

CFD Simulations of one Period of a Louvered Fin where the Airflow is Inclined Relative to the Heat Exchanger

Lisa Henriksson
Chalmers Univ. of Technology

Peter Gullberg and Erik Dahl
Volvo Group Trucks Technology

Lennart Lofdahl
Chalmers Univ. of Technology

ABSTRACT

This article presents Computational Fluid Dynamics (CFD) simulations for one period of a louvered fin, for a crossflow compact finned heat exchanger, where the incoming airflow was inclined relative to its core. Four inclinations were investigated: 90°, which was when the air flowed perpendicular to the heat exchanger, 60°, 30° and 10° angles relative to the vertical plane. The study included three heat exchanger designs, where two of them had symmetrical louvered fins and a thickness of 19mm and 52mm. The third had a thickness of 19mm and had the louvers angled in one direction. All heat exchangers have been simulated when the airflow entered both from above and below relative to the horizontal plane. Simulations have also been carried out when the airflow entered from the side, illustrating the heat exchanger to be angled relative to the vertical axis. Two air speeds have been investigated for each configuration, where the results were compared to experimental data.

The results showed that the airflow characteristics were strongly dependent on the inclination angle. A more inclined heat exchanger generated larger separated areas at the entrance of the heat exchanger core. After approximately six louvers the airflow was fairly similar for all inclinations. The pressure drop over the core was not affected by the direction of the entered airflow. It was also seen that the heat exchanger with the louvers in only one direction resulted in approximately the same pressure drop as the one with symmetrical louvered fins. For a constant velocity in the longitudinal direction within the core, the pressure drop did not vary between the inclinations. Depending on the definition of the heat exchanger arrangement the heat transfer rate was affected. Simulated pressure drop followed the same trend as experimental data, even though the values were over-predicted.

CITATION: Henriksson, L., Gullberg, P., Dahl, E., and Lofdahl, L., "CFD Simulations of one Period of a Louvered Fin where the Airflow is Inclined Relative to the Heat Exchanger," *SAE Int. J. Engines* 8(4):2015, doi:10.4271/2015-01-1656.

INTRODUCTION

An increased cooling demand is seen for medium and heavy-duty trucks as a consequence of increased engine power, together with additional heat sources, such as Exhaust Gas Recirculation (EGR), Charge Air Cooling (CAC) and Waste Heat Recovery (WHR). To be able to meet upcoming emission regulations and market requirements, additional systems will be required and a further increase in cooling demand will be seen. To fulfill the cooling demand increased efficiency of the cooling device or an increased size of the cooling package is required. Due to limitations of space at the front of the vehicle, alternative positions of extra heat exchangers have to be evaluated.

There are vehicles today that have the heat exchanger located where the airflow is not necessarily perpendicular to it. For example some busses have their cooling air intake positioned on the roof or at the side of the vehicle. Formula one racing cars have the heat exchanger

angled within the cooling air intake ducts. For some sport vehicles air intakes are positioned at the rear of the vehicle, behind the C-pillars as well as in front of the front wheels. Alternative cooling module positions have been investigated by for example [1, 2, 3]. A common feature for these is that the oncoming airflow is not perpendicular to the heat exchanger core. Evaluation of inclined airflows relative to the heat exchanger core must therefore be performed; to be able to understand the effects on pressure drop, heat transfer relations and airflow behaviour.

This study is part of a larger project where experiments have been performed for full-size heat exchangers, mounted at a number of angles relative to the incoming airflow. CFD simulations of the experimental set-up have also been performed. Previous work has generated a method for simulating one period of a fin, carried out by Henriksson et al. [4]. There are a number of investigations based on simulations of

louvered fins, for example [5, 6, 7, 8]. Chang et al. [5] carried out a CFD study including thermal effects for a period of a louvered fin of two different designs. Moreover, Junjanna et al. [6] carried out a study of the temperature within a louvered finned heat exchanger, as well as analyzing the airflow behaviour. Another work was performed by Achaichia and Cowell [7], which was one of the first attempts to simulate two-dimensional louvered airways. Yuan et al. [8] performed both experiments and CFD simulations to study flow field and heat transfer rates. The works presented in these studies did not consider the effect of angling the heat exchanger relative to the airflow.

The aim of this work was to understand the characteristics of the airflow when it was angled relative to the heat exchanger core, by using CFD simulations on one period of a fin. A second aim was to identify variations in static pressure drops between the configurations. The pressure drop results from the simulations were compared to experimental data. The focus has been on isothermal conditions, with additional information for thermal simulations.

HEAT EXCHANGER GEOMETRY

A compact louvered heat exchanger consists of louvered fins, where the air passes through and there are tubes in between, wherein the coolant is flowing. The fins are louvered in order to break up the boundary layer, to increase the heat transfer rate. As a consequence of increased mixing of the air the static pressure drop through the core will also increase. In Figure 1, a section of a heat exchanger core with tubes and fins is shown.

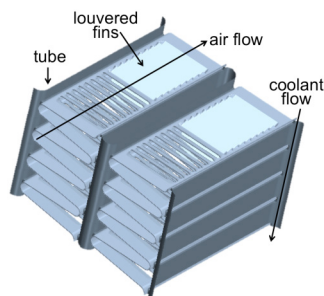


Figure 1. Louvered fins and tubes for a section of a heat exchanger core.

The simulated heat exchanger designs were based on the ones used for the experiments. Three heat exchanger geometries were tested as well as simulated. Two of them had a thickness of 19mm and the third a thickness of 52mm. One of the 19mm and the 52mm had symmetrically louvered fins. The third heat exchanger was specially made and had the louvers angled in only one direction. In Figure 2 one period of a louvered fin of the investigated heat exchanger cores can be seen. At each side of the fin there are tubes, where the coolant medium flows.

A compact heat exchanger is defined by a number of parameters, presented for the studied components in Figure 3. The top picture in Figure 3 represents the centreline plane through the core and the second picture shows the inlet.

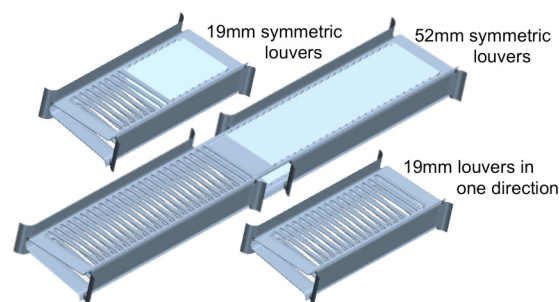


Figure 2. One period of a louvered fin for the three investigated heat exchangers.

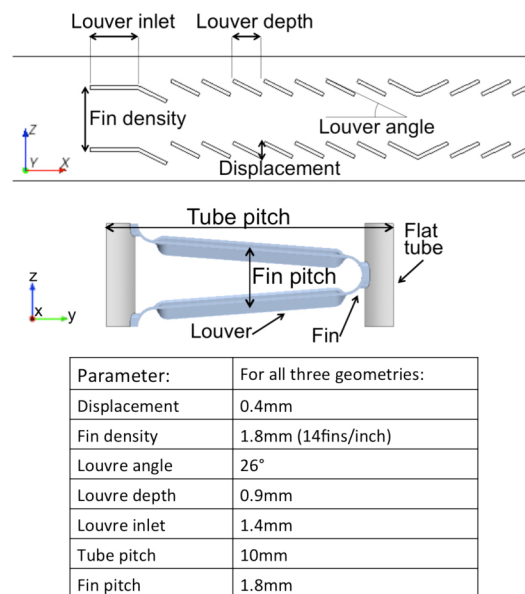


Figure 3. Parameter definition of the studied heat exchangers.

Four entrance angles of the incoming airflow, relative to the heat exchangers were investigated. The angles were defined from the vertical plane, hence the 90° case was when the airflow was perpendicular to the core. The additional three angles were defined as: 60°, 30° and 10°, where the 10° case had the most inclined airflow relative to the heat exchanger. Since the airflow characteristics will be affected differently depending on the entrance direction, both an entrance direction parallel and non-parallel to the louver angle have been simulated. A negative sign represents simulations where the entrance airflow was parallel to the louver angle.

For vehicle applications the heat exchanger can either be mounted with the tanks at the top/bottom or at the sides of the heat exchanger core. For the first arrangement the tubes were in a vertical direction, while they were horizontal for the second orientation. The airflow will most likely behave differently depending on which orientation that is evaluated during angling. This study was mostly based on the first arrangement, defined as downflow, but some comparison was also made to the second orientation, defined as crossflow.

For evaluating the results the centreline plane was used as a visualization plane. This plane is shown for all three heat exchangers in Figure 4.

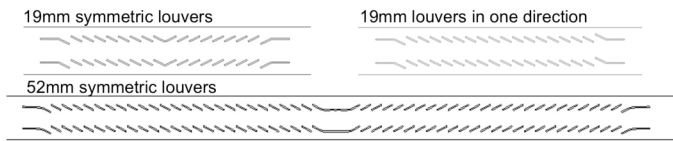


Figure 4. The centreline plan for all three heat exchangers.

SIMULATION SET-UP

The simulations were performed using CFD, where the software STAR-CCM+, version 9, from CD-Adapco, was used for building the simulation models, applied as the solver, and for the post-processing. The initial surface meshing and geometry modifications of the CAD geometry were performed in the software ANSA, from BetaCAE.

The method used for these simulations was based on the conclusions from a previous article [4], which defined a simulation method for simulations of a period of a louvered fin. Previous studies, performed by Beamer et al. [9] have shown that the airflow is periodic over one period of a louvered fin. Hence, the computational domain was defined as in Figure 5, which shows the 19mm thick heat exchanger. The figure also visualizes the definition of the centreline plane.

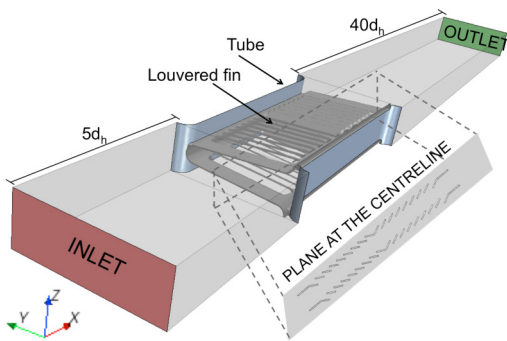


Figure 5. Computational domain.

The length of the upstream inlet duct was set to five times the hydraulic diameter, d_h , and the downstream outlet duct was set to $40d_h$. The hydraulic diameter was defined as in Equation 1 [10], where A was the cross-sectional area between the fin and the tube and P was the wetted perimeter around the fin and the tube, see lower picture in Figure 3.

$$d_h = \frac{4 \cdot A}{P} \quad (1)$$

The boundary condition at the inlet was defined as Velocity Inlet and the outlet was defined as Pressure Outlet, marked in red and green, in Figure 5. The solids of the model were defined as walls. All other duct walls were defined as periodic, assuming an infinite heat exchanger core. The periodicity boundary condition was only valid for the central part of the core and not near its edge, where the flow phenomenon will behave differently.

The experiments were arranged to have an upstream duct, in front of the heat exchanger, with a constant width and height. Applying the same velocity of the airflows resulted in a lower velocity in the core

for a more inclined heat exchanger, as a result of the increased core area. To simulate the angling effect of the heat exchanger relative to the airflow, the inlet velocity vector was divided into two components. For the downflow orientations the velocity components varied in the x- and z-direction, while for the crossflow arrangement it varied in the x- and y-direction. The coordinate system is defined in Figure 5. The resulting vector was the same for all cases, while the magnitude in the two directions varied. Two air velocity vectors at the inlet duct were simulated; 9.83m/s and 3.87m/s, corresponding to a Reynolds number of 1750 and 700, defined with the hydraulic diameter. By using the louver depth instead as the characteristic length the Reynolds numbers were 600 and 250 respectively. For the $\pm 30^\circ$ angled cases the x-component was +4.92m/s and the z-component was ± 8.52 m/s for the higher air velocity and +1.94m/s and ± 3.35 m/s for the lower velocity. From experiments at Volvo GTT [11] it was known that the air velocity in front of the radiator was approximately 2m/s when driving at 90km/h without any fan operation. For a more critical driving condition, occurring at lower vehicle speeds and when the fan is at maximum speed the corresponding velocity was approximately 9m/s for a heavy truck.

A third velocity was simulated for the 19mm and 52mm thick heat exchanger with symmetrical louvered fins. For this set-up the x-component of the velocity was constant for all heat exchangers and was defined as 4.92m/s, while the z-component was varied to obtain the correct angle of the airflow, relative to the core. These simulations were performed in order to detect areas of losses in the airflow, for the different angles, with a constant mass airflow through the fin.

The mesh settings were the same for all cases, independent of the airflow entrance angle. Using the trimmer mesh function generated the volume mesh, which resulted in a mesh of predominantly hexahedral cells with polyhedral cells nearest the prismatic layers at the surface. The boundary layer was fully resolved and the two-layer all y+ wall treatment was used as wall function, resulting in a y+ value below 1. 9 prismatic layers were used at the surface. Refinement volumes were located at the louvered fin and at the base, with a cell size of 0.05mm. The resulting volume mesh for the 19mm thick heat exchanger was ~32 million cells. In Figure 6 the mesh resolution is shown at the centreline plane along the computational domain to the left, and at a plane across the domain, at a yz-plane, to the right.

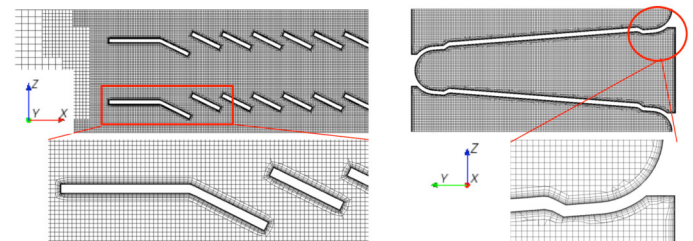


Figure 6. Mesh resolution at the centreline plane and at the yz-plane across the computational domain.

From the conclusions in [4] recommendations were drawn that these kinds of simulations are sufficiently defined as steady state. It was seen that the airflow was laminar within the core and turbulence was created behind it. The airflow was defined by the Reynolds number.