Analysis of frictional heating and thermal expansion in a disc brake using COMSOL

Masrat Bashir¹, Adnan Qayoum¹ and Shahid Saleem¹

¹ Mechanical Engineering Department, National Institute of Technology, Srinagar 190006, J & K, India

E-mail: masrat_13phd16@nitsri.net, adnan@nitsri.ac.in, shahid@nitsri.net

Abstract. This work illustrates the modeling of frictional heating in a disc brake of a car using COMSOL Multiphysics software. Finite element analysis technique is used to predict the frictional heat, temperature distribution and variation in disc thickness of the disc brake. The parameters for the disc and pad were selected from the existing literature. A three dimensional model of disc brake has been created and simulated. The model simulates the dynamics and frictional heating between pad and disc. The frictional heat is computed in the multibody dynamics module of the COMSOL while as temperature distribution is computed by the heat transfer module. Also stresses and disc thickness variation is computed utilizing solid mechanics module. The results obtained from the simulation provide a clear justification of the use of the COMSOL Multiphysics for the evaluation of frictional heat and thermal expansion.

1. Introduction

The braking system is a standout amongst the most basic framework in a vehicle. The basic principle of this system is to slow down or bring a vehicle to a stop, by converting the kinetic energy of a vehicle into the frictional heat at the contact surfaces between the brake pad and disc. The rate of heat generation in a disc brake has an important role in the execution of a braking system. During repetitive braking, large amount of heat is generated which results in the loss of brake effectiveness due to drop of friction coefficient. This condition is termed as “brake fade”. Heat generation may also cause excessive component wear, thermal crack, squeal, brake fluid vaporization and in extreme cases it may cause complete failure of the brake. In order to overcome these problems, caused due to frictional heat, the heat is to be effectively dissipated to the surroundings for the optimal performance of the braking system [1]. Any enhancement to the cooling qualities of a braking system reduces the problems due to heat generation and provides safer vehicle transport. Therefore, it is important to assess the frictional heating and thermo-mechanical stresses in the early design stage ([2],[3]).

Finite element technique has been used to portray the moving disc with variation in relative sliding speed. For the determination of temperature distribution between pad and disc, an analytical model has been adapted [4]. The finite element method is currently used for heat transfer problems to obtain the numerical solutions. The foremost common choice preferred, when using finite elements is a basic Galerkin formulation [5]. From finite element analysis, it is indicated that inconsistent contact between pad and disc could influence the deformation of material [6]. The analysis of the heat generation and dissipation between the rotor disc and pad during adverse braking has been done using COMSOL Multiphysics, based on finite element principle ([7],[8]). For describing the thermal behavior of disc
brakes numerous mathematical models have been addressed ([9],[10],[11]). The thermal and structural analysis of the brakes is carried out to enhance the performance of the disc using finite element method, by determining von-mises stress and deformation as referred to in ([12],[13],[14]).

In this study, frictional heating and thermal analysis is carried out on a simple finite element model of brake disc and pad to establish the temperature distribution, von-mises stress and disc thickness variation by utilizing COMSOL Multiphysics.

2. Modeling of disc brake

Thermal and mechanical model of the disc brake is created utilizing COMSOL Multiphysics. A three dimensional model of disc brake is generated. The radii of disc and pad is chosen as 0.14m and 0.02m respectively [15]. The thickness of disc and pad is fixed as 0.013m and 0.05m respectively. The input parameters used for the simulation of disc brake are shown in Table 1. The properties of brake disc and pad material are shown in Table 2 [16].

### Table 1. Input parameters of the model

| Description                  | Symbol | Expression       | value          |
|------------------------------|--------|------------------|----------------|
| Initial vehicle speed        | v₀     | 60[km/h]         | 16.667m/s      |
| Vehicle mass                 | m_car  | 1800[kg]         | 1800kg         |
| Wheel radius                 | r_wheel| 0.25[m]          | 0.25m          |
| Friction coefficient         | Mu     | 0.7              | 0.7            |
| Braking force                | Fb     | 18000[N]         | 18000N         |
| Wheel angular speed          | omega  | v₀/r_wheel       | 66.6671/s      |
| Effective vehicle inertia on | l_car  | m_car*r_wheel^2/4| 28.125kg.m²    |
| each wheel                   |        |                  |                |

### Table 2. Material properties of brake disc and pad

| Property                        | Variable | Disc | pad |
|---------------------------------|----------|------|-----|
| Heat capacity at constant pressure[J/(kg.K)] | Cₚ       | 475  | 935 |
| Thermal conductivity[W/(m.K)]    | K        | 44.5 | 8.7 |
| Coefficient of thermal expansion[1/K] | alpha   | 12.3e-6 | -   |
| Density[kg/m³]                   | rho      | 7850 | 2000|
| Young’s modulus[Pa]              | E        | 200e9| -   |
| Poisson’s ratio                  | nu       | 0.30 | -   |
| Compressive yield strength[MPa]  | Fᵧ       | Min 820| -   |
| Surface emissivity              | ε        | 0.28 | 0.8 |

The geometry is meshed with free triangular element to convert the continuous structure into finite number of elements. The developed model contains approximately 15620 elements and 20466 degrees of freedom. The maximum and minimum element size of the mesh is 0.0098m and 4.2e-4m. The transient analysis has been carried.
3. Results and discussion

The finite element model has been determined utilizing COMSOL Multiphysics. The multibody dynamics is used to model the dynamics and frictional heating between pad and disc. Planer joint with friction, available in the multibody dynamic interface is used, instead of contact with friction. Planer joint has three degrees of freedom, two translational and one rotational, where the disc acts as a source and the pad acts as a destination. The multibody dynamics is coupled with heat transfer to model the temperature distribution in the disc brake. Also stresses and variation in disc thickness is calculated using solid mechanics interface with the help of temperature distribution computed in the heat transfer analysis. The procedure of coupling techniques used in the present study involving multibody dynamics, heat transfer and solid mechanics is similar to that used in the study done by Hwang and Wu [17]. The disc brake undergoes a combination of angular and rotational motion shown in figure 1.

![Figure 1](image1.png)

**Figure 1.** Time variation of the disc rotation and angular speed.

![Figure 2](image2.png)

**Figure 2.** Time variation of the frictional power generated between the brake disc and pad.

The time variation of frictional power generated between the pad and disc is shown in figure 2. It can be noticed that with increment in time the frictional power of the brake decreases. For single brake application, the mutual sliding of rotational disc and immovable pad results in heat generation due to
friction. The data obtained from the numerical simulation of frictional heating has been implemented in the calculation of temperature distribution of brake disc and pad. Figure 3 shows that at the braking time of 1.2 seconds the pad and the disc surface generates a maximum temperature of $432^\circ$ C. It is observed that along the radius of the pad, the temperature distribution is uniform and the hotspot is visible at the contact between disc and pad. This is due to uniform contact between the pad and disc. At the contact surface, the surfaces are finely meshed and node to node contact is established for producing precise frictional heat results. Almost similar results have been obtained by Belhocine et al. [6] using ANSYS. The strong temperature rise at the contact surface of the present study obtained on application of brake for short duration of time is in accordance with the results as reported by Belhocine et al. [6]. In addition the results of the present study are in with the results reported by Adamowicz and Grzes [18] and are shown in Figure 4.

![Figure 3. Temperature distribution in the brake disc and pad at 1.2 seconds.](image)

![Figure 4. Validation of simulation results with the A. Adamowicz and P.Grzes [18].](image)

Figure 5 shows von-mises stress is distributed symmetrically on the disc and along the leading and trailing side of pad. The stress distribution is unaltered over braking time, with the exception of the
stress value. At a braking time of 1.2 s the stress builds up gradually and reaches a maximum value of 369 MPa. The maximum stress is predicted at the contact surface along the outer radius of disc whereas the minimum stress is along the edges of the disc.

![Figure 5. Time variation of von-mises stress distribution in the brake disc.](image)

The utilization of the coupled analysis is to pick up better understanding of disc deformation under the combined influence of braking forces and thermal expansion. Figure 6 shows the disc thickness variation along the selected position on the disc brake. A clear distinction in the variation of disc thickness can be found at different times of braking in the circumferential direction. At time $t = 1.2s$ the disc gets deformed up to 8 microns and becomes nearly constant. Therefore it is clear from the results that temperature has a significant influence on the thermo-mechanical response of disc brake. Figure 7 shows the disc thickness variation along the line drawn on the disc. It is observed that large deformation occurs along the outer radius of the disc which is the contact area with pad. Quite similar results during thermo-mechanical analysis have been obtained using ANSYS that show the maximum deformation occurring along the outer radius of disc and indicates that the temperature plays an important role in the structural behavior of the disc brakes [3].

![Figure 6. Time variation of disc thickness variation at selected position (a) of the disc brake (b).](image)
4. Conclusion
In this study, the thermal analysis of the disc brake during braking process has been presented. The analysis has been conducted using COMSOL Multiphysics software. For the single braking application, the simulation results for frictional heat generation shows hot spots being developed radially on the outer edge of the pad. This leads to hot banding around the disc. The stress distribution is uniform and the maximum value obtained is less than the permissible yield strength value. Disc thickness variation increases in a notable way when the braking time is increased. The analysed values are less than their permissible values found in the literature, which justifies the design of disc brake.

References
[1] A. J. Day, M. Limited, and W. Yorkshire, “The dissipation of frictional energy from the interface of an annular disc brake,” Proc. Inst. Mech. Eng., vol. 198, no. 1, pp. 201–209, 2016.
[2] P. Chavan and A. Apte, “Axisymmetric analysis of bolted disc brake assembly to evaluate thermal stresses,” Abaqus India Reg. Users Meet, pp. 1–7, 2008.
[3] A. Belhocine, “Numerical Simulation of Thermoelastic Contact Problem of Braking Systems,” J. Fail. Anal. Prev., vol. 15, no. 6, pp. 932–940, 2015.
[4] C. H. Gao and X. Z. Lin, “Transient temperature field analysis of a brake in a non-axisymmetric three-dimensional model,” J. Mater. Process. Technol., vol. 129, pp. 513–517, 2002.
[5] D. M. Mijuca, A. M. Žiberna, and B. I. Medjo, “A New Multifield Finite Element Method in Steady State Heat Analysis,” J. Therm. Sci., vol. 9, pp. 111–130, 2005.
[6] A. Belhocine, A. Rahim, A. Bakar, and M. Bouchetara, “Thermal and structural analysis of disc brake assembly during single stop braking event,” Aust. J. Mech. Eng., vol. 4846, pp. 1–13, 2016.
[7] H. Singh and H. Shergill, “Thermal Analysis of Disc Brake Using Comsol,” Int. J. Emerg. Technol., vol. 3, no. 1, pp. 84–88, 2012.
[8] M. Naji and S. Masoud, “Transient thermal behavior of a cylindrical brake system,” Heat Mass Transf., vol. 36, pp. 45–49, 2000.
[9] P. Dufrenoy, “Two- / three-dimensional hybrid model of the thermomechanical
behaviour of disc brakes,” *J Rail Rapid Transit part F*, vol. 218, pp. 17–30, 2004.

[10] F. Talati and A. S. Jalalifar, “Analysis of heat conduction in a disk brake system,” *Heat Mass Transf.*, vol. 45, pp. 1047–1059, 2009.

[11] M. Eltoukhy, S. Asfour, M. Almakky, and C. Huang, “Thermoelastic Instability in Disk Brakes: Simulation of the Heat Generation Problem,” *Proc. COMSOL Users Conf. Bost.*, no. January, 2006.

[12] T. Valvano and K. Lee, “An Analytical Method to Predict Thermal Distortion of a Brake Rotor,” *SAE Tech. Pap.*, vol. 2000-01–04, no. 724, 2018.

[13] D. S. P. M. Manjunath T V, “Structural and Thermal Analysis of Rotor Disc of Disc Brake,” *Int. J. Innov. Res. Sci. Eng. Technol.*, vol. 2, no. 12, pp. 7741–7749, 2013.

[14] A. Belhocine and M. Bouchetara, “Thermomechanical Coupling Analysis of a Disc Brake Rotor,” *J. Fail. Anal. Prev.*, vol. 12, pp. 697–706, 2012.

[15] COMSOL Multiphysics Users Guide, “Heat Transfer Module,” 2017.

[16] C. Multiphysics, C. Software, and L. Agreement, “Heat Generation in a Disc Brake,” 2017.

[17] P. Hwang and X. Wu, “Investigation of temperature and thermal stress in ventilated disc brake based on 3D thermo-mechanical coupling model †,” *J. Mech. Sci. Technol.*, vol. 24, pp. 81–84, 2010.

[18] A. Adamowicz and P. Grzes, “Analysis of disc brake temperature distribution during single braking under non-axisymmetric load,” *Appl. Therm. Eng.*, vol. 31, no. 6–7, pp. 1003–1012, 2011.