Palmer, Edward, Mishra, Rakesh and Fieldhouse, John D.

Analysis of air flow and heat dissipation from a high performance passenger car front brake rotor

ANALYSIS OF AIR FLOW AND HEAT DISSIPATION FROM A HIGH PERFORMANCE PASSENGER CAR FRONT BRAKE ROTOR

E. Palmer, R. Mishra and J. Fieldhouse
[1] University of Huddersfield, Queensgate, Huddersfield HD1 3DH, UK

ABSTRACT

The increasing demand from the consumer for higher levels of refinement from their passenger vehicles has put considerable pressure on the automotive industry to produce ever quieter cars. In order to prevent the occurrence of many forms of brake noise, especially judder and drone, excessive heating of the brake disc must be avoided, whilst minimising temperature variations across the rotor. In order for this to be achieved the brake rotor must be designed such that it ensures sufficient uniform heat dissipation and thermal capacity.

In high demand braking applications vented discs consisting of two rubbing surfaces separated by straight radial vanes are normally employed as they utilise a greater surface area to dissipate heat. Within this paper the convective heat dissipation from a high performance passenger car front brake disc has been investigated using computational fluid dynamics (CFD). The results obtained have been validated by those obtained in preliminary vehicle testing at Millbrook test facility. The computational model shows adequate correlation to the test results; overpredicting the average heat transfer coefficient by 18%. The CFD analysis enabled a detailed insight into the air flow and heat transfer distributions, that was not possible during the vehicle test regime

Keywords: Judder, Brakes, CFD, Heat Transfer,

1 INTRODUCTION

Due to the increased refinement of passenger vehicles the expectations and demands of the consumer have risen. Now even in high performance applications the highest level of refinement has become expected. This has put considerable pressure on the automotive industry to produce ever quieter cars, none more so than in the area of brake refinement. Of the classifications of brake noise, judder is now regarded as the most concerning of all brake problems, accounting for 75% of brake refinement issues [1]. This translates to a cost to the industry of $100 million (US) a year in warranty claims alone, so the magnitude of this problem is evident [2].

The mechanism of judder is a forced vibration with a frequency directly related to wheel speed. The vibration emanates from a variation in brake pressure caused by disc thickness variations, which results in high amplitude vibrations of the wheel hub and suspension system. This vibration translates to the driver and occupants as a strong pulsation of the brake pedal which can be accompanied by vibration of the steering wheel and floor pan of the vehicle leading to a 'drumming' or 'droning' sound within the cabin. Judder maybe categorised into one of two headings; cold or hot judder, but by far the most complex is hot judder. The disc thickness variation in hot judder is due to the thermal deformation of the rotor or an effect known as 'hot spotting' [3]. It is clear that in both instances, to prevent the occurrence of judder, excessive heating of the brake disc must be avoided, whilst minimising temperature variations across the rotor. In order for this to be achieved the brake rotor must be designed such that it ensures sufficient uniform heat dissipation and thermal capacity

The work presented in this paper studies the convective heat dissipation properties of a high performance passenger car front disc using computational fluid dynamics and experimental vehicle testing. The modeling and validation are both important aspects in gaining a more detailed understanding of the mechanisms of brake judder and drone. The analyses are considered a basis for further improvements to rotor design through determining a methodology to reduce brake judder and drone.

2 EXPERIMENTAL METHODOLOGY

An essential part of any computational fluid dynamic analysis is validation against a data set which is both repeatable and accurate. Within the field of automotive brakes there are two main sources of data for validation; the brake dynamometer and vehicle testing. The brake dynamometer however does not offer a true picture of the aerodynamic performance of the brake rotor due to the vast differences in the surrounding geometry of the disc which is known to have a large effect on disc performance [13]. Therefore it is necessary to perform vehicle testing in order to quantify the differences between dynamometer testing, computational analysis and real world application.

The vehicle test procedure was devised to replicate the energy inputs of a manufacturer's standard judder performance test within the restraints of testing in the UK. The test consists of a number of high speed low deceleration braking events. High speed stops are used in order to quickly dissipate a large amount of heat energy into the brake system therefore giving the potential for large thermal gradients within the rotor. This large input of energy into the brake rotor provides the ideal conditions for the thermo-elastic deformations that can lead to occurrence of judder. Low deceleration rates are utilised in order to prevent pressure from the braking system 'ironing out' deformations in the rotor and maximising the chances of the vehicle occupants to experience the symptoms of judder.

The vehicle testing equipment included:

- Embedded thermocouples positioned half across the face of both the inner and outer rubbing surfaces of the front brake rotors, connected via slip rings provided an accurate measurement of the disc surface temperatures. The exact positions of the thermocouples are shown in figure 2.
- A rubbing thermocouple mounted on the left hand rear brake rotor provided a reference temperature.
- Two axis accelerometers mounted on the brake calliper and inner brake pad back plate provided information on the tangential and axial vibrations.
- High frequency pressure transducers to measure fluctuations in the brake fluid pressure, allowing fluctuations caused by disc thickness variations to be analysed.
- Vehicle speed was provided via a GPS system.
- Vehicle deceleration.
- Pedal effect and travel.
- A modular data logging system capable of recording the 19 channels of data required in real-time.
- A laptop to enable real-time viewing of brake temperature, vehicle speed and vehicle deceleration. This increased the repeatability of the test procedure and reduced the level of experimental error.

3 COMPUTATIONAL METHODOLOGY

The accuracy of mass flow rate and heat dissipation predictions has been vastly improved over semi-empirical predictions by the advances in CFD; now commercially available codes are capable of giving highly accurate results for the flow field and cooling rate of brake discs [5][6][8]. The CFD programs employed in this analysis are the *Fluent 6.0* solver and *Gambit 2.0.0* pre-processor produced by Fluent Inc.

Fluent uses the finite volume method to solve the fluid flow transport equations in integral form and is known for its robustness in simulating many fluid dynamic phenomena. The Navier-Stokes equations form a system of non-linear partial differential equations describing momentum transport for Newtonian fluids. When coupled with the conversation equation and the equation of state it is possible to solve this set of equations to obtain a description of the flow field. For isothermal flow the energy equation can be ignored. Therefore for three dimensional flow problems the equations that are used are as given below, where u, v and w are velocity components in the directions x, y and z, ρ is the density, η is the dynamic viscosity and p is pressure.

Continuity Equation:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0$$

Navier-Stokes Equations:

$$\rho X - \frac{\partial p}{\partial x} + \eta(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}) = \rho(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z})$$

$$\rho Y - \frac{\partial p}{\partial y} + \eta(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}) = \rho(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z})$$

$$\rho Z - \frac{\partial p}{\partial z} + \eta(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}) = \rho(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z})$$

The finite volume method consists of three stages; the formal integration of the governing equations of the fluid flow over all the (finite) control volumes of the solution domain. Then discretisation, involving the substitution of a variety of finite-difference-type approximations for the terms in the integrated equation representing flow processes such as convection, diffusion and sources. This converts the integral equation into a system of algebraic equations, which can then be solved using iterative methods [4]. The first stage of the process, the control volume integration, is the step that distinguishes the finite volume method from other CFD methods. The statements resulting from this step express the 'exact' conservation of the relevant properties for each finite cell volume. This gives a clear relationship between the numerical analogue and the principle governing the flow.

To enable the modelling of a rotating body (in this case the disc) the code employs the rotating reference frame technique. The implementation of periodic boundary (cyclic symmetry) conditions meant that only a periodically repetitive 27-degree section of the disc, shown in figure 1, was modelled rather the whole disc. This had the effect of making the model much smaller, translating to lower hardware requirements and saving vast amounts of computational time.

The disc was modelled as rotating in still air by implementing atmospheric pressure and a temperature corresponding with vehicle test data of 14.72^{0}C, at the inlet and outlet boundaries. The walls of the disc are represented as smooth walls at a constant temperature of 500^{o}C. . Although this is a massive simplification of most real life braking events it is deemed an adequate method for the simulation of brake discs in CFD as previous studies by L Wallis and M Tirovic have shown strong correlations of this arrangement with experimental data [5] [6]. Symmetry boundary conditions have been used to generate zero-shear slip walls at the edge of the domain [7].

The speed of rotation for the simulation was taken as 1140rpm corresponding to the test vehicle speed that correlates to a rotor temperature of 500^{o}C.

The inlet velocity for the cases considered in this investigation has been calculated using empirical equations derived by Limpert [10] which yields a Reynolds number greater than 10^5 for the flow through the channels; hence the flow is considered turbulent. To model this the standard K-ε turbulence model is employed in this study as it has shown to give accurate flow predictions in previous work [5][6][8].

Through the use of a mesh independence analysis and volume mesh optimisation using a fine mesh near the walls of the disc and coarse mesh near the boundaries of the domain, the variation in results was found to become insignificant with approximately 1.3 million tetrahedral elements.

The solution was obtained on an AMD 3800+ Athlon 64 machine with 2GB of system RAM using a steady state solution scheme in a run time of approximately 10 hours.

4 RESULTS & DISSCUSSION

Experimental

The temperature data studied relates to the left hand front disc only. Averaging across the right and left front disc was not suitable due to the different thermal cycles for each disc caused by differing energy inputs created by braking on a bank high speed test track. The handed nature of the discs also requires the correct direction of rotation for meaningful correlation between model and test data.

The data relating to disc temperature, ambient temperature and vehicle speed were able to provide vital information for the generation of accurate boundary conditions such as flow and rotor surface temperature as well as speed of rotation. Implementation of boundary conditions that accurately replicate the real braking events is fundamental if the comparison of results is to be meaningful. For more information regarding boundary conditions implemented see section 3.

The same test data, presented in figure 3, was also able to provide the information required to validate the CFD model. The heat transfer rate and heat transfer coefficient can be calculated from the first derivative of temperature with respect to time. The temperature channels for the inner and outer rubbing surfaces of the rotor. This yields a value for heat transfer rate of 27.5KW and 104.5W/m^{2}K for the heat transfer coefficient. Preliminary findings from the other channels of data are presented in "The Development of a Design Methodology to Reduce the Probability of Brake Judder and Drone due to Thermo-Elastic Instabilities in the Brake Rotor" by David Bryant *et al* [11]

Computational

Figure 4 shows the three dimensional path lines through the rotor. The flow pattern shows the formation of vortices at the inlet generated by the flow tumbling over the vane trailing edge. The flow separates from the leading edge of the vane just after the change in vane angle. However unlike many straight radial vane vented rotors [5] [12] areas of recirculation are avoided, as the separated flow joins the vortices generated at the inlet. This results in a positive radial velocity distribution across the width and length of the vane passage giving the rotor a high pumping efficiency.

Contours of heat transfer coefficient extracted from the CFD study are presented in figure 5. The distribution of heat transfer coefficient in the disc is relatively regular when compared to other types of vented brake rotor. Peak values appear in the regions of stagnation at the vane inlet and lows in the regions of separation. The values vary from 190W/m^{2}K to just below 90W/m^{2}K and the average value is 128.2W/m^{2}K. This is a small range of values when compared to those generated for a pin type disc, where the maximum local heat transfer coefficient can be 300% larger than the minimum. This can be explained by the small range in velocity distribution across the vanes passage resulting from the lack the small regions of separation and lack of recirculation. The resulting heat transfer rate is 33.6KW.

Comparison and Conclusions

The results presented show that the brake rotor studied has very good thermo-aerodynamic properties, avoiding many of the aerodynamic problems associated with rotors of this type such as recirculation. The CFD model over predicted the average heat transfer coefficient and heat transfer rate by 18%, however this is considered an adequate correlation as it is widely accepted that values for predicted and measured heat transfer coefficients can vary by as much as 30% [10]. The reasons for the discrepancy between the two data sets can be explained by the simplifications of the real world made by the CFD model. The most important simplification is the geometry surrounding the rotor; in the model the disc rotates in open air ignoring the effect of the wheel or arch geometry which is known to have a large effect on the aerodynamic characteristics of the disc [13]. Without these features to obstruct the flow the mass flow rate through the disc will be higher than when the rotor is in situ, generating higher heat transfer. The effect of heat transfer through conduction to the hub and radiation is also ignored within the model; nevertheless this is more than compensated by the geometric differences.

It can be concluded from the visual comparison of figure 2 and figure 5 that the thermocouples were well positioned to gain true values for the average heat transfer coefficient. However it should be noted that conduction to the hub is generally credited as the reason for a difference in temperature between inner and outer rubbing surfaces, due to the large effective thermal mass of the inboard face. This is not represented within the CFD model; therefore caution is needed when making such comparisons.

With handed rotors the direction of rotation may have a large effect on the pumping efficiency of the rotor. The early separation of the flow from the leading edge of the vane implies a large adverse pressure gradient. This suggests that when rotated in the opposite direction separation will occur later leading to smaller separation regions; yielding improved thermo-aerodynamic properties.

REFERENCES

1 Thompson J, "State of the Art towards the NVH Robust Brake – USA Position" Brakes 2006 – An NVH Robust Brake Session, 07//05/2006

2 Akebono Braking Technology, "Brake Noise, Vibration and Harshness: Technology Driving Customer Satisfaction", Akebonobrakes.com, 2005

3 Steffen T, "Hot Spot Simulation", IMechE, Brakes 2006 pp 199-208, 2006

4 H K Versteeg & W Malalasekera – An Introduction to Computational Fluid Mechanics, The Finite Volume Method

5 Wallis L, Leonardi E, Milton B, "Air Flow and Heat Transfer in Ventilated Disc Brake Rotors with Diamond and Tear-Drop Pillars" Numerical Heat Transfer, Part A; 41 pp643-655, 2001

6 Tirovic M, Galindo-Lopez C H, "Analysis of Air Flow and Heat Dissipation from Wheel Mounted Railway Brake Discs", IMechE, Brakes 2006 pp 125-134, 2006

7 Fluent Inc. "Fluent User Guide" Fluent Inc. 2003

8 Daudi A & Narain M, "72 Curved Fin Rotor Design Reduces Maximum Rotor Temperature", SAE Paper No.1999-01-3395, 1999

9 Voller G P, Tirovic M, Gibbens P, "Analysis of Automotive Disc Brake Cooling Characteristics" IMechE, vol. 217, Part D: J Automobile Engineering, 2003

10 Limpert R, "Cooling Analysis of Disc Brake Rotors" SAE Paper 751014, 1975

11 Bryant D, Fieldhouse J & Talbot C, "The Development of a Design Methodology to Reduce the Probability of Brake Judder and Drone due to Thermo-Elastic Instabilities in the Brake Rotor" FSITA paper No. F2006V077, 2006

12 Kubota M., Hamabe T., Nakazono Y., Fukuda M., Doi K. Development of a Lightweight Brake Disc Rotor: A Design Approach for Achieving an Optimum Thermal, Vibration and Weight Balance. JSAE paper No. 2000-497-502, 2000

13 Sisson A, "Thermal Analysis of Vented Brake Rotors" SAE paper No.780352, 1978

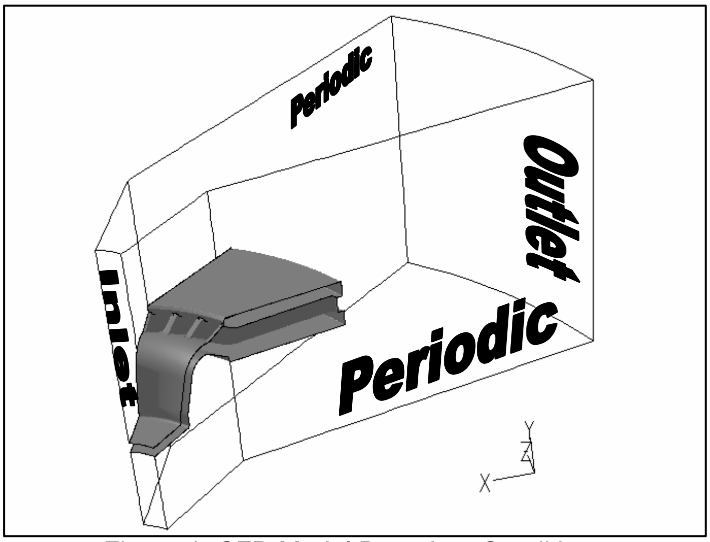

Figure 1: CFD Model Boundary Conditions

Figure 2: Position of Thermocouples in Front Brake Rotor

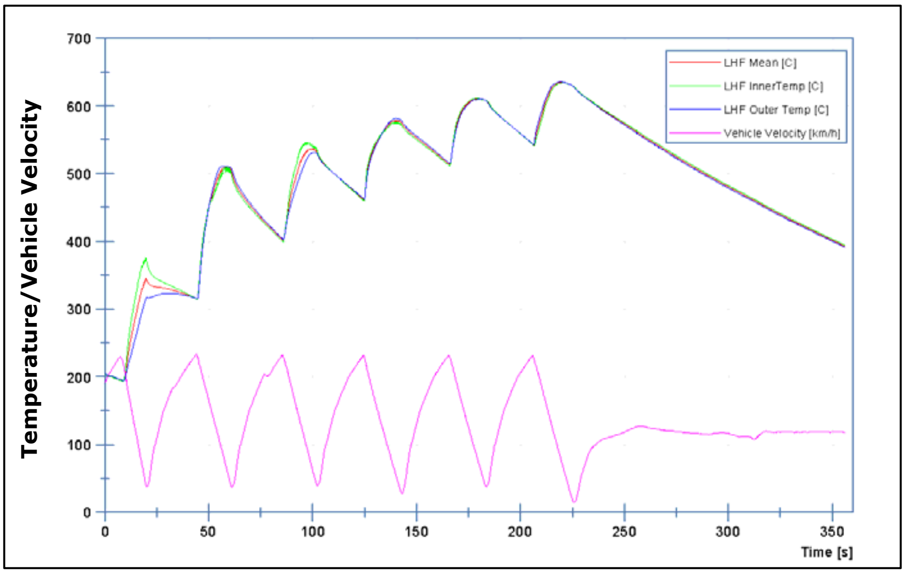

Figure 3: On Vehicle Test Results – Left Hand Front Disc Temperatures

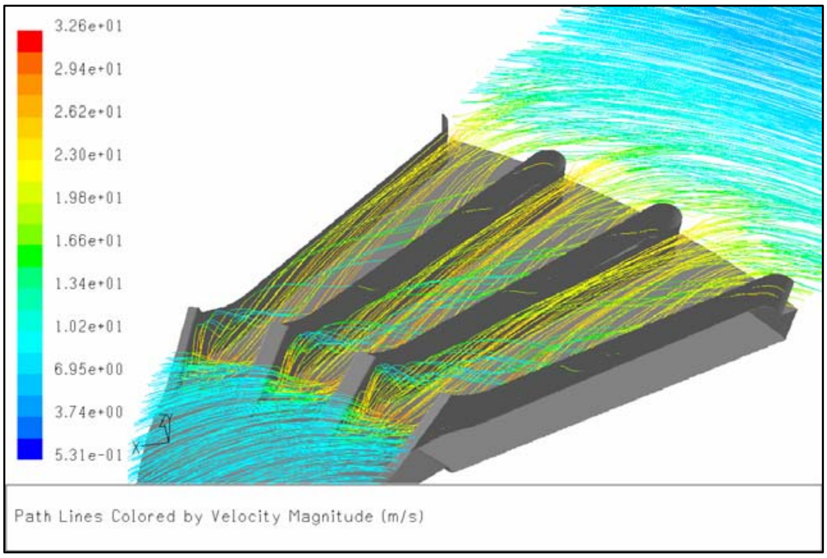

Figure 4: Three Dimensional Path Lines

Figure 5: Heat Transfer Coefficient Distribution on the Disc Surface