

Role of Accurate Numerical Simulation of Brake Cooldown in Brake Design Process

Devadatta Mukutmoni, Sacha Jelic, Jaehoon Han and Michael Haffey
Exa Corporation

ABSTRACT

An important metric in the vehicle brake design process, the cool-down time for a brake disk, strongly influences the durability and reliability of brakes. However, the brake cool-down time is a function of many vehicle and chassis factors, making it time consuming and expensive to evaluate and optimize in hardware testing.

In this study, we investigate an alternative approach to hardware testing for evaluating brake design cool-down time by implementing a CFD (Computational Fluid Dynamics) simulation based methodology. The simulation cases were all compared with test data and good agreement was observed between test data and simulation over a wide range of design parameters. It is therefore demonstrated that accurate simulation is a promising new approach to the brake design process.

CITATION: Mukutmoni, D., Jelic, S., Han, J. and Haffey, M., "Role of Accurate Numerical Simulation of Brake Cooldown in Brake Design Process," *SAE Int. J. Passeng. Cars - Mech. Syst.* 5(4):2012, doi:10.4271/2012-01-1811.

INTRODUCTION

In design process for brake components, a significant factor that must be accounted for is the time taken for the brake disc to cool down from one prescribed temperature to a lower temperature as the vehicle is driving. If cool-down time is too slow, brake discs could overheat, leading to serious brake quality issues such as brake fade, squeal, judder and increasing wear from thermal stresses. Therefore, OEMs and brake manufacturers have stringent requirements for cool-down time in order to improve brake reliability and reduce warranty costs.

The brake disc cool-down time is strongly influenced by the size, location and placement of the brake components, vehicle driving conditions, material properties of the brake parts and ducting strategies to direct cooling airflow to the brakes. The large number of parameters that influence cool-down time creates a unique challenge in the brake design process since design optimization from a testing only process could potentially be very expensive and time consuming. Furthermore, due to the many design inputs, the brake cooling design process becomes "non-intuitive" to even experienced brake development engineers thus making it challenging to define a base design configuration that may meet cool-down targets based on engineering judgment or experience. Additionally, the underlying flow structure in the brake components area that provides the cooling airflow is complex, three-dimensional and unsteady and does not lend itself to easy generalizations and predictability.

In this study, we look into an alternative approach to the hardware/test brake design process via numerical simulations. We demonstrate that air-flow and component temperatures can be accurately modeled in a CFD (Computational Fluid Dynamics) simulation and the cool-down time can be calculated as well. A simulation based process has several advantages over testing procedures such as being less expensive and time consuming. Speed and cost of simulation based processes are limited only by computer hardware resources that are getting faster and cheaper. Visualization of simulation results provides the design engineer insight into the flow field and how the design can be improved. Also, a simulation process is easily extendable to a formal optimization process to reduce brake-cooling time for a given design space and set of constraints.

In this study, we numerically investigate the transient temperature and flow field for several cases of a production vehicle for vehicle speeds ranging from 100 km/hr to 180 km/hr. We also look into sensitivity of the cool-down time to geometry changes such as open and closed grilles.

The simulations were performed using a coupling between the flow solver PowerFLOW version 4.3d and a thermal solver PowerTHERM version 10.0.1 that accounts for conduction and radiation effects. The experiments were performed at Forschungsinstitut für Kraftfahrwesen und Fahrzeugmotoren Stuttgart (FKFS), Stuttgart for an Audi Sedan as shown in [Figure 1](#). Details of the experiments are provided in [1].



Figure 1. Experimental Set up for Test Vehicle (Audi Sedan)

Brake Cooling simulations have been a subject of considerable interest given its relevance to the design process. Shen et al. [2] simulated airflow in a brake corner assembly and calculated the cool-down time using a Thermal Analytical Model using input from a CFD Model. Lee [3] applied a similar methodology and extended it to Brake Duty Cycles. In recent times, Brake Simulations have become more detailed in terms of mesh resolution and physics [4, 5]. In this study, we go further along in detailed modeling of the physics and geometry. The current set of simulations use a Solid Brake Model so that the conduction paths are more accurately represented leading to more accurate results. Furthermore, the methodology takes into account variation of heat transfer coefficients of the brake components as a function of the surface temperature in the entire duration of cool-down. Also, geometry modeling is done more rigorously to ensure an accurate coupling between the flow and temperature solver.

In the next section, Lattice Boltzmann Methodology (LBM) and turbulence modeling used in the flow solver is outlined. Following that, the test bench configuration and simulation methodology are described in some detail. We conclude with a discussion of the simulation results.

LATTICE BOLTZMANN METHODOLOGY

The flow solver is a general purpose CFD code that uses the Boltzmann Equation to solve for the velocity and pressure field [6]. The dependent variable is the distribution function $f(\mathbf{x}, \mathbf{v}, t)$, the number density of molecules at position $\mathbf{x}$ and speed $\mathbf{v}$. The Boltzmann equation can be written in the following form:

$$\frac{\partial f}{\partial t} + \mathbf{v} \cdot \nabla f = C(\mathbf{x}, \mathbf{v}, t) \quad (1)$$

The left hand side represents the convective motion of particles and the right hand side the collision term that accounts for intermolecular interactions. The flow solver uses the BGK collision operator [7] that can be expressed as follows:

$$C(\mathbf{x}, \mathbf{v}, t) = -\frac{f - f_{eq}}{\tau} \quad (2)$$

Where τ is the relaxation time of the fluid and f_{eq} is the Equilibrium Maxwellian Distribution Function. The Boltzmann Equation along with the BGK operator is discretized on a three dimensional cubic lattice using a D3Q19 model as shown in Figure 2. The resulting finite difference equation enables the solution of the distribution function.

It can be shown rigorously that the Lattice Boltzmann formulation thus outlined is equivalent to solving the time dependent compressible Navier-Stokes Equation. A review of the fundamentals of the Lattice Boltzmann Method can be found in [8].

The flow solver incorporates the Lattice Boltzmann Solver with an RNG $\kappa - \varepsilon$ type Turbulence Model for the small unresolved scales. The turbulence model used is inherently unsteady with low numerical diffusion and therefore complex three dimensional unsteady physics is very well represented. Furthermore, the Turbulence Model includes an advanced Wall Function Model valid for very low y^+ values and adverse pressure gradients. These features enable the model to be very robust over a wide range of mesh

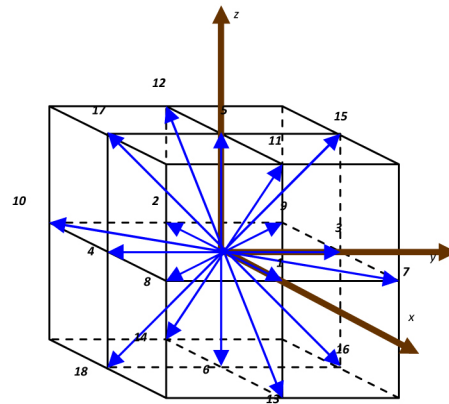


Figure 2. Finite Difference Scheme for Boltzmann Equation

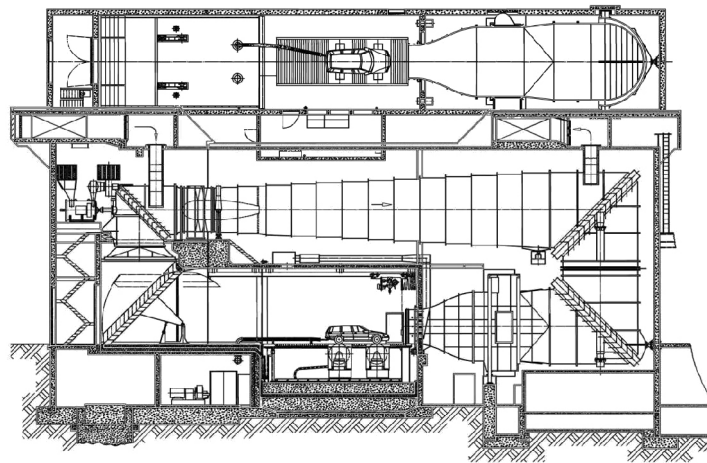


Figure 3. Schematic of FKFS Thermal Wind-Tunnel

sizes and be able to capture flow separation accurately. Further details on turbulence modeling are available in [9].

TEST BENCH

The experiments were performed in a thermal wind-tunnel at FKFS; the schematic of the Wind-Tunnel is shown in [Figure 3](#).

Some of the brake cooling tests were repeated in Audi Testing Ground as shown in [Figure 4](#) which represents open road testing.

A comparison of the brake cool-down test results show that difference in the cool-down time between the open road test and the thermal wind-tunnel test was not significant. Therefore, thermal wind-tunnel testing at FKFS was able to accurately replicate open road conditions. Another point to be noted is that there is an uncertainty in the measured cool-down time with respect to repeatability of the test data. Based on testing done at FKFS, brake cooling testing uncertainty is estimated to be a maximum of 2% due to measurement repeatability in the experimental setup.

In the cool-down testing, the brake disk was heated to a temperature of 600 C and cool-down time reported was the time taken for the brake disk to decrease in temperature from 500 C to 150 C at constant air and dynamometer speeds.

SIMULATION METHODOLOGY

The simulation methodology used in this study involves a combination of the flow and thermal solver. The flow solver calculates the flow and temperature fields in a volumetric flow domain and the thermal solver calculates the temperature in the solid surfaces by taking into account radiation from the surfaces, conduction within the solid surfaces and convection at the surface.

This process is enabled by a coupling process methodology illustrated in [Figure 5](#).

The flow solver calculates the near wall temperatures, T_f , and heat transfer coefficients, H , and exports these values to the thermal solver; the thermal solver calculates the wall temperatures of the solid surfaces, T_w by taking into account radiation, conduction in solid surfaces and convection from the heat transfer coefficients and near wall temperatures and



Figure 4. Audi Testing Ground

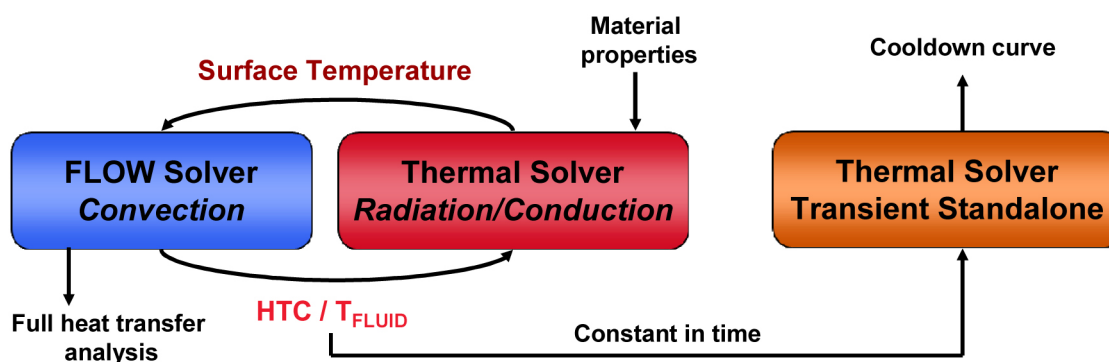


Figure 5. Schematic of the coupled flow and thermal solvers methodology and cool-down calculation

exports the surface temperatures back to flow solver. This “coupled process” continues until convergence is achieved for the solid surface temperatures. The converged heat transfer coefficients and near wall temperatures are exported to a standalone Thermal Model to simulate the cool-down.

The coupling process uses two different geometrical models one for the flow solver and the other for the thermal solver that have the same underlying geometry but different element types and size. The transfer of T_w , H and T_f between the flow and thermal solver is achieved by interpolation between different geometries. In this study, the coupling geometries are limited to brake and wheel components and are shown in Figure 6.

The thermal solver employs a finite-volume formulation that exactly conserves energy. The surface to surface radiation is calculated by solving a grey body radiation enclosure problem and calculating the radiation view factors by a ray-tracing algorithm. The conduction is calculated in the context of a finite volume formulation. More details of the methodology can be found in [10].

SIMULATION SET UP

The flow and temperature simulation is set up in a flow domain that represents a vehicle moving in a set up referred to as a “digital wind-tunnel” that represents a large enough domain or “wind-tunnel” to accurately model open road conditions (Figure 7). Symmetry was used in the y-direction to limit the flow domain and hence computational costs.

The flow solver discretizes the volume flow domain into mostly cubical elements known as voxels. The surface geometry is discretized into surface elements known as surfels. Furthermore, in order to optimize the flow resolution, the voxel size is varied over the entire flow domain so that more resolution is provided in regions where required. For instance, small gaps in the vehicle grille inlets would require smaller voxel element size than in regions far away from vehicle.

The voxel size distribution is specified by volumetric zones known as VR (Variable Resolution) regions that specify voxel size within the zone. Figure 8 shows the voxel size distribution around the vehicle and in Figure 9 we see the size distribution around the brake assembly.