

Combined Experimental and CFD Investigation of Brake Discs Aero-thermal Performances

G. Barigozzi and A. Perdichizzi
Università di Bergamo

M. Donati
Brembo S.p.A.

Copyright © 2008 SAE International

ABSTRACT

The paper reports on a combined experimental and numerical investigation of the aero-thermal behavior of a vented automotive brake disc. The aim is to assess to which extent nowadays available CFD codes can be confidently used in the automotive brake disc design process. The reference experimental investigation provided the aero-thermal flow characteristics at the exit of a vented brake disc, with a pin type internal geometry and drilling holes. Tests were performed at one disc rotational speed and at several braking conditions. Star CD software was employed to compute the internal flow. Results were compared and discussed in terms of both global and local quantities. A general good agreement between numerical and experimental results was obtained. Thus CFD simulation was proved to be a very useful tool for brake disc designers.

INTRODUCTION

The continuous demand of weight reduction and performance increase gives to cooling optimization an increasing relevance in the brake disc design. There are several ways to improve the convective heat transfer coefficient on the surface of a disc rotor, but the optimization of the internal brake geometry is often the heart of the design process. Of course, a convective heat transfer increase on the disc internal surface has to be related to the brake disc overall heat transfer rise; this in order to evaluate if this improvement is significant. This is particularly true when the vehicle cooling contribute is dominant.

The design and development of advanced vented brake discs nowadays demands for a detailed knowledge of complex phenomena, being the fluid dynamics involved three-dimensional, unsteady, and characterized by the simultaneous occurrence of flow separations and strong heat transfer fluxes in a domain of complex geometry. Detailed experimental investigations are not always convenient, especially at the beginning of a design process, when several geometries need to be tested. From this point of view, Computational Fluid Dynamic

(CFD) simulation, when thoroughly validated against experimental data, can be very useful, allowing a reduction of product developing time.

The prediction of an average heat transfer coefficient has been tried in the past using semi-empirical correlations [1,2], whereas recent attempts can be found in the literature [3,4,5] to predict it by CFD simulations. Experimental analysis of air flow exiting the disc channels are few [4,6,7] and performed by techniques (Pitot tubes) that are not suitable for unsteady flows with large angle variations, like those generated by the passing of the rotating disc channels.

In vehicle application tests, only the global convective heat transfer coefficient in the brake-vehicle environment is usually evaluated, as it is easier to be measured without expensive techniques. Experimental evaluations of inner heat transfer coefficients are rare. Stephens et al. [8] obtained this information in an indirect way, measuring the disc temperature and the cooling air heat flux.

In the last years Brembo has been involved in a long-term research program with the Industrial Engineering Department of Bergamo University aimed at the comprehension of thermo-fluid-dynamic aspects of brake discs. Results of this experimental activity are shown in previous papers [9-11]: internal heat transfer coefficient values obtained from measured exit air velocity and exit air and disc temperature distributions are presented for several rotor disc geometries and materials (pin and blades discs, made both of cast iron and CCM) and variable disc rotational speed and braking conditions. Even if performed on a simplified system (disc and caliper configuration without forced convection), these studies allowed to setup an accurate and repeatable testing and data processing procedure and to gain a deeper understanding of heat transfer phenomena taking place in a brake system. In particular, the energy balance of the CCM brake disc system [11] showed a quite relevant contribution of the vented airflow to the heat dissipation, in the range 50-70%, depending on the braking conditions.

It then naturally arose the idea of investigate to which extent nowadays available CFD codes can be

confidently used in the design process, particularly also for new light weight designs with specific geometrical conditions in the connection zone between hat and brake ring. In the present paper, results from the widely used commercial software Star CD by CD-Adapco, available by Brembo, are compared against experimental data.

The reference experimental investigation concerns the aero-thermal flow characteristics at the exit of a vented automotive brake disc, with a pin type internal geometry and drilling holes.

Results are compared and discussed in terms of both global (e.g. mass flow rate, heat transfer coefficient) and local quantities (radial velocity component and temperature distributions at the disc exit section).



Figure 1: The disc and the caliper.

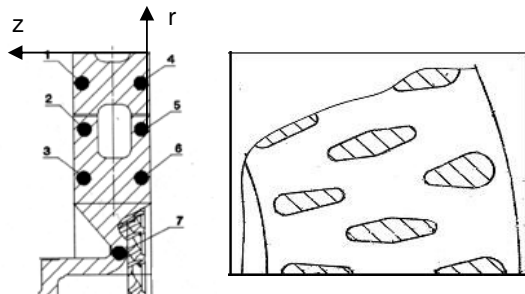


Figure 2: The disc geometry.

EXPERIMENTAL DETAILS

A cast iron disc with a pin type internal geometry and drilling holes was tested on the bench available at the Laboratory of the Industrial Engineering Department of Bergamo University. An electric motor (40kW maximum power, 1500 rpm maximum speed), whose speed was controlled by an inverter, was used to get the disc rotation. A more detailed description of the test bench can be found in the literature [9]. X-drilling holes are periodically distributed over both friction surfaces. Pins periodicity and drilling holes periodicity are not consistent, making the definition of a disc periodic sector practically impossible. Nevertheless, some preliminary tests have shown that the flow structure at disc exit is quite insensitive to drilling holes position and resembles

the pin disc flow field without drilling holes [10]. For this reason, the disc was shared into periodic sectors corresponding to the pin virtual channels.

The disc was rotating in still ambient air at room pressure, without any forced convection on it. A caliper was fitted to the test rig in its usual position (Figure 1).

In Figure 2 the internal geometry of the cooling channels can be seen in a section. For each set of tests, 7 thermocouples were installed underneath both friction surfaces at different radial and circumferential positions (Figure 2). Thermocouple signals are transferred to the absolute frame of reference through a 5 channels slip ring. One more thermocouple was used to measure the friction surface temperature on the air inlet side.

Two techniques were used for this investigation: the Cold Wire thermometry (CW), to measure the air temperature, and the Laser Doppler Anemometry (LDA) to measure the gas velocity. The temperature field was monitored using a Cold Wire Thermometer (DANTEC 55P31 or 55P11 depending on the air temperature [11]), connected to a DANTEC CCA module. For the velocity measurement a four beam, two color LDA system (DANTEC Fiber Flow) was used. A magnetic pick-up provided a once per revolution trigger signal, allowing the phase locked acquisition of CW and LDA samples. Details of both measurement techniques can be found in [11], while measurement uncertainties are summarized in Table 1.

Tests were carried out at a single disc rotational speed, 750 rpm, and under almost steady-state braking conditions. In fact, during tests, the pressure in the caliper air supply system was continuously controlled by a PID control system in such a way to maintain the friction surface disc temperature as constant as possible. The test conditions are summarized in Table 2, T_d being the mean disc temperature.

The air velocity and temperature were measured at the exit of the venting channels on a cylindrical surface coaxial to the disc axis having a distance from the disc outer border of 3.8 mm. The disc width was divided into 19 sections, whose distance decreased approaching the channel boundaries.

To correctly compare experimental and numerical results, all measured velocity and temperature data at the same pitch wise position belonging to all the circumferential periodic sectors were averaged.

Air velocity	$\pm 2 \%$
Air flow angle	$\pm 1^\circ$
Probe position	$\pm 0.1 \text{ mm}$
Air temperature	$\pm 0.51^\circ \text{C}$

Table 1 – Measurement uncertainties.

$T_d (^\circ \text{C})$
20
145
260
410

Table 2 – Test conditions.

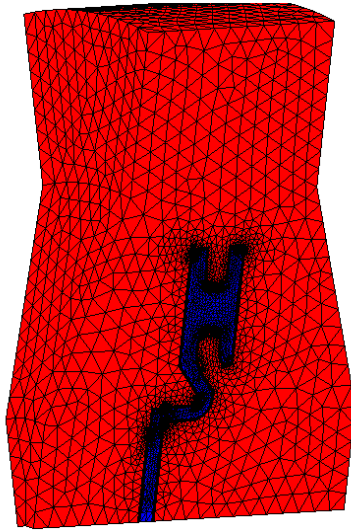


Figure 3: The computational mesh.

NUMERICAL DETAILS

Star CD software has been employed to compute the air flow of the disc. All simulations were performed in steady conditions, with rotational speed of 750 rpm, in the relative frame of reference, thereby ignoring the presence of caliper and any non-axisymmetric fixed obstacle.

The comparison with experimental data is firstly carried out in "cold" conditions, without any heat flux at side walls, assuming incompressible constant-properties flow. It is then repeated in "hot" conditions, with a steady state Conjugate Heat Transfer Analysis, which allowed calculating the heat transfer between solid rotor and air, solving the heat-conduction equation in the metal and imposing a heat flux at side walls, in order to reach the experimentally measured disc temperature.

To reduce the computational time, only a segment of the disc was modeled, thanks to the use of cyclic boundaries. The computational domain thus includes one disc periodic sector, the internal channels air path and an external air region surrounding the disc. In this way the heat flux entering the simulated braking system can be dissipated by convection with both the internal channels air and the external friction surfaces surrounding air, while radiation is not considered. The mesh, for fluid and solid, was composed of about 440000 cells: extrusion prisms were used for the interface zone between the rotor and the air, tetrahedral for other zones. (Figure 3). The computational grid covers two pitches in the tangential direction, it's about twice the disc height in radial direction, and it has a varying width in the axial one (from 3 up to 6 times the distance between the friction surfaces). Cyclic symmetry was imposed on the periodic boundaries (with normal vector in tangential direction), while quiet air at ambient condition was imposed on the other ones. Previous simulations have shown that the drilling holes distribution do not have a great influence on fluid structure at disc

exit, so drilling holes distribution was forced to be periodic, with a period of two pins, with the advantage to get a great reduction of computational time.

The air was modeled as turbulent and incompressible, using the low Reynolds Number standard quadratic k - ϵ turbulence model, in which transport equations for k and ϵ are solved everywhere, including the near-wall regions.

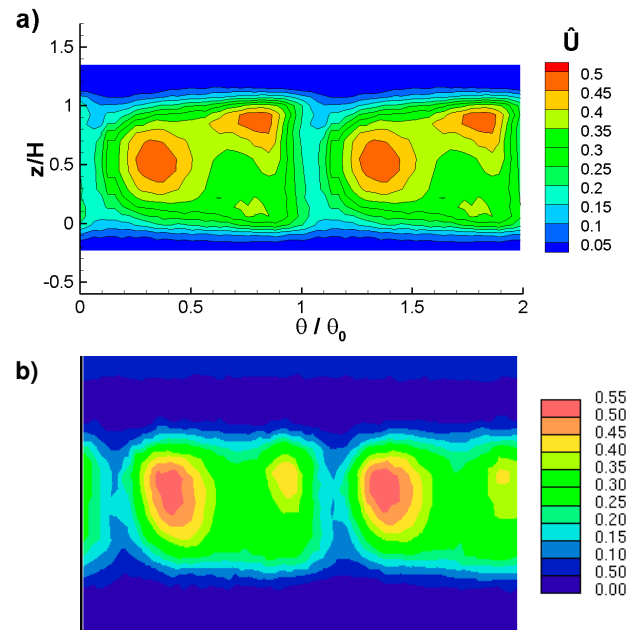


Figure 4: $\hat{U}$ distribution in non braking condition at 750 rpm –a) experimental and b) CFD.

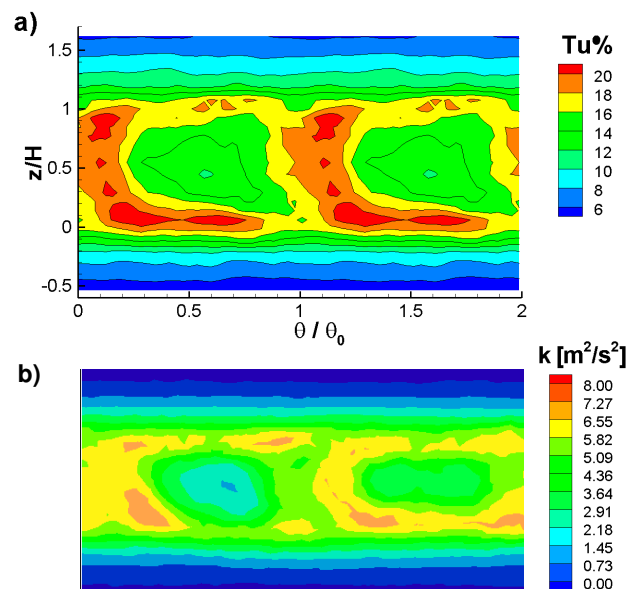


Figure 5: a) measured $Tu\%$ distribution and b) computed k distribution - non braking condition at 750 rpm.