REVIEW ON SIMULATION OF HEAT DISTRIBUTION OF CONTINUOUS VARIABLE TRANSMISSION (CVT)

Dinesh S[1], Sudarsan L[2], Sujith R[3], Yashwanth S[4]
[1]Assistant Professor, Department of Mechanical Engineering
[2,3,4]Students of Mechanical Engineering

Abstract

In this paper, Heat transfer in CVT rubber-belt is analysed. This paper covers the thermal loses due to friction between the rubber belt and pulley of CVT. The belt life suffers under increased thermal loads, also the overall efficiency of CVT reduces. A time efficient Heat transfer analysis for an enclosed CVT is done by introducing numerical model by using Computational Fluid Dynamics (CFD).The simulation is done using C++ toolbox called Open Foam v1806 by ESI. A block mesh is created, then the flow is stimulated using "cht Multi Region Foam" solver.

Introduction

Continuous Variable Transmission (CVT) is an automatic transmission that can change seamlessly through a continuous range of effective gear ratios. It has three major components comprising of the primary and secondary pulleys and the belt. The pulleys are made of two conical sheaves each. The primary pulley is the one connected to the crank shaft, also called Drive pulley. The secondary pulley also called as driven pulley is attached to the drive shaft. The belt slides over these sheaves and transmits the work from drive pulley to the driven pulley. The major advantage of the CVT system is that the transmission ratio can be changed without interrupting the torque output. CVTs are used in vehicle industry like all-terrain-vehicles or snowmobile, also scooters and even some cars work with a continuously variable transmission.

A review on belt and chain driven CVT systems examined, has been published by Srivastava and Haque [1]. The transmission efficiency is a key factor affecting the wide use of CVT and has been under investigation for a long time [2,3]. Earlier researches have already provided the various reasons for power losses. E.g. Gerbert [4] distinguish between losses due to internal friction and external friction. The impact of the radii of the small pulley and belt wedge angle on the power losses is provided in [5,6]. Chen and Sung conducted several experimental studies that focuses on improving rubber V-belt CVT's transmission efficiency. [7]. Numerical modelling of the dynamic behaviour becomes more important with increasing computational power. Gerbert [8,9], Dolan and Worley [10] and Sorge [11] conducted investigations that studeied analytically on V-belt CVTs under steady state condition. Researches that focuses on the transient condition [12] and analytical approximations have been studied e.g. [13,14]. Bertini et al published a journal capable of predicting the power losses of rubber V-belt CVT [15]. Julió and Plante's [16] model which is able to predict the transmission ratio time response when the conditions on the drive pulley are changing, validated by experiments. CVT 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. Julió and Plante [16] present a CVT dynamic model by using Newton's law of motion on a discretized belt which is sufficiently validated by experiments.To compute transmission ratio and expected power loss. 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. A review, where both systems have been examined, has been published by Srivastava and Haque[3].That is on both the belt and chain of continuous variable transmission system. The amount of kinetic energy which is transformed into heat is substantial in heavy vehicles with high power. Zhu et al. [18] have carried out an experimental investigation on power losses of a rubber belt CVT which is used in all snowmobiles. It points out that the efficiency of the transmission in CVT depends on the applied

load as well as on the engine rpm and amounts approximately 0.7.Then the efficiency is strongly influenced by the clamping forces as examined by Bonsen et al. [19]. Increasing the clamping pressure leads to a strong decrease in efficiency, however, a certain pressure is necessary to ensure safety and to avoid high losses due to slip condition. In this a metal push-belt is used and the

measured efficiency factor is between 0.85 and 0.94 as it strongly depends on the working conditions. van der Sluis et al. [20] presented a study on optimizing the efficiency of a metal push-belt CVT. The heat generated in CVT is huge and leads to high thermal loads. Particularly rubber belts, suffer from high temperature peaks and their life span reduces gradually. Therefore, efficient cooling should be done. This is already a delicate task for open systems but if the CVT is enclosed by a casing the conditions get even worse. In All Terrain Vehicles the CVT needs to be protected from dust, mud and water to avoid belt slipping, hence a casing is built around which makes sufficient cooling airflow and intelligent component design indispensable. Whereas the dynamics of a continuously variable transmission(CVT) are well understood and the reasons for the occurring power losses are known, hardly any studies are focusing on the thermal aspect. Small scale thermal analysis where simulation results are compared with experiments are presented by Junhui et al. [21,22]. Moreover, experiments have been carried out in [23,24] where the temperatures inside a CVT housing were measured already. Main scope is to increase the convective heat transfer to reduce the temperature level by redesigning the casing. Additionally, numerical tools are being used to compute the flow field inside the domain. Karthikeyan et al. [25] uses computational fluid dynamics (CFD) to reduce dust ingress into a CVT housing. For scooters, a design study to improve the airflow through the housing in CVT has been published, see [26]. Both works use state-of-the-art numerical settings but, important physical aspects like belt motion and temperature equalization were neglected. In the research study presented by Johannes Wurm and Mathias [34], the thermal effects within an enclosed CVT is studied. The design of a commercial available CVT system was taken and measurements were conducted on an engine test stand. The heat transfer and the flow conditions were computed using Computational Fluid Dynamics (CFD) resulting from the pulley rotation and the belt movement. STARCCM+, a state-of-the-art commercial software code. CFD codes were used to execute all simulations. The transient case was reduced to a quasi-steady-state case to reduce the computational effort . The moving reference frame (MRF) method was used to model motion in steady-state case. A novel method was developed in another work [27] by J.Wurm and Mathias were the computed flow is asymmetric and pulley cooling differs due to the presence of casing and its applicability is verified [34] paper. The target of the paper was to implement the rotation of the temperature profile to the state-of-the-art MRF method. Finally, the developed model was verified by conducted measurements.

In this paper, a Continuous Variable Transmission (CVT) of a All-Terrain vehicle is taken and the simulation of the heat distribution from the component is done using multiregion-coolingsphere solver in OpenFoam. The boundry conditions are acquired from the reference papers and the simulation is carried out.

2.Experiments carried out

All tests have been carried out on an engine test stand to provide reproducible boundary conditions. The CVT under investigation, is fully enclosed and has one inlet and outlet, respectively. Therefore, the heat release can be balanced when the mass flows and temperature differences are known. This is essential for validating the developed numerical model. This is of specific interest because it enables an insight into the temperature distribution of the pulleys and provides further data which can be used for the validation of the numerical simulation. Mass flow sensors require sufficient damping sections. Hence, pipes are mounted on the inlet and outlet ports to calm down the airflow and minimize the measurement error. The drive pulley is connected to an internal combustion engine and the driven pulley is attached to a dynamometer.

Thus it is possible to monitor the provided engine torque and the torque acting on the drive shaft. The rotational speed of both pulleys is also measured and therefore the engine power and the power output of the CVT can be computed. The difference is the power loss of the CVT unit which is converted into heat, warming up the whole system. Beside of monitoring the engine operating point, additional sensors are used to supervise the working conditions of the CVT. The temperature, mass flow and the differential pressure of the inflow and outflow are measured. Moreover, a novel system has been developed for the wireless measurement of the pulley surface temperatures during operation. It is based on the surface acoustic wave (SAW) technology and consists of a reader, a SAW transponder and a transponder antenna. The principle idea is that temperature influences the velocity of the SAW signal which is evaluated by the radar unit. Hence, the environmental temperature of the passive SAW transponder can be determined. A detailed description of a comparable test rig can be found in [28]. Two readers are placed close to the CVT casing to capture the signal of the sensors.

3. Boundary conditions

As they mentioned in the review paper carried out by Johannes Wurma, Matthias Fitl b, Michael Gumpesberger b, Esa Väisänen c, Christoph Hochenauer a [1] the boundary conditions are taken. Considering inlet a constant ambient pressure , thus the mass flow rate of air, entering the domain, is determined by the rotation of the pulleys as given by them. Again, ambient pressure is defined at the outlet. The wall of the casing is defined as adiabatic walls. As measured, the belt flank temperature is held constant at 130°C in the simulation.

4. Load case

With regard to the numerical modeling, one specific load case
has been defined. The engine speed was set to 3800 rpm and it runs on full load which is approximately 7.5 kW. This is one of the most crucial load cases concerning heat generation inside the casing. During the test runs the outflow temperature and the surface temperature of the pulleys were monitored. A measurement point was taken as soon as constant temperatures were reached and thus steady conditions can be assumed. The transmission ratio and the position of the pulleys are well known for this load case and a virtual model representing the corresponding conditions can be generated.

5. Numerical modelling

The main challenge of the presented task is the generation of a
transient CFD simulation of a CVT is currently difficult because the computational time would exceed reasonable limits. The reason is that on the one hand very small time steps would be necessary to model the pulley rotation adequately E.g. for a pulley which is rotating with 3800 rpm the time step should be set to 0.000022 s to rotate the pulley one degree per time step. On the other hand, the measurements have shown that the minimum time to reach thermodynamic equilibrium amounts approximately 5 min. Main reason is the thermal inertia of the solid components. In other words, 13.5 million time steps would be necessary.

Only rotational and translational movements can be modeled and they can only be applied if the motion is symmetric. The existing case has three major issues which need to be taken into account for the modelling. First, the complex movement of the belt needs to be separated into rotational and translations regions. Second, special attention needs to be paid to the contact faces between belt and pulley. These zones are static in a steady state simulation. However, they obviously vary as soon as the pulleys are rotating. Moreover, the heat transfer coefficient h at the pulley surface depends on the current position of the disk. Without the developed method this would lead to inhomogeneous cooling of the pulley surface. Considering the thermal analysis, this must be considered to gain a tangential homogeneous and radial symmetric temperature profile. Finally, the heat generation inside the domain needs to be defined and validated.

5.1. CAD model and meshing

The assembly contains the cover, drive and driven pulley and the rubber belt. In total there are two ports, one inlet port and one outlet port. Beside of the fluid domain also the solid parts are taken into account to model heat conduction inside the pulleys accordingly. In a first step a grid independency study has been carried out using 2.5 million, 5 million and 10 million cells and it can be shown that the surface temperatures, outlet temperatures and flow velocities hardly vary between the last two versions. Therefore, an approximate cell number of 5 million is sufficient to resolve the computational domain. The meshing process starts with the generation of a triangular surface mesh. As block mesh type in Open Foam is the whole domain is discretized with this kind of cells. Their advantage is that they are numerically less diffusive and the solution is more accurate than a comparable tetrahedral mesh. Moreover, the number of cells is approximately five times lower which is helpful to reduce computational time. For the accurate modelling of convective heat transfer near the wall regions, three prism layers have been extruded from the contact interfaces separating the fluid and the solid region. The polyhedral structure of the cells is shown in two cutting planes, one through the fluid domain on the left and one through the solid domain on the right. In the enlargement the wall near prism layers are visible which grow with a rate of 1.25. The final mesh consists of 3.4 million cells in the fluid region and 2 million cells discretising the solid region which leads to a total amount of 5.4 million cells.

5.2. Modelling motion

As aforementioned, the belt motion is split into two rotational and two translational regions. Therefore, the belt is enwrapped to separate the belt near region from the common fluid domain and thus the according rotational and translational velocities can be specified for each section. The same procedure is applied on each side of both pulleys to define the rotational regions close to the pulley surface. In total, the fluid domain consists of nine regions which are connected via interfaces. Additionally, three solid regions are defined representing the drive pulley, driven pulley and belt. Gullberg et al compared numerical results to experimental data [30].

5.3. Modelling of generated heat

As mentioned before, the efficiency factor of common CVTs is moderate for the defined load case. The power loss leads to an increase of the component temperatures because the kinetic energy is converted into heat. Generally, three different mechanisms are responsible for the power losses in rubber belt CVTs.

First, there are sliding losses between belt and pulley resulting from contact forces and relative slip. Second, there are hysteresis losses resulting from cyclic deformation of the belt in longitudinal and the transverse directions and third, work is required to engage and disengage the belt in and from the pulley, compare [15]. However, these effects cannot be modelled in CFD and thus, alternative methods need to be found to enable a heat transfer analysis. A useful approach is the definition of a constant temperature on the flanks of the belt, as well as in the contact surface. In these zones are highlighted.

5.4. Modelling of heat transfer between solid and fluid region

Convective heat transfer between fluid and solid regions depends on the flow profile within the fluid region and on the temperature gradient between wall and fluid zones. Generated mesh: (a) cutting plane through the fluid domain; (b) cutting plane through the solid domain. In case, the flow profile inside the casing is strongly asymmetric. As a result, the local boundary heat release varies significantly on the pulley surface, whereas it should be rotational symmetric because of the high rotational speed. This effect is pointed out in the left contour plot. It shows the relative surface heat flux with regard to the surface averaged heat flux. However, the rotational symmetric profile, shown on the right side, is far more realistic. This result has been computed by using the heat flux averaging method developed in the previous study [27]. The homogenization of the heat flux profile is necessary to compute accurate temperature values. The applied approach to define the thermal boundary conditions for the interfaces between solid and fluid region is based on coordination transformation and interpolation. In contrast to the fluid-fluid interfaces, where the flow quantities are directly transferred from one region to another, the fluid-solid interfaces are operated before they interact with each other. Starting with an initialized surface wall temperature a corresponding heat flux profile is computed in the fluid domain. The profile is highly asymmetric as shown before on the left side. Before the subsequent iteration, the cell value of each surface cell is extracted and stored in a matrix. Furthermore, the matrix contains the coordinates of each cell referring to a cylindrical body fixed coordinate system. Thus, it consists of 4 columns and n rows, where n corresponds to the number of surface cells. A function is defined which extracts the flow quantity at a defined position referring to a specified coordinate system. Going back into the previously defined matrix the corresponding flow quantity can be extracted due to the fact that the displacement vector. Finally, the arithmetic average of the extracted heat flux values is computed and a tangential homogenous and radial symmetric temperature profile can be achieved which grants that the energy balance is fulfilled. It must be mentioned that the developed method works only, if the pulley geometry is symmetric. Afterwards, the averaged profile of the surface heat flux is used as boundary condition for the solid region. In accordance to BCs a surface wall temperature is computed. Again, the resulting temperature profile is stored in a matrix Temperature which is used as BC for the solid region in the following iteration.

The developed method is capable to compute the heat distribution on rotating disks in transient cases and can therefore be seen as expansion to the state-of-the art overset grid method which has not been published before. In [27] a more detailed description of the numerical scheme can be found.

5.5. Numerical settings

An implicit pressure based coupled solver has been used to solve the governing equations for fluid dynamics. The statistic modelling of the turbulent flow, known as Reynolds Averaged Navier Stokes (RANS)-method, demand the definition of a turbulence model. The 2-equation k-x- model has been chosen because of its good stability and acceptable computational time. The fluid medium is air and its density is computed by the incompressible ideal gas equation. This assumption is appropriate due to the reason that the maximum temperature is 130°C and only low Mach number occur within the fluid domain. Thus the change of the fluid density is negligible. For a coupled solver it is necessary to define a Courant number, also denoted as CFL number. The reason is that multiregion Cooling sphere uses a time marching scheme to derive a steady state form of the governing equations. In the presented case the CFL number has been set to 5 as recommended in [32]. The second-order-upwind scheme is used for the spatial discretization of the governing equations. For the solid region two different parameters have been defined. The pulleys consist of an aluminium alloy with constant density and conductivity. For the belt, a specific material is defined with an extraordinary high thermal conductivity of 350W=mK. This is necessary to ensure that the heat is spread in the belt region homogeneously, like in a fast rotating belt. Moreover, a surface-to-surface (S2S) radiation model has been applied to investigate the impact of radiation. However, it can be stated that its influence is marginal. The advantages of these numerical settings were scrutinized in previous simulations, see [33].

4. Results and discussion

In a first step the results gained from the simulation will be compared to the measurements. CFD enables an insight on the temperature distribution which is hard to achieve on a test rig. However, it needs to be mentioned, that experiments are indispensable to validate the computed results to ensure an appropriate conclusion. Once a validated modelling strategy exists, important information concerning flow conditions and heat transfer can be gained from the numerical results. The computed surface temperature can be seen. Obviously, the highest temperatures occur in the contact regions between belt and pulleys. The pulleys are made out of an aluminium alloy with a thermal conductivity of 350W/mK. Hence, the heat transport to colder regions is rather efficient and the temperature gradients within the solid are low. Nevertheless, the temperature difference in radial direction between the contact zone and the outer border is 50 K on the driven pulley. On the drive pulley similar ranges can be observed in axial direction. The measurement values have been taken after running the engine on full load for 5 min at 7500 rpm. As described in Section 2, in total five temperature sensors have been mounted on the pulleys to observe the surface temperatures online. The temperature spread between to outer sensor (P3) and the inner sensor (P1) amounts 28 K. Almost the same range, namely 29 K, is computed by the numerical model. Taking into account that the measurement tolerance lies within 1.5 K, good accordance is given. The simulation model slightly overestimates the surface temperature. The maximum deviation between measurement and simulation amounts 4 K in the middle section of the drive pulley. The radial surface temperature profile along the cross section through one side of the driven pulley is presented . It points out that the peak temperatures are reached in the contact zones. It also shows that the temperature drop is stronger in the outer regions than towards the centre.

One issue which has been discussed in Section 3, is the homogeneity of the radial temperature distribution. It can be shown that the resulting temperature profile is perfectly rotational symmetric like it can be expected for a fast rotating pulley. Another indicator, which is taken for the validation of the numerical model is the air temperature of the outflow. If the total mass flow is known, it can be used as indicator to compute the total energy that is transformed into heat by the CVT. The comparison between the simulated and measured temperature. The inlet temperature used for the simulation is determined by the ambient conditions at the test rig. Inside the casing the air is heated up from 25 °C to approximately 60 °C. The measurements showed that the outflow temperature at the drive outlet is slightly higher than at the driven outlet. The same behaviour can be observed in the simulation model. Again, the CFD results are in good agreement with the data gained by testing and the maximum deviation amounts 2 K. Furthermore, the measured pressure value gives a good understanding of the airflow that is sucked in from the environment. The low-pressure value is related to the ambient air pressure. On the drive inlet the pressure difference is rather small compared to the measured value at the driven inlet. One cause for this effect is the layout of the blades on the pulley surface and a second reason is the casing design. As previously mentioned, ambient pressure is defined at both inlet regions in the simulation, hence the airflow entering the fluid domain is determined by the low-pressure produced by the pulley rotation. Again, the simulation results show the same tendency as observed in the measurements. To summarise, the simulation model with constant belt temperature reproduces the measurement data accurately. This is substantiated by corresponding inflow and outflow condition and conforming component temperatures. A huge advantage of the simulation model is that space resolved results are available. The air velocity distribution inside the casing can be examined. Hence it is possible to visualize flow structures and identify dead water regions. Also peak temperatures can be spotted, as well as regions of high and low convective heat transfer. This is of special interest when it comes to the point of optimizing an existing system. The resulting air temperature in the middle cutting plane through the cover. It clearly points out that between the pulley the air temperature is rising. This correlates where it can be seen that the air velocity is comparatively low, thus the heat exchange is limited. Another indicator to evaluate the characteristics of the heat transfer. The heat transfer coefficient depends on the temperature gradient between wall and fluid as well as on the air flow conditions. Hence it is high on the driven pulley surface (right disk). Cold air is entering the domain and the fast rotating disk directly influences the forced convection. In contrast the heat transfer coefficient of the drive pulley (left) is approximately three times lower.

The next step of an optimization process is to decide what kind of changes could have a positive effect. Therefore, the information which can be gained from the previously described plots can be used. By comparing the results differences can be immediately identified. Hence the major advantage of the presented model is that new designs can be investigated and evaluated with little effort and costs.

5. Conclusion

With increasing engine performance, loads on the CVT components rise as well. The power loss of the transmission leads to high thermal loads and the resulting peak temperatures drastically reduce the life span of the belt. To overcome this issue new designs which focus on improving the heat transfer and reducing high temperature zones need to be found. The scope of the presented work is to build a numerical model which is capable of computing heat transfer effects within an enclosed CVT. Main target is the development of an efficient method to evaluate design changes rapidly and focus cost intense experimental work to promising concepts only.

The Overset grid approach has been used to imply motion of the belt and pulleys, however, the resulting surface temperature greatly depends on the flow conditions inside the casing and with ordinary approaches the computed results were insufficient. Therefore, a novel method has been developed which can be seen as add on to the state-of-the-art Overset grid modelling. As a result, tangential homogenous and radial symmetric temperature distribution on the pulley surface can be computed. Measurements have been conducted on an engine test stand to evaluate the generated virtual model and the comparison shows excellent accordance. The developed model includes full motion of all moving parts and predicts realistic temperature distributions and mass flow rates. Thus, it can be used to optimize the design of CVT components to increase the heat transfer rate and lower peak temperatures. Parameter studies as well as design changes can be evaluated within a short time.

References

[1] N. Srivastava, I. Haque, A review on belt and chain continuously variable transmissions (CVT): dynamics and control, Mech. Mach. Theory 44 (2009) 19– 41.

[2] D.J. Bents, Axial Force and Efficiency Tests of Fixed Center Variable Speed Belt Drive, SAE Technical Paper No. 810103, 1981.

[3] T.F. Chen, D.W. Lee, C.K. Sung, An experimental study on transmission efficiency of a rubber V-belt CVT, Mech. Mach. Theory 33 (1998) 351–363.

[4] B.G. Gerbert, Power loss and optimum tensioning of V-belt drives, J. Eng. Ind. 96 (1974) 877–885.

[5] T.H.C. Childs, D. Cowbum, Power transmission losses in V-belt drives. Part 1: mismatched belt and pulley groove wedge angle effects, Proc. Inst. Mech. Eng. D201 (1987) 33–40.

[6] T.H.C. Childs, D. Cowburn, Power transmission losses in V-belt drives. Part 2: effects of small pulley radii, Proc. Inst. Mech. Eng. D 201 (1987) 41–53.

[7] T.F. Chen, C.K. Sung, Design considerations for improving transmission efficiency of the rubber V-belt CVT, Int. J. Veh. Des. 24 (2000) 320–333.

[8] G. Gerbert, Force and slip behaviour in V-belt drives, in: Mechanical Engineering Series, Acta Polytechnica Paper Series No. 930667, 1972, pp. 1–101.

[9] G. Gerbert, Traction Belt Mechanics, Chalmers Univ. of Technology, 1999.

[10] J.P. Dolan, W.S. Worley, Closed form approximations to the solution of V-belt force and slip equations, J. Mech. Des. 107 (1985) 202–300.

[11] F. Sorge, A qualitative–quantitative approach to V-belt mechanics, J. Mech. Des. 118 (1996) 15–21.

[12] F. Sorge, Analysis of the ratio shift transient in V-belt variators, in: Proc. 11th World Congress in Mech. Mach. Science, 2003.

[13] F. Sorge, Transient mechanics of V-belt variators, in: CVT 2004 Conference, San Francisco, 2004.

[14] M. Cammalleri, F. Sorge, Approximate closed-form solutions for the shift mechanics of rubber belt variators, in: Proc. ASME Int. Design Eng. Tech.
Conferences, 2009.

[15] L. Bertini, L. Carmignani, F. Frendo, Analytical model for the power losses in rubber V-belt continuously variable transmission (CVT), Mech. Mach. Theory (2014) 289–306.

[16] G. Julió, J.S. Plante, An experimentally-validated model of rubber-belt CVT mechanics, Mech. Mach. Theory (2011) 1037–1053.

[17] Y. Xinhua, C. Naishi, L. Zhaohui, Electro-Mechanical Control Devices for Continuously Variable Transmissions, SAE Technical Paper No. 2008-01-1687, 2008.

[18] C. Zhu, H. Liu, J. Tian, Q. Xiao, X. Du, Experimental investigation on the efficiency of the pulley-drive CVT, Int. J. Autom. Tech. (2010) 257–261.

[19] B. Bonsen, T. Klaassen, K. van de Meerakker, M. Steinbuch, P. Veenhuizen, Analysis of slip in a continuously variable transmission, in: Proceedings, ASME International Mechanical Engineering Congress, 2003.

[20] F. van der Sluis, T. van Dongen, G.-J. van Spijk, A. van der Velde, A. van Heeswijk, Efficiency Optimization of the Pushbelt CVT, SAE Technical Paper No. 2007-01-1457, 2007.

[21] J. Li, B. Ma, R. Wang, Study on a cooling system based on thermoelectric cooler for thermal management of high-power LEDs, Microelectron. Reliab. 51 (2011) 2210–2215.

[22] J. Li, X. Zhang, C. Zhou, J. Zheng, New applications of an automated system for high-

power LEDs, IEEE-ASME Trans. Mech. 21 (2) (2016) 1035–1042.

[23] A.L. Vaishya, S. Phadnis, Experimental Investigations of Forced Air Cooling for Continuously Variable Transmission (CVT), SAE Technical Paper No. 2013-32- 9073, 2013.

[24] S. Dhongde, V. Chandran, Experimental Study of Cooling of Continuously Variable Transmission (CVT) in Scooter, SAE Technical Paper No. 2014-01- 2003, 2014.

[25] N. Karthikeyan, A. Gokhale, N. Bansode, Minimizing the Dust Ingress into the CVT Housing of a Scooter Using CFD Analysis, SAE Technical Paper No. 2014-01-0579, 2014.

[26] N. Karthikeyan, A. Gokhale, N. Bansode, A Study on Effect of Various Design Parameters on Cooling of Clutch for a Continuously Variable Transmission (CVT) System of a Scooter, SAE Technical Paper No. 2014-01-2595, 2014.

[27] J. Wurm, M. Fitl, M. Gumpesberger, E. Väisänen, C. Hochenauer, Novel CFD approach for the thermal analysis of a continuous variable transmission (CVT), Appl. Therm. Eng. (2016) 159–169.

[28] L.M. Reindl, I.M. Shrena, Wireless measurement of temperature using surface acoustic waves sensors, IEEE Trans. Ultrason. Ferroelectr. Freq. Contr. 51 (11) (2004) 1457–1463.

[29] P. Gullberg, L. Löfdahl, P. Nilsson, S. Adelman, Continued Study of the Error and Consistency of Fan CFD MRF Models, SAE Technical Paper No. 2010-01-0553,2010.

[30] P. Gullberg, R. Sengupta, Axial Fan Performance Predictions in CFD, Comparison of MRF and Sliding Mesh with Experiments, SAE Technical Paper No. 2011-01-0652, 2011.

[31] ANSYS FLUENT, Theory Guide 14.0, Ansys Inc., Canonsburg, PA, USA, 2010.

[32] Star CCM+ User Guide, CD Adapco, 2015.

[33] J. Wurm, M. Fitl, M. Gumpesberger, E. Väisänen, C. Hochenauer, Numerical and experimental investigation of thermal conditions inside the engine compartment of snowmobiles, SAE Int. J. Commer. Veh. 8 (1) (2015) 225–235.J. Wurm et al. / Applied Thermal Engineering 114 (2017) 545–553 553

[34] Advanced heat transfer analysis of continuously variable transmissions (CVT) Johannes Wurma,⇑ , Matthias Fitl b, Michael Gumpesberger b, Esa Väisänen c, Christoph Hochenauer a