a coefficient of total air flow.
4. Conclusion
1. The flow rate of cooling air passing through the radiator of the cooling system decreases when a condenser is installed in front of it, due to an increase in the total resistance.

2. When the air conditioner is operating, the temperature of the air entering the radiator is higher than the ambient temperature. The condenser has a significant impact on the flow of air passing through the air path of the car. This factor has an additional effect on the efficiency of the cooling and air conditioning system. This is due to a change in the flow rate of cooling air passing through heat exchangers and an increase in the non-uniformity of the velocity field along their frontal surface.

3. The air flow passing through the condenser is significantly less than that through the radiator. This circumstance must be considered when calculating. Moreover, it must be borne in mind that this part of the air is not heated in the condenser.

4. The degree of the flow around the condenser depends on several factors, such as on the distance between the cores of the heat exchangers, on their relative frontal geometrical dimensions, on the resistance of the one and the other and on the external elements located near the heat exchangers.

References
[1] Thomas Schütz. Hucho – Aerodynamik des Automobils. Strömungsmechanik – Wärmetechnik – Fahrdynamik – Komfort. 6. vollständig neu überarbeitete und erweiterte Auflage mit 1171 Abbildungen und 49 Tabellen
[2] Williams J 2003 Aerodynamic Drag of Engine-Cooling Airflow with External Interference SAE 2003-01-0996
[3] Marion D’Hondt1 and Patrick Gilliéron 2010 Aerodynamic drag and flow rate through engine compartments of motor vehicles, 28th AIAA Applied Aerodynamics Conference (Chicago, Illinois)
[4] Marion D’Hondt 2010 Etude theorique, experimentale et numerique de l'ecoulement de refroidissement et de ses effets sur l'aerodynamique automobile Eng. Sci.
[5] Katz J.: Race Car Aerodynamics. Cambridge, Ma.: Bentley Publishers, 1995.
[6] Petrov A 2016 Effect of Inner Air Flow on the Aerodynamics of the Car Periodica Polytechnica Transportation Engineering doi: https://doi.org/10.3311/PPtr.10376
[7] Petrov A 2016 Factors linking the aerodynamics of the car with its internal aerodynamics Journal for automotive engineers 1(96) 8-11
[8] Teddy Hobeika 2015 On Tyre Rotation Modelling and Cooling Flow Measurements Thesis for the degree of Licentiate of Engineering in machine and vehicle Systems. Department of Applied Mechanics. Division of Vehicle Engineering and Autonomous Systems. (Chalmers University of Technology. Goteborg, Sweden)
[9] Pang S C, Kalam M A, Masjuki H H, Badruddin I A, Ramli R and Hazrat M A 2012 Underhood geometry modification and transient coolant temperature modelling for robust cooling networks Int. J. of Mechanical and Materials Eng. (IJMME) 7(3) 251-258
[10] Shome B, Kumar V, Kumar R, Arora G 2006 CFD Prediction to Optimize Front End Cooling Module of a Passenger Vehicle Int. Refrigeration and Air Conditioning Conf. 845