

Thermo-Mechanical Contact Analysis of Car Disc Brake Squeal

Muhammad Zahir Hassan, Peter C. Brooks and David C. Barton

The University of Leeds

Copyright © 2008 SAE International

ABSTRACT

Disc brake squeal can be classified as a form of friction-induced vibration. The elimination of brake squeal noise is very important as the problem causes discomfort of the vehicle occupant as well as pedestrians. This paper presents a new methodology for predicting disc brake squeal using finite element analysis considering both thermal effects and the structural compliance of brake components. An integrated dynamic study of non-linear contact pressure analysis and a fully coupled transient thermal analysis under variation of contact algorithm are performed before executing the instability study of a typical passenger car brake system using the complex eigenvalue analysis method. Based on the results of this exercise, a parametric study on the materials of brake components is carried out to define suitable design guidelines to reduce or to eliminate squeal problems.

INTRODUCTION

Automotive brake squeal continues to challenge engineers despite the efforts to reduce its occurrence during the past years. The problem is complex and influenced by many factors. During braking, the kinetic energy of the rotating brake disc is converted to thermal energy through friction at the pad and disc interface. Frictional heat is generated on the rubbing surface due to the interaction between the pad and disc. This allows the rotor disc to absorb 90% of generated energy by means of conduction from the friction interface between disc and pad [1]. Prediction of the surface temperature of the disc brake as well as the pad is complicated [2]. This difficulty is due to the fact that the thermal prediction is presented by the complex interaction between thermal and mechanical structure behavior of the brake components itself [3]. Finite element (FE) method is the most popular tool that is used widely to simulate the influence of temperature in brake system design and performance.

Intensive research has been undertaken to tackle the problem through experiment and simulation. Most researchers have omitted to study the brake squeal phenomenon that includes the structural thermal deformation and material properties with temperature dependence, due to the complexity of the brake system

and the additional computing load. The finite element method has become a favorable analysis tool to study squeal as experimental methods are more costly. The accurate numerical modeling of brake squeal performance must consider the influence of temperature under different operating conditions as well as pressure, velocity and coefficient of friction. Temperature changes due to frictional heat generation cause axial and radial deformation of the rotor along with pad expansion. The change in shape affects the contact between the rotor and pad surface area and thus will influence the load distribution at the friction interface. Therefore, brake squeal numerical modeling should be analyzed as a coupled thermo-mechanical problem so that full account is taken of the effect of thermal deformation when the pad and disc are in contact.

The present work utilizes the complex eigenvalue analysis technique to investigate squeal propensity of a brake system due to thermal loading condition. An integrated study of non-linear contact analysis and coupled thermal analysis is carried out that considers both the thermal effect and the structural compliance of the brake components. The contact pattern is examined between friction materials and rotor with respect to the coefficient of friction and load. The main reason to conduct a parametric study of brake components is to simulate the effect of material property changes on the squeal propensity. ABAQUS, which is a commercial finite element software, is used to meet the research objective.

METHODOLOGY

An overview of the modeling procedure to conduct the fully coupled thermo-mechanical contact analysis of disc brake squeal is given in Figure 1. The development of the validated FE model is aided by conducting free-free modal analysis of the structure and verified by the experimental modal analysis based on the actual brake components. This is followed by extracting the nodal temperatures of the brake components based on the applied pressure and coefficient of friction as the disc is subjected to rotational motion. Once completed, an additional contact analysis is carried out at specific time instants by the user. The heat flux input to the model is recalculated at these specific user load conditions. The temperatures obtained from the fully coupled thermal

analysis are used to update the thermal model to derive the new contact pressure distribution. Thereafter, the transient fully thermal dependent contact pressure distribution result is used to examine the stability of the disc brake system.

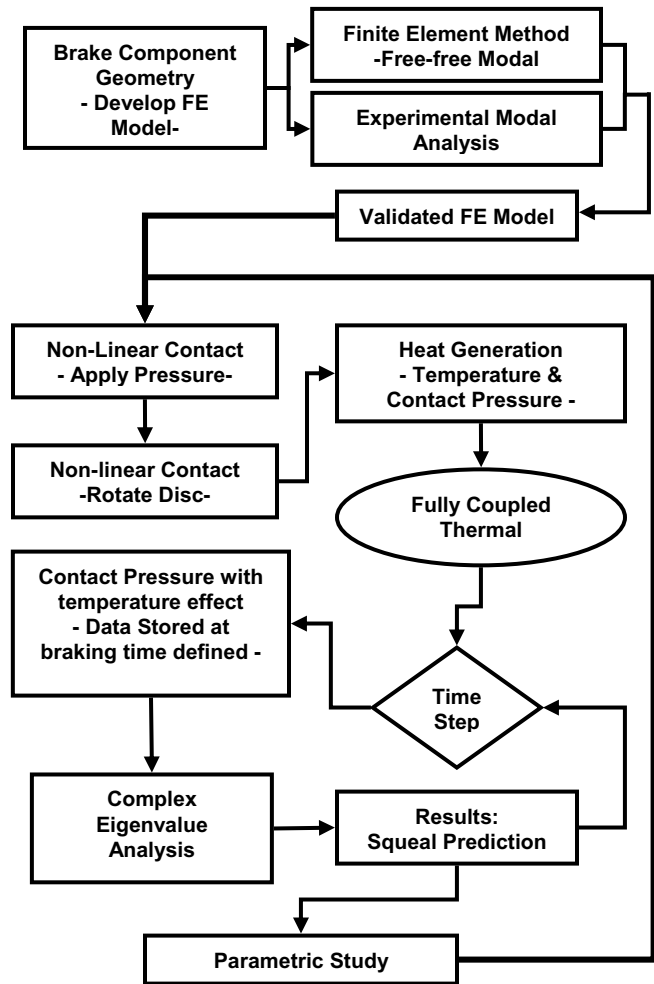


Figure 1: Methodology of Thermo-Mechanical contact analysis of brake squeal numerical modeling approach

FINITE ELEMENT MODEL DESCRIPTION

The development of the finite element model to investigate the brake stability is composed of the following procedure:

- A representative 3-dimensional simplified finite element model is created based on the geometry of Rover 75 brake components given by Table 1, which consists of 2 pads, rotor and pistons.
- Mesh refinement and material properties tuning based on experimental free-free modal analysis to validate and correct the finite element model.
- Create the contact scheme between the pad and disc contact surface.
- Constrain the brake system so that the disc can rotate and the pads move only in the axial direction.

- Execute the fully coupled thermo-mechanical contact analysis followed by the instability study.

The full disc brake assembly model of Rover 75 brake is shown in Figure 2. The model consists of 15848 elements. Three dimensional 8-noded solid brick elements with coupled thermal and stress capability have been used. This type of element permit large scale sliding between the contact surface (pad) and sliding target surface (disc). In order to simulate the axial movement of the pad, no constraint is applied in the direction normal to the disc surface. To simulate the hydraulic pressure, the two opposing pistons have the same constraints as the pad, to allow them to move freely in the axial direction. The centre of the disc is constrained in the normal direction only so that the disc is able to rotate.

Table 1: Geometrical and material properties of disc brake system

	Disc Brake	Pad	Backplate	Piston
Diameter (mm)	284	-	-	22
Width (mm)	-	120	128	-
Thickness (mm)	22	11	6	20
Density (kg/m ³)	7050	2620	7887	7887
Poisson's ratio	0.27	0.29	0.3	0.3
Young's Modulus (GPa)	116	1.25	210	210
Conductivity (W/m.°C)	53.3	2	32	32
Specific Heat (J/kg/°C)	103	1100	595	595
Thermal Expansion (K ⁻¹)	1.04 E-6	1.61 E-5	1.17 E-5	1.17 E-5

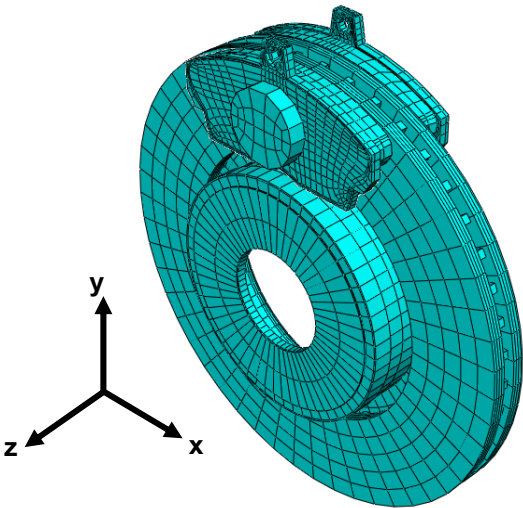


Figure 2: Rover 75 Simplified FE Model

There are three steps taken to define contact simulation for the present methodology:

- i. Definition of the surfaces of the bodies to be in contact.
- ii. Definition of the surface that will react to another surface area.
- iii. Definition of the contact thermal and mechanical properties when two surfaces are in contact.

To simulate the contact scheme between the pad and disc surface, surface-to-surface contact surfaces have been defined using the finite sliding contact algorithm. For the finite sliding, the slave nodes come into contact on the entire master surface and transfer loads are updated throughout the analysis. The finite sliding algorithm takes into consideration geometric nonlinearity. The master-slave approach is adopted, where the disc surface is chosen as the master surface since it has a coarser mesh and is stiffer than the pad. The contact surface frictional behavior is simulated by both the penalty method and Langrange multiplier. Although the penalty method is the default formulation constraint, the Langrange method must also be performed for the purpose of comparison. Langrange multiplier formulation is computationally expensive compared to the penalty method as it permits motion on the surface interface when the surface is sticking. Surface traction stresses are required in the complex eigenvalue extraction for the stiffness matrix to be asymmetric.

MODAL ANALYSIS

The initial stage of the finite element modeling carried out in this research involves running the free-free modal analysis. Each natural frequency is associated with a particular mode shape that the component model adopts when vibrating at that frequency. Experimental Modal Analysis (EMA) is conducted to verify the finite element model of the brake rotor. Results from the EMA are compared with natural frequencies and mode shapes obtained from the finite element free-free modal analysis to validate the FE brake model. Table 2 shows very good correlation with finite element modeling output frequencies which are reconciled with the experimental results within a satisfactory error range.

Table 2: Disc brake natural frequencies obtained from experimental modal analysis and finite element modeling

Mode	FEA (Hz)	EMA (Hz)	ERROR (%)
1 ND	920	918	-0.21
2 ND	1720	1703	-0.98
3 ND	2208	2187	-0.95
4 ND	3537	3628	+2.57

THERMO-MECHANICAL CONTACT ANALYSIS

The heat is generated between the pad and disc interface as braking is applied repeatedly over certain period. The surfaces of the bodies are deformable and non-uniform temperatures are generated. The heat Q produced when the pad and disc rubbing surface are in contact is proportional to the sliding velocity v and frictional traction force t_f as below:

$$Q = \int_{A_r} t_f v dA$$

where A_r is actual area of contact and t_f is frictional force per unit area. Under frictional sliding, the Coulomb's friction theory is employed and the traction force per unit area is presented as:

$$t_f = \mu P$$

Here μ is coefficient of friction and P is the normal contact pressure. Therefore, the area of actual contact, frictional traction force and sliding velocity are entirely depend on the thermal mechanical properties of the disc-pad interface.

ANALYSIS PROCEDURE

The SAE J2521 standard criteria for drag brake application have been used to conduct the simulation [5]. The equivalent vehicle test speed is set to either 3 km/h (1.92 rad/s) or 10 km/h (6.42 rad/s). An initial temperature of 20 °C is assumed for the brake model. The friction coefficient varies is set to 0.3, 0.4 and 0.5 under uniform pressure of $P = 1.0$ MPa to 3.0 MPa. The transient Lagrangian approach is used in this analysis where the mesh which discretises the disc model rotates relative to the disc brake assembly. Therefore a fine mesh is require along the surface of the disc.

A very stiff beam is used to connect the reference point created at the centre of the brake with the brake rotor. The rotational motion is applied at the reference point. The disc is constrained in all directions except in z-axis of direction which allows the disc to rotate by the increment of 1 to 360 degree angle of rotation. The friction material and piston are constrained in x and y-direction whilst allowed to move in the normal direction towards the rotor rubbing surface.

The simulation begins by applying a uniform pressure on the piston surface to press the pad against the disc surface. The brake line pressure is assumed to be uniformly distributed on the piston surface. The disc is rotated by an angle prescribed by the user based on the vehicle speed and braking time period. In this particular analysis, a time step to complete 1 revolution cycle is by increments of 1 degree using an applied boundary condition at the centre of the disc. Therefore one revolution consists of 360 time steps. The coefficient of friction is applied at the disc-pad interface and remains constant throughout the time step. The temperature field and the resulting contact profile at each time increment are stored to be used in the complex eigenvalue analysis. To reduce the physical size of the output data