

Convective Heat Transfer Optimization of Automotive Brake Discs

Zhongzhe Chi, Yuping He and Greg Naterer
University of Ontario Institute of Technology

Copyright © 2009 SAE International

ABSTRACT

Under intensive braking, such as continuous down-hill braking, high temperatures could be generated in automotive brake disks. The heat dissipation and thermal performance of vented brake discs strongly depends on the aerodynamic characteristics of the air flow through the rotor passages and the geometry configurations of brake discs. In this paper, commercial software GAMBIT is used for geometrical modeling and automatic mesh generating for brake rotors. Then, a computational fluid dynamic package, FLUENT, is employed to simulate the turbulent motions of air flow through the vented discs. Through the numerical simulations, the design criteria regarding the heat transfer rate and air flow rate of the discs are predicted. To optimize the 2-D and 3-D geometrical configurations of the brake discs, commercial software iSIGHT is used to integrate the geometrical modeling with GAMBIT and numerical simulations based on CFD software FLUENT. With the design of experiment studies implemented through this integrated design synthesis process, the thermal performance of brake rotors is greatly improved by optimizing disc outer and inner radii, vane numbers, vane angles, and the radius of vane curvature.

INTRODUCTION

Automotive braking systems are one of the most important safety systems of road vehicles. They are mainly used to decelerate vehicles from an initial speed to a given speed. Excessive thermal loading can result in surface cracking, judder and high wear of brake rubbing surfaces. High temperatures can also lead to

overheating of brake fluid, seals and other components [1, 2]. Therefore, one of the main tasks of braking system design is to reduce the surface temperature of brakes [3].

Much attention has been focused on improving the thermal performance of brake discs. Numerical simulations and Computational Fluid Dynamics (CFD) in particular are commonly applied to braking disc thermal performance analyses [5]. Many experimental studies have also been conducted to measure the air flow and temperature field inside the discs in the application of braking [6, 7, 8]. It has been demonstrated that CFD simulation results have achieved good agreement with those based on experimental studies.

Various CFD tools for a computer-aided design have been applied to the development of brake discs. For example, FLUENT has been used by the automotive industry to model and design various configurations of brake discs [5]. From a design point of view, the drawback of most commercial packages such as FLUENT is that they only provide analysis of the brake discs whose design variables have been specified. Design syntheses including parametric and sensitivity analyses have been difficult. Instead, engineers must decide by trial and error to change design variable values and re-perform the analysis until a set of performance criteria becomes acceptable. This "manual" process, often accompanied by prototype testing, can be difficult and time-consuming for complex systems, such as vehicle disc brakes.

However, little attention has been paid to the potential of a comprehensive automated design synthesis process of brake discs [4]. Numerical optimization may help

automate the design synthesis by altering variable values in a search to optimize performance criteria, subject to constraints. Due to the complexity of the CFD model of brake discs, even a single iteration of the optimization may take a large amount of time. However, if these optimizations are implemented in a massively-parallel computer system, the computation time could easily be reduced approximately by a factor equal to the total number of slave computers.

To explore the potential of an automated design synthesis process of brake discs, in the current paper, an integrated design synthesis has been proposed and implemented. Commercial software GAMBIT is used for geometrical modeling and automatic mesh generation for vented discs. Then, the CFD package, FLUENT [9], is employed to simulate the air flow through the vented disc. To automate the design process of the disc rotor, a software framework, iSIGHT [10], is used to integrate the geometrical modeling using GAMBIT and numerical simulations based on FLUENT. Through this integrated design synthesis process, the disc rotor geometrical shape optimization is performed with design of experiments (DOE) studies. The main design criterion of the geometrical optimization is to maximize the convective cooling by increasing the heat transfer rate inside the rotor passages.

GEOMETRICAL OPTIMIZATION FRAMEWORK

The geometrical optimization was implemented by integrating geometrical modeling and CFD numerical simulations. The framework for automated design synthesis is depicted in figure 1. This framework consists of the following three software packages: GAMBIT, FLUENT and iSIGHT.

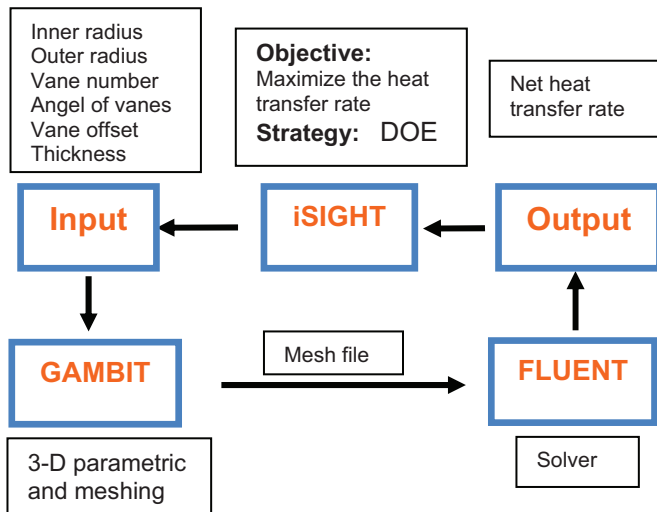


Figure 1: Schematic representation of the framework for automated design synthesis of brake discs

GAMBIT is a geometrical modeling and automatic mesh generation software package. With various input parameters such as inner radius, outer radius, vane

number, vane angle and vane offset, GAMBIT generates various mesh files for CFD and other numerical simulations.

FLUENT is an effective CFD solver, which is used to simulate the complex turbulent air flow inside the rotor passages.

iSIGHT can enable the rapid integration of commercial CAD/CAE software, internally developed programs, and Excel spreadsheets and automate their execution to accelerate the evaluation of many more design alternatives. In addition, it provides leading edge design exploration and optimization technology to ensure that an optimal design is discovered using techniques such as optimization, approximations and design of experiment (DOE) [10].

As an integration and optimization tool, iSIGHT can integrate GAMBIT and FLUENT, leading to optimal design parameters. With a given set of input parameters, such as the inner radius, outer radius, vane number, vane angle, vane offset, and rotor thickness, from iSIGHT, GAMBIT will build geometrical models and generate corresponding mesh files. Then, FLUENT will call these meshing files to calculate specified performance indices, such as heat transfer rates. The resulting performance indices will be returned to iSIGHT. Through objective fitness evaluations, iSIGHT will coordinate the trade-off relations among various design criteria and constraints and identify a set of better design variables. The resulting design variable set will be forwarded to GAMBIT as the new input parameters for the next iteration. The above procedure will repeat until an optimal set of design variables is obtained [1].

NUMERICAL SIMULATIONS WITH CFD

In order to achieve accurate results, in the current study a 3-D disc rotor model has been generated and the corresponding numerical simulations with CFD have been conducted. Figure 2 schematically shows the 3-D disc rotor model.

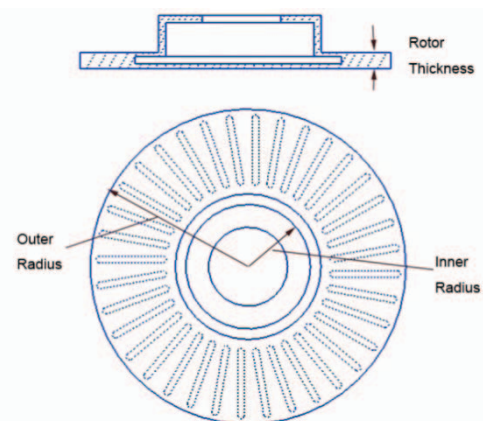


Figure 2: Representation of the 3-D rotor model

In the numerical simulations and design optimization, the design variables include the rotor thickness (RT), inner radius (IR), outer radius (OR), vane number (VNu), vane angle (VA), and vane offset (VO). The design variables RT, IR and OR are defined geometrically in figure 2. Note that the rotor thickness is the total distance between the two disc brake surfaces that touch braking pads. When rotor thickness changes, the brake plate thickness remains constant and the cooling passage width varies accordingly. VA is defined as the angle between the axis of a vane and the corresponding radial line in the rotor plane, which is perpendicular to the rotating axis of the rotor. If the rotor is rotating clockwise and the axis of the vane is oriented counterclockwise with respect to the corresponding radial line, the vane angle has a positive sense. As shown in figure 3, the vanes in the left and right take the vane angle with the values of $+10^\circ$ and $+30^\circ$, respectively. VO is used as a measurement for the curvature of a curved vane. As shown in figure 4, it is defined as the offset of the middle point of the curved vane from its straight vane counterpart, assuming that the two ends of the former coincide with those of the latter. Figure 5 shows two curved vanes with different offsets. With the introduction of the periodic side boundary conditions, the whole disc rotor can be modeled as a partial section, as shown in

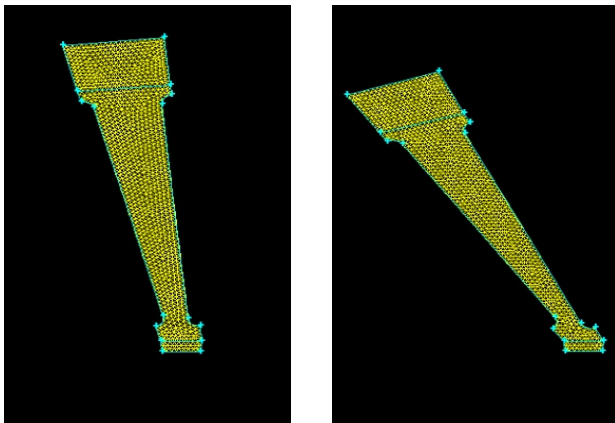


Figure 3: Representation of vanes with the vane angles of $+10^\circ$ (left) and $+30^\circ$ (right) [1]

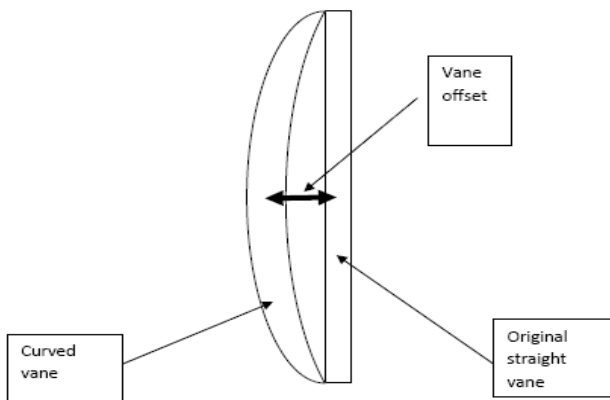


Figure 4: Geometrical definition of the offset of a curved vane

figure 6. Since the vanes are evenly arranged along the whole disc rotor, a given vane number corresponds to a specified section angle. The vane number can be changed by varying the section angles. When changing the vane number, the vane thickness remains constant.

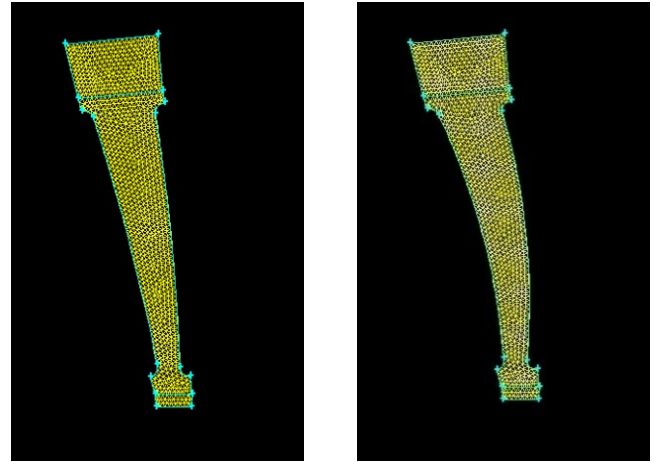


Figure 5: Representation of vanes with the offsets of 1 (left) and 3 mm (right) [1]

The 3-D mesh model was constructed with GAMBIT. Due to the axial symmetric of the rotor, a partial cooling passage was generated with boundary conditions applied on both sides, as shown in figures 6, 7 and 8.

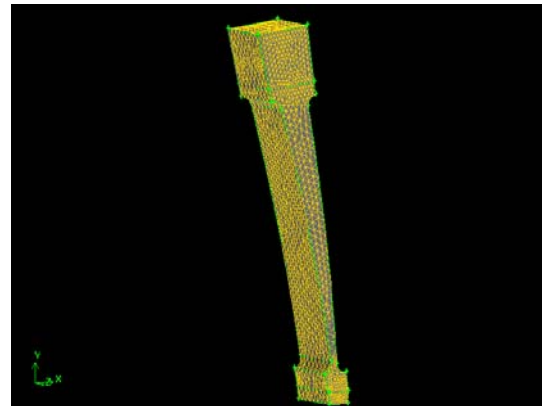


Figure 6: 3-D mesh produced by GAMBIT

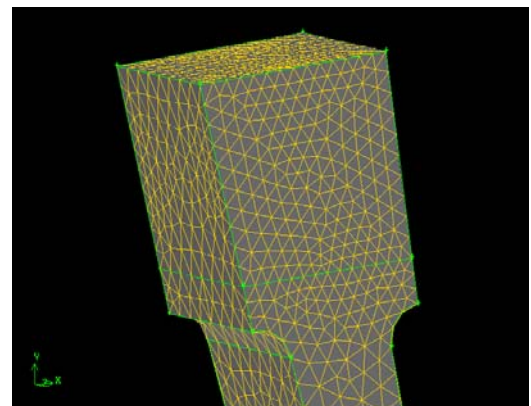


Figure 7: Outlet section of a 3-D section produced by