

Advanced Heat Transfer and Underhood Airflow Investigation with Focus on Continuously Variable Transmission (CVT) of Snowmobiles

Johannes Wurm
Graz University of Technology

Eetu Hurtig, Esa Väisänen, and Joonas Mähönen
BRP Finland Oy

Christoph Hochenauer
Graz University of Technology

ABSTRACT

The presented paper focuses on the computation of heat transfer related to continuously variable transmissions (CVTs). High temperatures are critical for the highly loaded rubber belts and reduce their lifetime significantly. Hence, a sufficient cooling system is inevitable. A numerical tool which is capable of predicting surface heat transfer and maximum temperatures is of high importance for concept design studies. Computational Fluid Dynamics (CFD) is a suitable method to carry out this task.

In this work, a time efficient and accurate simulation strategy is developed to model the complexity of a CVT. The validity of the technique used is underlined by field measurements. Tests have been carried out on a snowmobile CVT, where component temperatures, air temperatures in the CVT vicinity and engine data have been monitored. A corresponding CAD model has been created and the boundary conditions were set according to the testing conditions. In a first step a simplified study is presented, to gain basic knowledge about the system, followed by a full underhood airflow simulation. The modelling process is presented in detail and necessary adaptations are identified.

The results show, that the numerical model is able to predict the surface temperatures within a range of 5 % for different load cases. Thus, the developed method is validated and can be used for future development processes. The influence of the pulley design on the underhood airflow will be evaluated to identify optimization potential. Moreover, the presented work is the first of its kind, where a numerical heat transfer simulation of a CVT is compared to field tests carried out on snow.

CITATION: Wurm, J., Hurtig, E., Väisänen, E., Mähönen, J. et al., "Advanced Heat Transfer and Underhood Airflow Investigation with Focus on Continuously Variable Transmission (CVT) of Snowmobiles," *SAE Int. J. Commer. Veh.* 10(2):2017, doi:10.4271/2017-01-9180.

INTRODUCTION

CVTs are widely spread in the recreational vehicle industry like all-terrain-vehicles (ATVs) or snowmobiles. Also scooters, some cars or tractors use a continuously variable transmission. Basically it must be differed between belt and chain driven CVTs. A review, where both systems have been examined, has been published by Srivastava and Haque [1]. The transmission efficiency is a key parameter and has been under investigation for a long time [2, 3]. Reasons for the power loss have already been identified in earlier researches. E.g. Gerbert [4] distinguish between losses due to external friction like sliding between belt and pulley and the losses due to internal friction like the sliding between molecules-hysteresis. Moreover, the impact

of small pulley radii and belt wedge angle on the power losses have been studied in [5, 6]. Further experimental studies which focus on improving the transmission efficiencies of rubber V-belt CVTs have been conducted by Chen et al. [7, 8]. With increasing computational power, numerical modelling of the dynamic behavior becomes more and more important. Basic investigations, which focus on steady state rubber V-belt CVTs analytically, have been conducted by Gerbert [9, 10], Dolan and Worley [11] and Sorge [12]. Further researches focused on the transient case [13] and analytical approximations have been found e.g. [14, 15].

A recently developed model, capable of predicting the power losses of a rubber V-belt CVT, has been published by Bertini et al. [16]. In their research, different effects which reduce the

efficiency are discussed. Julió and Plante [17] present a model which is able to predict the transmission ratio time response when the conditions on the drive pulley are changing. Their model is sufficiently validated by experiments. This is especially interesting for CVT's using electro-mechanical control devices for modulating the axial forces applied on the drive pulley [18]. This allows a separation of the transmission ratio from the engine speed and torque. Current researches focus on the impact on the fuel economy of these systems and the results look quite promising to increase the efficiency of CVTs. Especially in vehicles with high engine power the amount of kinetic energy which is transformed into heat is substantial. As a result, the thermal loads on the components increase drastically. Particularly the belt, which is often made out of rubber, suffers under high temperature peaks and its life span reduces significantly.

Therefore, an efficient cooling system must be guaranteed. This is already a delicate task for open systems but if the CVT is enclosed by a casing the conditions get even worse. In ATVs the CVT needs to be protected from dust, mud and water to avoid belt slipping, hence a cover is built around which makes sufficient cooling airflow and intelligent component design indispensable.

Whereas the dynamics of a CVT are well understood and the reasons for the occurring power losses are known, hardly any studies are focusing on the thermal aspect. Therefore, detailed investigations of heat transfer effects are emphasized in the current literature. Experiments have been carried out in [19] and [20] where the temperatures inside a CVT housing were measured. Main scope was to increase the convective heat transfer to reduce the temperature level by redesigning the casing. Additionally, numerical tools are used to compute the flow field inside the domain. Karthikeyan et al. [21] use CFD to reduce dust ingress into a CVT housing. For scooters, a design study to improve the airflow through the housing has been published, see [22]. Both works use state-of-the-art numerical settings. However, important physical aspects like belt motion and temperature equalization are neglected. In this work a heat transfer analyses of a CVT which is built into a snowmobile is carried out. The main objective is to improve the cooling rate and reduce peak temperatures. CFD is used to compute component surface temperatures and the corresponding flow field under the hood of the vehicle. In a first step a simulation model of a rotating disk inside a flow channel is created. It is used to investigate the effect of various parameters like mass flow and design of the disk. According to this study, two novel pulley designs have been produced and are tested in the field. In compliance with the experiments, a CFD Simulation is performed taking the whole sled inclusive all components under the hood into account. The focus of this simulation is laid on the CVT region. A previous study, where a novel numerical model is developed to simulate conjugated heat transfer in relation to CVTs is presented in [23]. The method has been adopted for the current work.

SIMPLIFIED CASE STUDY

It can be assumed that the power loss of a CVT is invariant for a constant load case and therefore the lost kinetic energy is continuously transformed into heat. In the following case study, it is assumed that heat is generated in the core of the belt and transferred to the pulleys in the contact zone between belt and pulley. Concurrent the pulleys are cooled by forced convection which is created by its own rotation and the external airflow. According to Newton's law of forced convection, see [equation \(1\)](#), for a constant thermal energy $\dot{Q}$ the wall temperature can either be reduced by increasing the heat transfer coefficient, denoted as h , or by increasing the surface area A .

$$\dot{Q} = h \cdot A \cdot (T_{wall} - T_{\infty}) \quad (1)$$

For completeness, another option would be the reduction of the fluid temperature T_{∞} . However, as ambient air is used for cooling, its temperature is predetermined and the effort to cool it down is not feasible. For the simplified case study only one part of a pulley is used which is positioned in the middle of a virtual wind tunnel (VWT), compare [Figure 1](#). On one side of the flow channel a velocity inlet is defined where the airflow enters with a defined speed. Opposite, a pressure outlet region is defined where the air exits the computational domain. Blades can be found on one side of the disk whereas the other side is cone-shaped. The pulley is made out of an aluminum alloy with a thermal conductivity of 120 W/mK and has a diameter of 0.25 m . To induce motion in a steady-state simulation, the Moving-Reference-Frame Method (MRF) has been used. Therefore, a region enclosing the moving side of the pulley needs to be separated from the main flow domain. A non-inertial reference coordinate system is created for this region where rotational settings can be specified. On the right side of [Figure 1](#) a ring section is separated from the surface which can be regarded as the contact area between belt and pulley. In this section a constant energy source term is defined to induce heat into the system. For a conjugated heat transfer (CHT) analyses the solid and the fluid region are both meshed. Due to the simple geometry the spatial discretization can be very fine. The total amount of tetrahedral cells is 1.5 million plus 5 million prism cells which were extruded from the previously generated surface mesh. To ensure a high quality mesh, the cell skewness is below 0.85 and the maximum wall near aspect ratio is restricted to 5. Turbulence is modelled by using the realizable k - ϵ -model. An overview of further numerical settings is given in [Table 1](#).

Table 1. Simulation settings

Solver	Implicit Pressure based (segregated)
Time	Steady-state
Turbulence model	Realizable k - ϵ - Model
Near wall treatment	Enhanced wall treatment
Motion	MRF

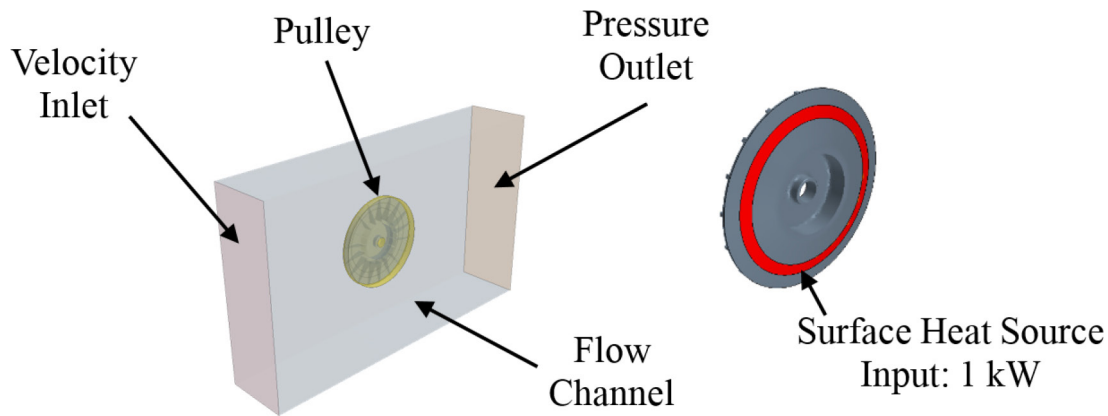


Figure 1. Simplified model to optimize heat transfer

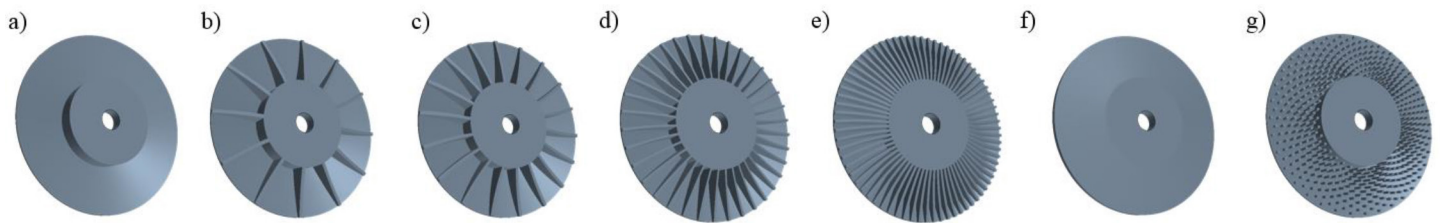


Figure 2. CAD Models of the seven different pulleys: a) Thin pulley; b) Pulley with 12 blades; c) Pulley with 18 blades; d) Pulley with 36 blades; e) Pulley with 72 blades; f) Thick pulley; g) Pulley with nubs

VARIANTS

To evaluate the impact of geometrical changes of the disk as well as the influence of an increased heat transfer coefficient, various simulations have been performed. On the one hand the flow parameters have been varied. The velocity of the incoming airflow is increased from 1 m/s to 5 m/s and finally to 20 m/s. These values correspond to an inlet mass flow rate of $\dot{m}_1 = 0.14$ kg/s, $\dot{m}_2 = 0.7$ kg/s and $\dot{m}_3 = 2.8$ kg/s, respectively. The rotational speed of the pulley was set to 3500 rpm. Additionally, a further simulation at a constant inlet velocity of 20 m/s but doubled rotational speed has been conducted.

Table 2. Matrix of test cases

Mass flow rate	$\dot{m}_1$	$\dot{m}_2$	$\dot{m}_3$	$\dot{m}_3$
Rotation Rate	3500 rpm	3500 rpm	3500 rpm	7000 rpm
Thin Pulley	-	-	•	•
Pulleys with 12 blades	-	-	•	•
Pulleys with 18 blades	•	•	•	•
Pulleys with 36 blades	-	-	•	•
Pulleys with 72 blades	-	-	•	•
Thick pulley	-	-	•	•
Pulleys with nubs	-	-	•	•

For all these cases a pulley with 18 blades has been used, compare Figure 2 - pulley c. On the other hand, the pulley geometry has been changed. In total, seven different disks have been tested at two

different rotational speeds, namely at 3500 rpm and 7000 rpm. A CAD model of each geometry is shown in Figure 2. The number of blades has been varied from zero to a thick pulley. Moreover, a pulley with nubs has been designed with the sole function of increasing the effective area. A summary of all test cases can be found in Table 2. All cases marked with a black dot have been carried out.

RESULTS & DISCUSSION

Figure 3 shows the resulting surface averaged pulley temperature at the three previously defined inlet mass flow rates. The black line represents the average temperature in the ring section where the energy source term is defined and the grey line shows the overall surface temperature of the pulley. Obviously both lines follow the same trend.

Increasing the mass flow rate from 0.14 kg/s to 0.7 kg/s leads to a significant surface temperature reduction of 24 K. An additional quadruplication of the mass flow rate to 2.8 kg/s leads to a further temperature drop of 12 K. Hence it can be stated, that the heat flux from the solid to the fluid is significantly improved when Reynolds is between 25.000 and 125.000.

However, increasing the airflow rate to decrease the overall temperature becomes less efficient with increasing Reynolds number. The reason is that less time is available for heat exchange and thus, the positive effect flattens in relation to the additional mass flow. Taking the effort to generate the additional mass flow into account, an optimum can be derived.