Alexandria Engineering Journal (2016) 55, 1127–1141

HOSTED BY

ELSEVIER

Alexandria University

Alexandria Engineering Journal

www.elsevier.com/locate/aej
www.sciencedirect.com

ORIGINAL ARTICLE

A numerical parametric study of mechanical behavior of dry contact slipping on the disc–pads interface

Ali Belhocine [a,*], **Wan Zaidi Wan Omar** [b]

[a] *Faculty of Mechanical Engineering, University of Sciences and Technology of Oran, USTO, L.P 1505 El-Mnaouer, 31000 Oran, Algeria*
[b] *Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM Skudai, Malaysia*

Received 23 January 2016; revised 2 March 2016; accepted 20 March 2016
Available online 31 March 2016

KEYWORDS

Finite element method (FEM);
Ventilated disc brake;
Gray cast iron;
Pads;
Total distortion;
Shear stress

Abstract The aim of this contribution was to present a study based on the determination and the visualization of the structural deformations due to the contact of slipping between the disc and the pads. The results of the calculations of the contact described in this work relate to displacements, Von Mises stress on the disc, and contact pressures on the inner and outer pad at various moments of simulation. We first proceed to view the meshed models and predicting variations of tensile or compressive stress normal to the plane and shear stress in rotating disc and ring bodies. One precedes then the influence of some parameters on the computation results such as rotation of the disc, the smoothness of the mesh, the material of the brake pads and the friction coefficient entering the disc and the pads, the number of revolutions and the material of the disc, the pad groove.

1. Introduction

With the development of new technologies in the automotive industry, vehicles have become more and more efficient. Braking systems should follow the same rhythm. The brake, as a major security organ, constantly arouses great interest to engineers. In addition competition in the automotive field is increasingly harsh, putting pressure on efficiency, reliability, comfort, cost and production time of all automotive systems. For an engineer, the goal is to find the best compromise between the requirements of security, technology and economic constraints. To achieve an optimal design, it should implement all available economic technologies to solve the technical problems, thus complementing experimental studies. In the aerospace and automotive industry, many parts are subjected to simultaneous thermal and mechanical loads, constant of fluctuating. The thermo-mechanical stresses cause deformations and may even damage the systems. For example, in friction braking systems, heat is generated in the disc and brake pads, causing high stresses, deformations and vibrations as cited in [1]. Finite element methods are now widely used to solve structural, fluid, and Multiphysics problems numerically. The methods are used extensively because engineers and

* Corresponding author. Tel.: +213 556803713, +213 664251761.
E-mail address: al.belhocine@yahoo.fr (A. Belhocine).
Peer review under responsibility of Faculty of Engineering, Alexandria University.

http://dx.doi.org/10.1016/j.aej.2016.03.025
1110-0168 © 2016 Faculty of Engineering, Alexandria University. Production and hosting by Elsevier B.V.

scientists can mathematically model and numerically solve very complex problem. Today, finite element methods probably are used for the analysis of every major engineering design and probably in every branch of scientific studies. The method is now used primarily through the application of commercial finite element programs. These programs are used on mainframes, workstations and PCs and are employed to solve very complex problem. Recently, several authors ([2–7]) have been successful in applying the finite element method in their research. In order to use the advantages of both finite volume and finite element methods, new numerical method has been introduced, namely Control Volume based Finite Element Method (CVFEM). Multi-physics problems in complex geometries can be simulated via this powerful method [8,9]. Sheikholeslami et al. [10] used the CVFEM as an application to solve their numerical problem. In work carried out by Kandelousi [11] and Sheikholeslami et al. [12], the same technique was used to solve the governing equations considering both Ferrohydrodynamic (FHD) and Magnetohydrodynamic (MHD) effects. Forced convection heat transfer of ferrofluid in the presence of non-uniform magnetic field was studied by Sheikholeslami et al. [13]. CVFEM was also applied in the numerical investigation. This method was applied in a different field of science [14–16].

Reibenschuh et al. [17] studied the thermo-mechanical analysis of the brake disc, with an elaborate model to determine the effects of thermal and centrifugal loads on the brake disc and its associated system. Subramanian and Oza [18] studied ventilated brake disc hub assembly subjected to braking torque and bolt pretension. The induced stresses due to the bolt pretension were found to be negligible compared to the braking torque. Shinde and Borkar [19] carried out another analysis of the brake disc system using ANSYS software to study the performance of two different pad materials – Ceramic and composite Fiber. This research provided useful design tools and improved braking performance of the disc brake system based on the strength and rigidity criteria.

Jungwirth et al. [20] carried out a thermo-mechanical coupled analysis of design brake discs and calipers. The simulation model was tested on a brake dynamometer to determine the deformations and its fatigue strength. The study was focused on the mechanical interactions between the calipers and brake disc, including the influence of heat power distribution on the brake disc. In work carried out by Söderberg and Andersson [21] a three-dimensional finite element model of the brake pad and the rotor was developed primarily for the calculations of the contact pressure distribution of the pad onto the rotor. Abdullah et al. [22] used the finite element method to study the contact pressure and stresses during the full engagement period of clutches using different contact algorithms. In this study, the sensitivity of the results of the contact pressure was exposed to show the importance of the contact stiffness between contact surfaces. Dhiyaneswaran and Amirthagadeswaran [23] guided a comparative study of disc brake with two different materials. The disc brake model was analyzed in dynamic load conditions and the contact stress pattern was modeled. The displacement and the elastic constraints of the existing material and alternative materials of the disc brake were also compared. Kumar and Vinodh [24] proposed a new automotive brake rotor design after they compared it with the ventilated disc rotor. The work used finite element analysis for both static structural and thermal transient analyses in order to evaluate and compare their performances. The analysis of the deformations of the rotor under extreme loads was carried out using a static structural analysis method.

Belhocine and Bouchetara [25] used the finite element software ANSYS 11.0 to study the thermal behavior of full and ventilated disc brake rotor. A transitory analysis of the structural thermo-mechanical couple was employed in order to visualize the stress fields of the constraints and their deformations in the disc. The contact pressure distribution on the brake pad was also established. Belhocine et al. [26] investigated the structural and contact behaviors of the brake disc and pads during the braking phase in the design case using FE approach, with and without thermal effects. The results of thermo-elastic coupling on Von Mises stress, contact pressures and total deformations of the disc and pads were presented. These are useful in the brake design process for the automobile industry.

In another study by the same authors as, Belhocine et al. [27] on structural and contact analyses of disc brake Assembly during a single stop braking event using the same commercial software, the stress concentrations, structural deformations and contact pressure of brake disc and pads were examined.

The principal objective of this paper was to develop a three-dimensional (3D) numerical model for the mechanical behavior of an automotive disc brake pad under dry contact slipping conditions during the braking process. The calculations were based on the static structural rested analysis in ANSYS 11.0. The main strategy of the analysis is to initially visualize the normal constraints and shear stresses and thus the sensitivity of some of the computation results, which will then be approached in detail. Thus, this study provides effective reference for design and engineering application of the brake disc and brake pad.

2. Study of mechanical contact – brake disc–pad

The disc and the pad were modeled by characterizing the mechanical properties of materials of each part. The type of analysis chosen was the static structural simulation. The total simulation time for braking was $t = 45$ s, and the following initial time steps were adopted:

- Increment of initial time = 0.25 s.
- Increment of minimal initial time = 0.125 s.
- Increment of maximal initial time = 0.5 s.

2.1. ANSYS simulation of the problem

The finite element code ANSYS 11 (3D) was used to simulate the behavior of the contact friction mechanism of the two bodies (pad wafer and disc) during a braking stop. This code has the frictional contact management algorithms based on the Lagrange multipliers method, or the penalization method. The Young's modulus of the disc was about 138 times higher than that of the pads. The simulations presented in this study, are considered the frictional pad to be deformable pad on a rigid disc (see Fig. 1).

The application of the contact pressure on the brake pad was input as frictional contact data, and the disc rotational speed was kept constant during the entire simulation. The