

Numerical and Experimental Investigation of Thermal Conditions Inside the Engine Compartment of Snowmobiles

Johannes Wurm
Graz University of Technology

Matthias Fitl and Michael Gumpesberger
BRP-Powertrain GmbH & Co KG

Esa Väisänen
BRP Finland Oy

Christoph Hochenauer
Graz University of Technology

ABSTRACT

Nowadays, investigating underhood airflow by using numerical simulation is a standard task in the development process of passenger cars and commercial vehicles. Numerous publications exist which deal with simulating the airflow through the engine compartment of road vehicles. However, hardly anything can be found which deals with off-road vehicles and nothing exists which focuses on snowmobiles. In the presented paper the airflow and the thermal conditions inside the engine compartment of a snowmobile are investigated by the usage of computational fluid dynamics (CFD) as well as experimental methods. Field tests at arctic conditions have been conducted on a serial snowmobile to measure temperatures inside the compartment and to gain realistic boundary conditions for the numerical simulation. Thermocouples (type K) were attached under the hood to measure exhaust, air, coolant and surface temperatures of several components at previously defined load cases. Wind tunnel tests have been performed to examine the external airflow. Hence a direct comparison between simulation and experiment is possible and excellent agreement can be achieved. State-of-the-art CFD methods for simulating underhood airflow are applied in this investigation. The modelling process and the various numerical methods which are necessary to take all effects of underhood airflow into account are described in detail. The structure of the airflow and the component temperatures are analyzed to identify hot spots and recirculation zones influencing the system's cooling performance and bear the risk of thermal failure. Proposals to decrease the time-consuming modelling process and improvements for fast simulation set-up are made.

CITATION: Wurm, J., Fitl, M., Gumpesberger, M., Väisänen, E. et al., "Numerical and Experimental Investigation of Thermal Conditions Inside the Engine Compartment of Snowmobiles," *SAE Int. J. Commer. Veh.* 8(1):2015, doi:10.4271/2015-01-9017.

INTRODUCTION

Engine compartment packaging has become a key factor in the vehicle design process. The complexity and functionality of underhood components of passenger cars or commercial vehicles are continuously rising. The structure of the underhood flow is mainly defined by the vehicle's compartment and front end design. Due to the increasing power of combustion engines and tight packaging, providing sufficient cooling airflow proves to be a difficult task. To overcome these challenges, investigating the thermal conditions under the hood has become a major issue. Numerical flow simulations have proven their strength to predict aerodynamic but also thermal behavior of new developments and to reduce the product's time to market. Hence the integration of computational fluid

dynamics (CFD) analysis into the development process is state of the art in the automotive industry. In smaller sections like snowmobiles CFD is just getting started to be used in the development process.

The evaluation of new designs at a very early stage and an increase in the efficiency of prototype testing are the driving factors to implement CFD into the development process. In the literature, numerous approaches can be found varying in method and scope due to the complexity and variety of aspects influencing the underhood airflow. In [1] a thermal model is integrated into the aerodynamic design process and the applied method is extensively studied in terms of the correct modelling of the turbulent and thermal boundary layer. The necessary steps for conducting an under hood compartment (UHC) airflow analyses are described in detail in [2].

Whereas most CFD codes are based on the Navier-Stokes equations an alternative for airflow simulations is offered by solving the lattice Boltzmann equations (LBE) as it is shown in [5, 6, 7]. One of the main benefits is that it only requires a surface mesh and the volume discretization is completely automatic. In [8] the commercial code *Powerflow*, which is using the lattice Boltzmann theory, is applied to compute the external and underhood airflow. The target is to evaluate the velocity and pressure distribution on the vehicle inlets and outlets. In a second step the temperature and velocity fields inside the compartment are computed with a different CFD code using the vehicle inlet velocity distribution from the previous simulation. However, a huge disadvantage of the LEB approach is that simulations are computationally expensive. Moreover, only inherently transient simulations can be conducted.

Various publications focus on the assessment of underhood cooling airflow only and thermal effects are totally neglected. By evaluating pressure and velocity distributions inside the compartment, the efficiency of the cooling airflow can be predicted [9, 10, 11]. The advantage of this approach is that computational time can be drastically reduced due to the fact that it is not necessary to take the energy equation into account. However, it must be mentioned that this approach is only reasonable if forced convection is the main driving factor for the underhood airflow. In [4] CFD is used in terms of studying the cooling module performance. Coupling approaches to evaluate the interaction of complex 3D cooling airflow with simplified 1D coolant liquid flow can be found in [12, 10]. Another common approach is to compute the flow field and the air temperature by means of CFD and create an iterative process where surface component temperatures are computed separately [13, 14, 15]. The possibility of taking the effects of radiation into account by using the Surface to Surface (S2S) method is presented in [16] and has been validated in [17]. The effects of different turbulence models on the external airflow around a simplified car body have been investigated in [18].

Whereas forced convective airflow is the responsible factor for cooling when the vehicle is moving, buoyancy driven flow becomes relevant when the vehicle stops. Nowadays, developing strategies to simulate the heat soak phase, which follows after the engine is switched off, is the current field of research. Due to the fact that in this case the flow quantities change with time, setting up a transient simulation is necessary which is very time-consuming. Experiments and numerical simulations for geometrically simplified engine compartments can be found in [19, 20, 21].

Reducing the necessary time until a converged solution is reached is of great interest for the industry. In [22] the advantages of a flow network modelling (FNM) approach are presented. Its principle idea is to break the entire flow domain into various airflow regions. An innovative method to compute underhood compartment airflow is introduced in [23].

In the presented paper the airflow and the thermal conditions inside the engine compartment of a snowmobile are analyzed for the first time using state-of-the-art CFD modelling methods. Critical load cases are defined and the temperature distributions, as well as the velocity fields under the hood, are evaluated to detect hot spots and recirculation zones. Moreover, the efficiency of the air cooler

mounted in the front end of the sled in dependency of fan activity is investigated. To ensure the significance of the simulation, it is necessary to set appropriate boundary conditions. Therefore, wind tunnel tests, experiments on a component test bench, as well as winter field tests have been conducted to confirm the quality of the simulation results. Simulation settings have been defined according to the conducted tests; hence the computed results reflect realistic driving situations. Furthermore, the test data are used to validate the accuracy of the numerical solution. In the simulation, the effects of radiation are also considered by applying the Surface to Surface (S2S) radiation model. The transport equations for computing radiation are solved simultaneously when calculating mass, momentum and energy equation, thus the time-taking coupling process of different software can be avoided. Furthermore, simulations using the Discrete Ordinate (DO) model have been used to investigate the influence of radiation modelling. The purpose of the study was to confirm that CFD is an important tool to virtually investigate the thermal and airflow conditions under the hood of snowmobiles and to develop ideas for optimizations.

Load Cases

Driving constantly at low speed whilst pulling a trailer on packed snow or icy trails are the most crucial but still common conditions a snowmobile is used for. Therefore, driving at 30 km/h and high engine loads is defined to be the first investigation point in the simulation. Figure 1 shows the test vehicle which is a touring sled equipped with a 90 hp 4- stroke engine. Moreover, the trailer which is necessary to adjust the engine load can be seen. Secondly, constant driving at 70 km/h should be simulated as an oppositional load case. In both cases convective airflow is the main driving factor for heat transfer under the hood.

EXPERIMENTAL SETUP

The accuracy of numerical simulations mainly depends on the quality of the defined boundary conditions. Moreover, empirical values are necessary for verifying the computed solutions. Experiments inside a wind tunnel, winter field tests, as well as tests of single components have all been conducted to ensure the quality of the CFD simulations. In [24], temperature and heat flux measurement methods inside a passenger car's engine compartment are presented. Further information on how to study the complex heat transfer behavior occurring under the hood is given in [25, 26], where different phases of a driving cycle are examined.



Figure 1. Test vehicle inclusive test trailer used to increase engine load

Wind Tunnel Tests

Wind tunnel tests were performed in a previous study aiming on simulating the external airflow around the sled. Due to the fact that the same model was in use the results gained from these tests can be directly taken for evaluation. Figure 2 shows a picture of the wind tunnel testing where a smoke generator was taken to visualize the external airflow. Static wall pressure was measured at different positions and those results are taken as reference values for this publication.



Figure 2. Using smoke to visualize external airflow in wind tunnel testing

Field Tests

Surface temperatures of various components, including engine, muffler, exhaust pipe, air cooler, air box and CVT have been measured. Furthermore, the air temperature at various positions has been monitored. Temperature sensors were also attached inside the exhaust pipe as well as inside the cooling system to measure the exhaust gas and coolant liquid temperature, respectively. In total, 32 thermocouples (type K) were connected to μ -THERMO K8 measurement modules. The results were stored on an ISAAC data logger. A sampling rate of 1 Hz has been defined for each sensor. Moreover, data monitored by the ECU has been recorded. Thus, information about vehicle speed, RPM, engine temperature and throttle position is available and the engine load can be reconstructed on engine test benches.

To provide sufficient testing data, numerous test drives have been performed. Therefore, it was possible to monitor the thermal conditions inside the engine compartment at the previously described load points. A single test run was defined to last for 10 minutes whilst driving with constant speed, which is adequate to reach stable conditions under the hood. Computing a time averaged value from the last two minutes of each test helps to receive more feasible data. Moreover, averaging the results of all test runs undertaken with the same vehicle speed and comparing the deviation of the individual tests with the computed mean average value shows, that stationary measurement results have been gained. These values are taken as boundary condition or reference data for the CFD simulations.

However, fluctuations of the test results caused by a variety of outer influences could be observed. These outer impacts, e.g., wind speed, wind direction or ambient temperature, do influence the measurements. Additionally, slight deviations of engine load and driving speed due to manual controlling need to be regarded. Figure 3

shows the resulting temperature of 3 components under the hood at six different test runs. All of them were carried out with 70 km/h driving speed but on different days and time of days, respectively.

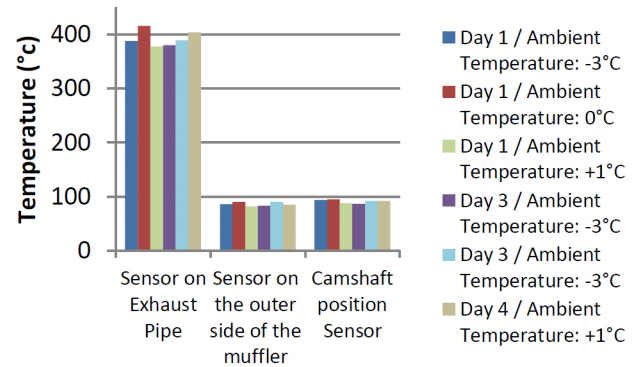


Figure 3. Influence of external aspects on measurements

It can be stated that the measured temperature deviations are minor. Thus the mean average value is acceptable to be used as input value for the CFD simulations as well as for validation purpose.

Nonetheless, the huge advantage of field tests is that especially in snowmobiling the trail conditions have a great influence on the thermal conditions inside the compartment. Depending on the snow conditions on the track, the cooling works better or is less efficient and it is hardly possible to simulate real driving conditions in a wind tunnel. Therefore, the simulation can be initialized with far more realistic boundary conditions when using values resulting from field tests rather than by using values gained from wind tunnel experiments.

Component Tests

Pressure drop measurements were necessary to provide accurate data for modelling air cooler and air intake grids. Therefore, a test rig has been built as shown in Figure 4. The volume flow through the wind tunnel with the following dimensions $L \times W \times H = 500 \text{ mm} \times 300 \text{ mm} \times 240 \text{ mm}$ is regulated by a mass flow sensor.

Tests have been performed at two different volume flow rates to ensure the significance of the results. The static pressure right before and 40 mm behind the tested component has been monitored enabling the computation of the resulting pressure loss. Moreover, the velocity distribution behind the air intake grid has been measured at different positions. Both, static pressure and velocity distribution are compared with the numerical results.

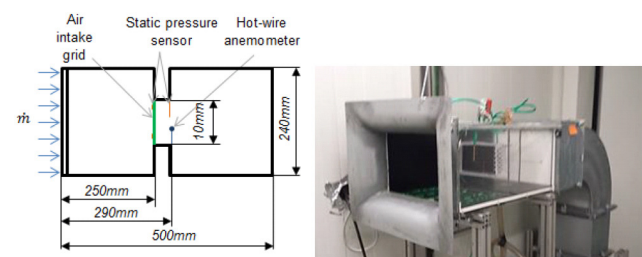


Figure 4. left: Sketch of the component test bench to illustrate the porous layer settings; right: Component test bench

NUMERICAL SIMULATION

Computational Model

Accurate computation of the airflow going through the engine compartment of a vehicle also demands the computation of the external airflow. A rectangular shaped virtual wind tunnel (VWT), shown in [Figure 6](#), is designed with the following dimensions: $L \times W \times H = 30 \times 11.5 \times 11$ m. The CAD model of the sled is positioned 10 m behind the inlet of the VWT. For the simulation all components under the hood, e.g., engine, airbox, muffler, gearbox, exhaust pipes, CVT, parts of the frame, cooling system, radiator and fan are modelled. The hood covering these components is modelled as shell surface. Further parts affecting the external airflow and hence the flow through the engine compartment like ski, front suspension, track and a manikin in an upright position, are also taken into account. [Figure 5](#) shows the geometrically simplified model of the sled including the driver. A transparent hood enables an insight into the engine compartment.

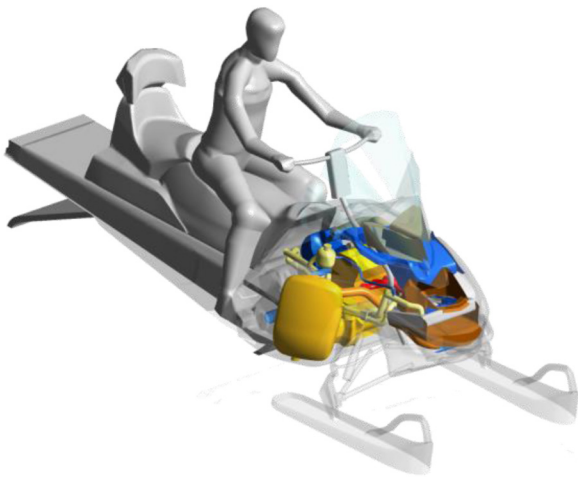


Figure 5. CAD- Model of the snowmobile including components inside the engine compartment

For the mesh generation process, several refinement boxes are defined to control the mesh size in specific regions. Especially in regions where part design has a large influence on the flow, i.e., inside the engine compartment but also in the near wake region, the cell size is limited.

In a first step a triangular surface mesh is generated. From the surface mesh thin prism layer cells can be extruded. They are necessary for a proper computation of the turbulence and the heat transfer in wall near regions. Finally, the remaining volume is filled with tetrahedrons in the sled vicinity and in the outer regions the numerical domain is spatially discretized by defining hexahedral elements. The whole 3D mesh consists of approximately 11 million volume cells. [Figure 6](#) shows the generated mesh when cutting through the middle plane of the domain.

Turbulence Modelling

For the numerical computation of the flow field it is necessary to consider the effects of turbulence. Therefore, the so called *Realizable k-ε*-model has been used which belongs to the category of 2-equation models. Apart from the governing equations for continuity and momentum, two further transport equations, one for the turbulent kinetic energy (k) and another for the turbulent dissipation rate (ϵ), are solved at each iteration. The advantage of the *Realizable k-ε*-model is that it is a very stable and time-saving method. Its applicability for external flow simulations is confirmed by [18] and suggested by [27]. For the computation, a pressure based (segregated) solver is used, where the governing equations are solved sequentially. The settings for the spatial discretization of the various transport equations and the defined under-relaxation factors are shown in [Table 1](#).

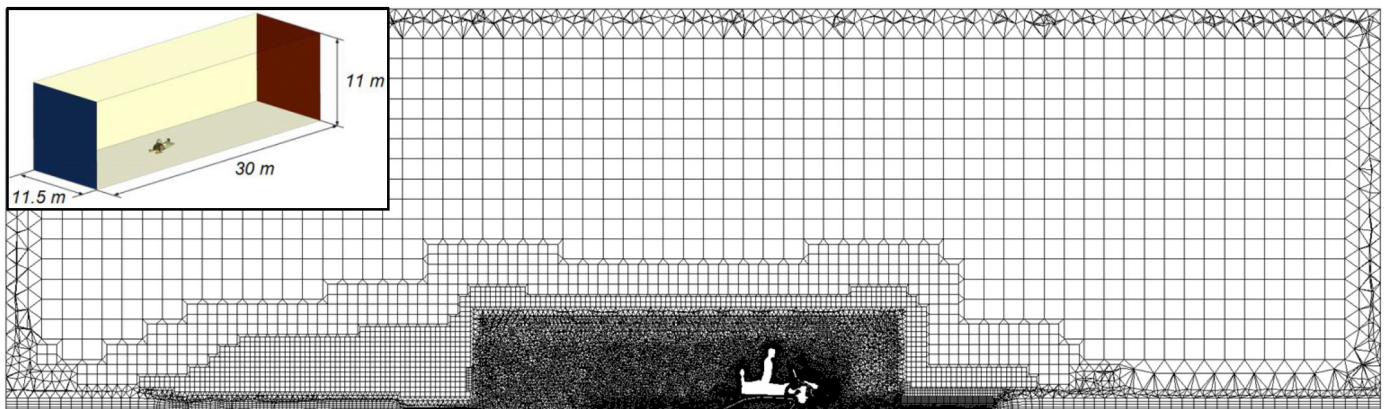


Figure 6. Virtual Wind tunnel and defined mesh shown in the longitudinal cutting plane