

Integrated Numerical and Experimental Approach to Determine the Cooling Air Mass Flow in Different Vehicle Development Stages

2010-01-0287

Published
04/12/2010Stefan Wille, Timo Kuthada, Nils Widdecke and Jochen Wiedemann
FKFSHolger Maier and Andreas Koller
Porsche AG

Copyright © 2010 SAE International

ABSTRACT

This paper presents an integrated numerical and experimental approach to take best possible advantage of the common development tools at hand (1D, CFD and wind tunnel) to determine the cooling air mass flow at the different vehicle development stages.

1D tools can be used early in development when neither 3D data nor wind tunnel models with detailed underhood flow are available. A problem that has to be resolved is the dependency on input data. In particular, the pressure coefficients on the outer surface (i.e. at the air inlet and outlet region) and the pressure loss data of single components are of great importance since the amount of cooling air flow is directly linked to these variables. The pressure coefficients at the air inlet and outlet are not only a function of vehicle configuration but also of driving velocity and fan operation. Both, static and total pressure coefficient, yield different advantages and disadvantages and can therefore both be used as boundary conditions. For the characteristic pressure losses of single components in the air path, data from a predecessor can be utilized as a first approximation. The negligence of changed flow topology (flow separations or changing approach flow angles) results in a loss in accuracy. Improvements can be achieved if measurements on test benches or virtual testing of single components as well as component assemblies are integrated in the development process as early as possible. Later in development, CFD simulations of the full vehicle with detailed underhood flow and wind tunnel measurements complement the determination of the cooling air mass flow.

An example for such an integrated procedure and some selected results are presented and discussed for the Porsche Carrera.

INTRODUCTION

Today the OEMs are faced with various development targets and limitations, e.g. increasingly stringent thermal management requirements which encompass thermal reliability, reduced fuel consumption, low emissions and thermal comfort of the passengers [1, 7, 10]. The cooling system design has to ensure a sufficient cooling air mass flow, so that not only the radiator but also the numerous other heat exchangers (e.g. engine oil cooler, gearbox oil cooler, condenser, etc.) can meet their functional requirements. Furthermore, the sole consideration of steady state operating conditions, e.g. maximum speed or hill climb, is no longer sufficient. Transient operating conditions like the NEDC (New European Driving Cycle - consisting of 4 repeated inner city driving cycles and an extra-urban driving cycle) or race track (in case of the Porsche Carrera) have to be accounted for [11,12,13]. The same is true for various ambient temperatures and altitudes.

Concept decisions like the dimensions and necessary performance data of heat exchangers or the position and size of the air inlet(s) and outlet(s) have to be made early in the concept phase [18]. Any design modification later in the vehicle development cycle leads to an increase in costs resulting from possible production tool changes or necessary add-on parts. To meet all these requirements the hardware driven development process of a new vehicle has to be extensively complemented by virtual testing.

**Documents**
Document Sharing Hub

Fig. 1 shows the simplified development process of a new vehicle at Porsche. The concept phase is divided into 2 stages. During the proportion studies different basic dimensions of the vehicle are analyzed. First models are realized as mock-up variants in both the model scale wind tunnel and CFD.

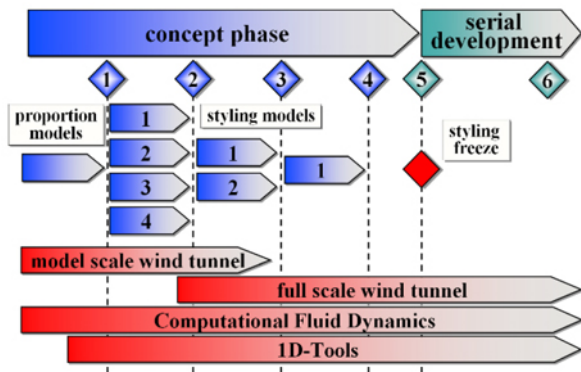


Figure 1. Vehicle development process and the available development tools

At the beginning of the styling phase the competing designers integrate recesses for the not yet defined cooling air inlet(s) in the front-end of their vehicle concepts. At this point the engineer should be able to interact with the designers. Therefore, the necessary dimensions of the air inlets have to be known. Since the inlet size and position are a function of the local pressure ratio and the desired cooling air mass flow through the radiators, which is still based on estimations of heat to be dissipated, the engineer needs simulation tools at hand to cover that early stage [2, 6, 15]. Here common commercial 1D tools [16] can help the engineer to calculate the cooling air mass flow. Based on pressure coefficients at the air inlet and outlet of the latest styling surface and known air flow resistances of the cooling air duct from a predecessor, a qualitatively good estimation regarding the expected cooling air mass flow can be made [20].

Another approach to assess the performance of a cooling system was presented by Williams et al. [21]. He introduced the so-called ram curve which represents the pressure recovery / loss of the vehicle front-end including the inlet openings, underhood and the underbody. He showed that the projected inlet area (frontal projection of the radiator through the front-end structure) plays a major role in the inlet aerodynamic performance. The introduced ram curve is independent of the fan / shroud system installed in the vehicle. Due to different packaging with cooling air inlets positioned outside of the stagnation point, this approach is not easily transferable to the vehicle presented here.

This paper presents a short overview of selected development tools regarding the cooling air mass flow at hand and points out their benefits in the different development stages. Focus

lies on the concept phase when only little data of the new vehicle is available but nevertheless essential decisions concerning the cooling system are required to be made.

DETERMINATION OF THE COOLING AIR MASS FLOW

Fig. 2 shows the cooling air path of the Porsche Carrera. Traditionally the Carrera has a rear mounted engine with the trunk, fuel tank and heat exchangers in the front. The cooling air path consists of the three air inlets in the front-end. One on each side plus one in the center that diverts the air to the sidewise air ducts. The two cooling modules are in each case made up of one climate condenser, one coolant radiator and one electrically driven fan and air flaps in the shroud. The two air outlets are placed in the wheel houses in front of the wheels and consist of five vertical slots.

Leakage in front of the radiator is limited to a minimum through tight packaging and a closed air duct till the cooling module. Behind the radiator, leakage due to assembly tolerances leads to an increase in cooling air mass flow compared to a fully sealed air path.

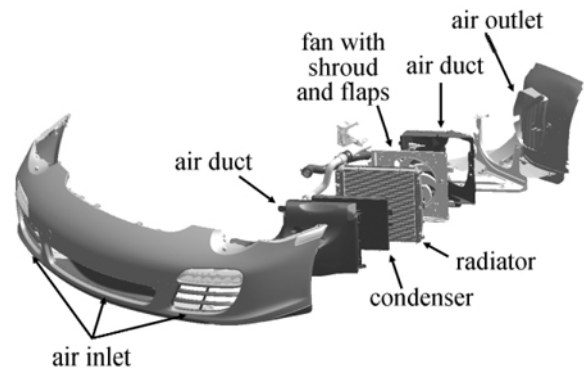


Figure 2. Cooling air path of the Porsche Carrera

The following sections give a brief summary of the advantages and constraints of the common development tools (wind tunnel, CFD and 1D) in an integrated process to determine the cooling air flow rate in the different vehicle development stages. The choice of tool is thereby a function of data availability and has in turn great influence on the efficiency of the development process. Since the two air paths of the Porsche Carrera are symmetrical, only the results for the left air path are shown in this paper.

MEASUREMENT OF THE COOLING AIR MASS FLOW

The determination of cooling air mass flow in the wind tunnel yields advantages like good accuracy, widely spread acceptance, the possibility to test and optimize different air path configurations in a short time and is still used as a

reference for CFD. The first results, on the other hand, cannot be expected before gate 2 (cp. fig. 1) since models with detailed underhood flow are not available before then.

With detailed models at hand, the cooling air mass flow can be measured by specially prepared radiators. Cylindrical pressure probes mounted directly in front and behind the radiator (fig. 3 left) are used to measure the static pressure difference along a streamline through the radiator (fig. 3 right). The cooling air mass flow can then be obtained through a correlation with the before calibrated module (radiator with probes) on a flow test bench.

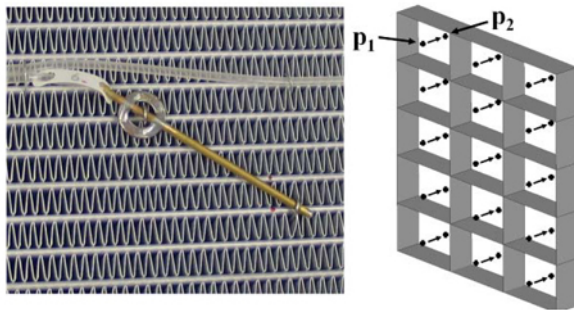


Figure 3. Static pressure probe on the radiator (left) and functional principle (right)

The number of probes has to be considered thoroughly, since the discretization of the radiator surface has a not negligible influence on the measurement results. At this point, flow inhomogeneity [17] defined as

$$i = \frac{1}{N} \sum_{i=1}^N \frac{|v_i - v_m|}{v_m}, \quad \text{eq. (1)}$$

where v_i is the local and v_m the average velocity, can be used to analyze the influence of probe numbers. Fig. 4 shows for a virtual test case (CFD results with virtual probes) the differences in flow inhomogeneity and cooling air mass flow as a function of used pressure probes referred to a detailed resolution with 100 probes. The decrease in flow inhomogeneity (right axis) with smaller probe numbers leads to a reduced capability to resolve local gradients and has therefore an effect on the cooling air mass flow determination (left axis).

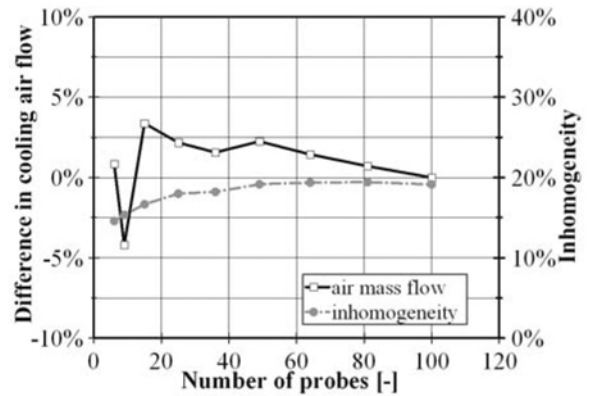


Figure 4. Difference in cooling air mass flow (left axis) referred to 100 probes and flow inhomogeneity as a function of probe numbers. Results are based on a virtual test case at 140 km/h.

In experiment, the needed approx. 50 probes to ensure a constant inhomogeneity would result in a high blockage and higher pressure losses of the module due to the blockage of probes and required tubing. This conflict shows that different configurations (radiator areas and installation positions) can result in different approaches. In the case of the Porsche Carrera investigations showed that 15 probe pairs yield the best results with a relative precision of about 3 % [4, 17, 22]. The achieved accuracy is sufficient since in most cases only differences in air mass flow between different variants are of interest.

The same methodology can also be applied to model scale radiator simulators (screens and combs with different pressure losses fitted in a frame) that are in use in earlier development stages. At this point, focus of this method is on a realistic representation of the pressure drop of the cooling module and the therewith connected representative cooling air mass flow. This way, interference effects of outer and inner flow are accounted for in the aerodynamic development of the basic shape of a new vehicle [10, 22]. The determination of cooling air mass flow is possible but the smaller radiator dimensions increase the before mentioned conflict between necessary discretization and resulting blockage which in sum leads to a loss in achievable accuracy.

3D SIMULATIONS OF THE COOLING AIR MASS FLOW

CFD is widely used in the vehicle development process [1, 3, 9]. 3D simulations enable the engineer to determine the local flow parameters and flow structures around the car as well as in the cooling air path. The significance of CFD is increasing in both the early development and the serial development phase.

Based on detailed tessellated surface data from CAD, the fluid domain is discretized with small volume elements. The size of those elements depends on the proximity to the vehicle and the expected velocity gradients. Smallest fluid cell sizes (< 1 mm) are used in critical regions of the cooling air flow, e.g. at the fan blades and the inlet and outlet grille. Typical cell sizes are in the range of 1 - 6 mm in the vehicle vicinity. The accuracy of the simulation results as well as the total computational time strongly depends on the resolution of the fluid domain and the surface mesh. In daily practice the heat exchangers are not calculated directly since the necessary resolution of the small tubes and baffles would lead to very high calculation times and costs. Therefore, they are simulated as porous media with a pressure drop imposed on the fluid based on test bench measurements [1, 5, 14]. The fans are modeled with a technique called MRF (Multiple Reference Frame). This means that forces consistent with rotation are applied to the fluid within the MRF region (most likely a cylinder around the blades). The fluid velocities are then transformed across the boundary between the rotating frame and the stationary global reference frame. This method is widely used and shows good agreement with experimental data [7, 19].

Fig. 5 illustrates a possibility in CFD to visualize the flow properties in the air path. In this example, fluid streamlines and isocontours of velocity magnitude in the cooling air path are plotted. The engineer can use the derived knowledge of flow separations in the air path and its vicinity to adapt the ducting to increase the cooling air mass flow or reduce cooling air drag.

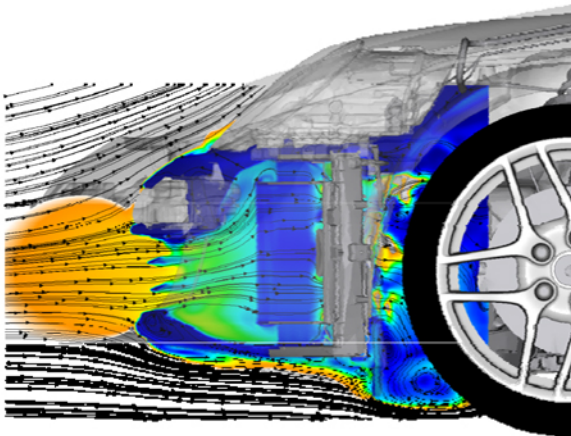


Figure 5. Flow visualization of the air path in CFD

The calculation of cooling air mass flow in CFD is, in contrast to the experimental approach, not limited to a small number of probes. In case of the Porsche Carrera a plane with about 32,000 elements is used to calculate the integral cooling air mass flow. This way it can be ensured that the mentioned inhomogeneity effects are accounted for. Fig 6 shows a comparison of local flow velocity of the original

discretization of the radiator area in CFD (left) and a reduced resolution with the 15 pressure probe locations of the experiment.

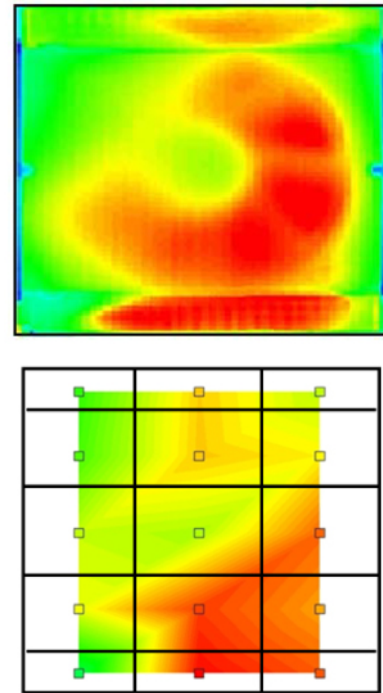


Figure 6. Detailed (top) and discretized (bottom) velocity distribution in the center of the porous media

It can be observed, that strong gradients in the fan area and at the edges of the condenser can not be assessed by the approach of the experiment. As already mentioned, these inhomogeneity effects lead to discrepancies in the calculated mass flow as can also be seen in fig. 7. The difference between the integral results in CFD and the experimental results with the discrete approach (15 probes) is between 7 - 10 % depending on the driving velocity.

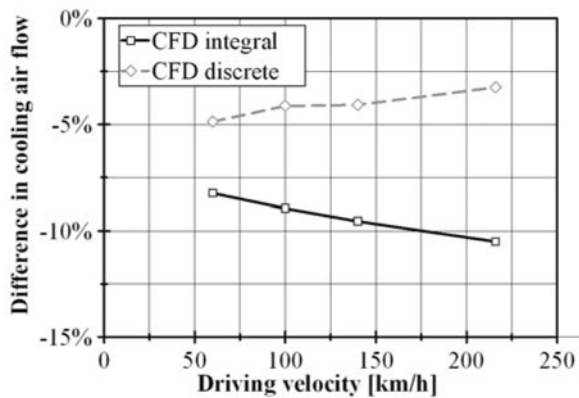


Figure 7. Comparison of cooling air flow between CFD (integral with 32,000 elements and discrete approach with 15 sample points) and experiment

By reducing the number of sample points for the integral mass flow calculation, the difference between CFD and experiment can be reduced to 3 - 5 %. These results provide confidence in CFD as a development tool.

1D CALCULATION OF THE COOLING AIR MASS FLOW

In comparison to wind tunnel measurements and CFD calculations which require a detailed representation of the underhood geometry, 1D simulations are based on simple, interconnected components. In a first step, the characteristic component behavior from a predecessor or first component tests (virtual or on a flow test bench) can be utilized. This data, accompanied by pressure coefficients of either CFD or wind tunnel tests of the latest outer surface of the vehicle, can be used for first rough estimations of the expected cooling air mass flow [2, 15, 20]. One major advantage over CFD is the calculation time. A discrete air mass flow can be calculated in a matter of seconds, whole driving cycles with full vehicle cooling system models including complex engine and gearbox models in a matter of minutes. This advantage makes 1D tools an essential part of today's cooling air development process. The following sections give a short overview of aspects that have a major influence on the cooling air mass flow calculation as well as some ideas how 1D methods can be used in an integrated development process.

Basics of 1D models

1D tools are based on the streamline theory (Bernoulli equation = energy equation of fluid mechanics for attached, smooth and incompressible flow) [16]. The various components of the air path are arranged along a stream filament. At this, every component inside the air path boundaries is modeled as a pressure drop or a pressure source. Driving pressure differences at the air inlet and outlet as well as pressure sources (e.g. fan) in the air path regulate the air mass flow [9, 15]. The components are connected via

pressure nodes as can be seen in fig. 8. The depicted detailed air path model consists of the following components [16]:

- pressure boundary conditions at the air inlet and outlet
- area resistances which represent the pressure loss of components like the inlet grille, bend losses or leakage
- transitions that cover the pressure rise in the air duct
- heat exchangers (here only the radiator, the condenser is simplified as an area resistance)
- resistance matrixes that ensure a realistic flow pattern at the radiators
- electrically driven fans

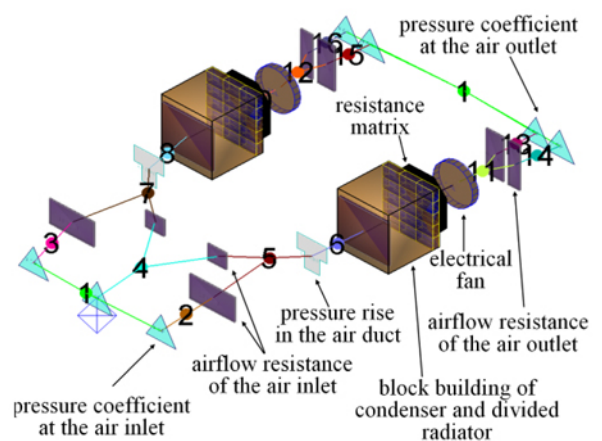


Figure 8. Example of a 1D air path model of the Porsche Carrera

In case of a vehicle cooling system with velocities far below the local speed of sound ($Ma < 0.3$), the medium air can be approximated to be incompressible and the pressure loss of a component in the cooling air path can be expressed as

$$\Delta p_c = \xi_c(Re) \cdot \frac{\rho_c}{2} \cdot v_c^2 \quad eq. (2)$$

where ξ_c is a dimensionless pressure loss coefficient, ρ_c the local density and v_c the local velocity at the component.

The necessary pressure loss data of the components can be obtained from different sources. Component data for heat exchangers or fans come directly from the manufacturer where those components are being tested by default. Other components like the ducting, inlet and outlet grilles (in general all in-house parts) can be obtained from experiments on test benches or virtual testing in CFD. One of the difficulties at this point are interference effects that occur in installation position in the vehicle [20]. Those 3D effects like flow separation and changing approach flow angles are hard