International Journal of Thermal Sciences 132 (2018) 257–266

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

journal homepage: www.elsevier.com/locate/ijts

A coupled approach for vehicle brake cooling performance simulations

Alexey Vdovin[a,*], Mats Gustafsson[b], Simone Sebben[a]

[a] Department of Mechanics and Maritime Sciences, Chalmers University of Technology, 412 96, Gothenburg, Sweden
[b] Braking Department, Volvo Car Corporation, Torslanda PVV2:1, Gothenburg, 405 31, Sweden

ARTICLE INFO

Keywords:
Brake cooling
Alpine descent
CFD
Brake thermal management

ABSTRACT

Advances of CFD methods together with the constant growth of computer capacity enables simulations of complex coupled fluid and thermal problems. One such problem is the evaluation of brake cooling performance. The brake system is a critical component for passenger vehicles and ensuring correct brake operation under all possible load scenarios is a safety issue. An accurate prediction of convection, conduction and radiation heat fluxes for such a complicated system is challenging from modelling as well as numerical efficiency perspectives.

This study describes a simulation procedure developed to numerically predict brake system component temperatures during a downhill brake performance test. Such tests have stages of forced and natural convection, and therefore, the airflow is influenced by the temperature changes within the system. For the numerical simulation, a coupled approach is utilized by combining aerodynamic and thermal codes. The aerodynamic code computes the convective heat transfer using a fully-detailed vehicle model in the virtual wind tunnel. The thermal code then uses this data and combines it with conduction and radiation calculations to give an accurate prediction of the component temperatures, which are subsequently used for airflow recalculation. The procedure is described in considerable detail for most parts of the setup.

The calculated temperature history results are validated against experimental data and show good agreement. The method allows detailed investigations of distribution and direction of the heat fluxes inside the system, and of how these fluxes are affected by changes in material properties as well as changes in parts within or outside the brake system. For instance, it is shown that convection and especially convection from the inner vanes is the main contributor for the heat dissipation from the brake disc. Finally, some examples of how changing the vehicle design affects the brake cooling performance are also discussed.

1. Introduction

The brake system is one of the most critical components of a vehicle since it converts the vehicle kinetic, or potential, energy into heat, allowing the vehicle to decelerate or stop when needed. Overheating of the brake system components is a safety issue that can cause serious problems such as decreased friction coefficients, i.e. brake fading. Moreover, such overheating can result in generating brake judder and/or brake squeal, increasing wear, thermal cracking and even brake fluid vaporization [1,2]. Furthermore, other wheel suspension parts, like wheel cover or tyre, can also be affected by high temperatures, especially during thermal soak (natural convection) [3,4]. In order to avoid these problems, the brake system should be appropriately dimensioned and designed to ensure the correct operation under various braking scenarios.

Historically, this has been achieved by extensive experimental testing of the brake components using various test benches and full vehicle testing. However, in recent decades, the constantly growing computation capacity has led to significant advances in simulation methods. Computer Aided Engineering (CAE) methods are not only cheaper but also provide more detailed information about heat dissipation from and within parts, and heat transfer coefficients, and rates. In addition, CAE can be used during early stages of the vehicle development when physical prototypes may be unavailable or when many variants are being developed.

Full-scale simulations of a brake system can be done with different levels of complexity. Whereas simple simulations can focus on a single or several ventilation channels inside the brake disc [5,6]; or a complete brake disc and selected parts around it [7–10]; more complex simulations include the entire vehicle [11,12]. As shown by Stephens et al., the latter allows better representation of the on-road conditions [13], but requires at the same time much more data about the components and their relative material properties. While most numerical simulations focus either on computing convection heat transfer

---

[*] Corresponding author.
E-mail addresses: alexey.vdovin@chalmers.se (A. Vdovin), mats.r.gustafsson@volvocars.com (M. Gustafsson), simone.sebben@chalmers.se (S. Sebben).

https://doi.org/10.1016/j.ijthermalsci.2018.05.016
Received 25 September 2017; Received in revised form 9 May 2018; Accepted 9 May 2018

coefficients from a Computational Fluid Dynamics (CFD) point of view, or on conduction modelling through the solid parts, a significant step was taken [14–16] by coupling aerodynamic and thermal codes. In such cases, the aerodynamic code is responsible for the computation of the convective heat transfer rates whereas the thermal code handles the conduction and the radiation behaviour. Considering the convection heat transfer, the flow around the rotating disc is often considered the same, and independent of, the surface temperatures. However, if the thermal soaking process is taken into account, the airflow is then driven by the buoyancy forces and is then dependent on the surface temperatures [17].

As simulation methods develop, more and more effort is being put into replacing long and expensive physical testing of brake systems. These simulations need to not only have very good geometrical representation of vehicle parts but also good material data. The present paper aims at describing the implemented coupled simulations of a 30-min mountain descent brake test. The simulation method used is similar to the one presented in other studies [14–16], with the main differences being in the software products used, different loading scenario and most importantly different approach taken for the coupling between the aerodynamic and thermal solver. Due to relatively low flow velocities and the presence of the natural convection stage in the brake test simulated, the heat transfer coefficients cannot be assumed independent of the surface temperatures and therefore two-way coupling is required.

2. Driving cycle and the test object description

There are a number of different driving cycles that are commonly used for brake cooling performance tests. The coupled procedure described in this paper is developed to numerically simulate one of them: the Alpine Descent brake test. The sketch in Fig. 1 shows the real Alpine Descent test, which consists of two phases:

1) downhill driving at 10 m/s with more or less constant drag braking (heat-up phase) on a slope of about 10% for about 20 min;
2) thermal soaking, when the vehicle stands still and the heat dissipates from the disc into the brake system and environment (cool-down phase) for about 15–20 min.

During the experiment, the focus is on temperatures of the brake disc and the brake fluid. In order to consider the worst-case scenario for the fluid, the brake disc and brake pads are kept in constant contact with the disc even during the cool-down phase.

To improve development efficiency, simplified variants of this test have been devised. The replacement tests can be conducted either on a level test track where the braking vehicle is being pushed by a second car, or in a wind tunnel. Both variants are representing the Alpine descent scenario relatively good with the slight difference being in vehicle pitch angle due to vehicle mass distribution on the slope and on the flat ground. The wind tunnel variant is more expensive, but it provides a much more controlled test environment with fixed air temperature and speed, no weather complications, and constant braking force. Consequently, in order to reduce uncertainties in the simulation results inherent to experimental testing, the present computational procedure replicates laboratory experiment performed in a wind tunnel.

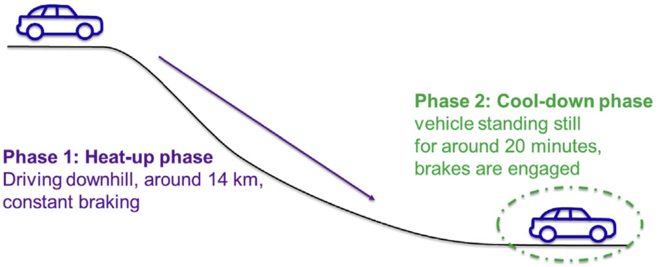

Fig. 1. Sketch of the studied Alpine descent driving cycle.

A full-size production sedan type vehicle with 300 mm brake discs was used as a test object for both simulations and wind tunnel experiments. From previous experience, it was known that due to engine bay packaging, the right-hand wheelhouse of the specific vehicle experienced higher temperatures. Therefore, the experiment and modelling focused only on this side of the vehicle.

3. Numerical method description

The following procedure is based on the coupling of a CFD code with a Finite Element Analysis (FEA) thermal simulation. StarCCM+ is used for calculation of the flow and convective heat transfer, TAITherm is the main thermal solver, and CoTherm works as a coupling software that allows easy setup and control over data exchange between the two solvers.

The Alpine descent braking scenario dictates the approach that needs to be taken for recalculation of the airflow around the brake disc and hence the convective heat fluxes. While for high velocities the assumption of the surface heat transfer coefficients being independent of the surface temperatures can be taken, this is not true for the Alpine descent since the velocities are not high enough. Moreover, during the cool-down phase, the flow is completely driven by the buoyancy forces and thus highly dependent on component surface temperatures. In this study, the airflow and convective heat fluxes are recalculated several times and linearly interpolated in-between the recalculations for both heating-up and cooling-down phases.

The description of the aerodynamic model and the thermal models as well as the coupling process are presented and discussed below. The two phases of the test (heat-up and cool-down) are quite different from a computer simulation perspective and are therefore defined by two aerodynamic and two thermal setups.

3.1. Aerodynamic model

In order to calculate the surface convective heat fluxes for the parts of the brake system a CFD simulation needs to be prepared. A fully-detailed vehicle model is positioned in the virtual wind tunnel according to Fig. 2. It is important that the vehicle geometry is well represented in order to capture the flow around the parts of interest. On the other hand, since the mapping between codes is based on the coordinates there is no need to have matching mesh or matching surface names between them.

To decrease the mesh count while maintaining mesh-independency of results, only the right half of the vehicle is simulated with the second half imitated using a symmetry plane. The virtual tunnel is 60 m long, 40 m wide and 20 m tall to ensure that there is no unwanted interference from the tunnel boundary conditions. The model is positioned 20 m away from the inlet. The mesh is predominantly hexahedral with refinements concentrated around the front wheelhouse since it is the

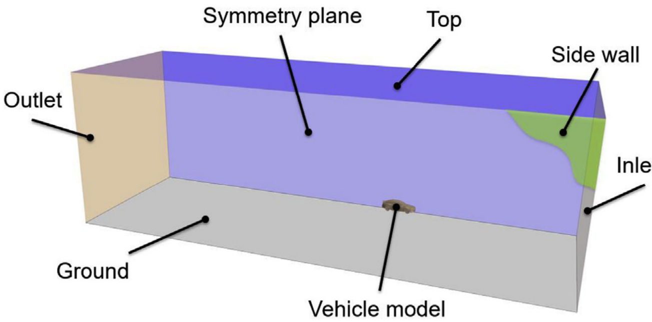

Fig. 2. CFD model of the vehicle in the virtual wind tunnel.