Paper Reference ID: RTC 011

SURFACE TEMPERATURE DISTRIBUTION IN A COMPOSITE BRAKE ROTOR

A.A. Adebisi[1], M. A. Maleque[1] and Q. H Shah[2]

[1]Department of Manufacturing and Materials Engineering,
[2]Department of Mechanical Engineering
International Islamic University Malaysia
53100 IIUM Gombak, Kuala Lumpur, Malaysia
e-mail: debisi1@yahoo.com, maleque@iium.edu.my, hqasim@iium.edu.my

ABSTRACT

The prediction of surface temperature for brake rotor is regarded as an important step in studying the brake system performance. The frictional heat generated on the rotor surface can influence excessive temperature rise which in turn leads to undesirable effects such as thermal elastic instability (TEI), premature wear, brake fluid vaporization (BFV) and thermally excited vibrations (TEV). The purpose of this study is to investigate the temperature distribution profile for brake caliper pressure application of 0.5, 1.0 MPa with a speed of 60km/h braking condition on the disc rotor surface. The brake rotor assembly is built by using a 3 dimensional finite element model of a real car brake rotor. To verify the simulation results, an experimental investigation is carried out. It is believed from the study that composite brake rotor influences the temperature distribution and heat dissipation rate which could prevent excessive temperature rise and subsequently prolong the service life of the rotor. The finite element method is cost effective and also assists the automotive industry in producing optimised and effective brake rotor for thermal distribution analysis.

Keyword: brake rotor, temperature distribution, finite element model, frictional heat

1.0 INTRODUCTION

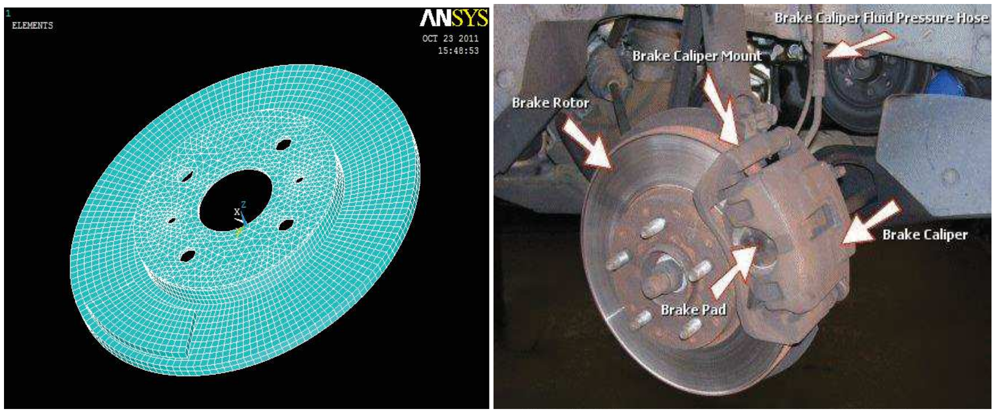
Brake system is an essential component in the automotive industry due to its safety concern to reduce or stop a vehicle on high speed. The braking performance is significantly affected by the temperature rise in the process of halting the vehicle. Each moment (time step) during the continuous braking process gives a different value of temperature distribution as a result of the frictional heat generated on the rotor surface which can cause high temperature rise (Qi and Day, 2007; Hwang and Wu, 2010). When the temperature rise exceeds the critical value for a given material, it leads to undesirable effects in the operation of the rotor such as thermal elastic instability (TEI), premature wear, brake fluid vaporization (BFV) and thermally excited vibrations (TEV) (Gao and Lin, 2002; Kao, et al., 2000). The material properties of the brake rotor play an important role by influencing the thermal conductivity and heat dissipation during braking. Recent studies have shown that advanced composite such as aluminium matrix reinforced with silicon carbide particle is a potential material for brake rotor development due to its thermo-physical properties (Qi, et al., 2001). In a study by Gao and Lin (2002), they observed that considerable evidence has shown that the contact temperature distribution is an integral factor influencing the combined effect of load, speed, friction coefficient and the thermo-physical and durability properties of the materials. In another study Lee and Yeo (2000) stated that the uneven distribution of temperature at the surfaces of the rotor could bring about thermal distortion which causes thermal judder and excited vibration.

Finite element (FE) method for brake rotor analysis has become a preferred method in studying the thermal distribution performance because of its flexibility and diversity in providing solutions to problems involving advanced material properties. Chandrupatla and Belegundu (2002) stated that temperature distribution analysis is mostly performed using FE method due to its powerful tool for numerical solutions for a wide range of engineering problems. Day (1988) conducted a study using FE to predict temperature, wear, pressure distribution and thermal distortion of a brake drum which is generated during high pressure brake application from two different road speed and friction materials. Valvano and Lee (2000) proposed a thermal analysis on disc brake based on a combination of computer based thermal model and FE based techniques to provide reliable method to calculate the temperature rise and distortion under a given brake schedule.

In this paper, the FE model of a real brake rotor assembly is developed and simulated using the commercially available FE software packages, ANSYS and LS-prepost respectively. The model is simulated using a 3D thermo-mechanical coupling model in order to observe the surface temperature distributions profile for different applied braking condition.

2.0 FINITE ELEMENT MODEL

The finite element model of a real brake rotor consists of the composite rotor disc and two friction materials as shown in Figure 1. The FE model is developed based on the actual Proton Wira 1.3 solid brake rotor assembly.

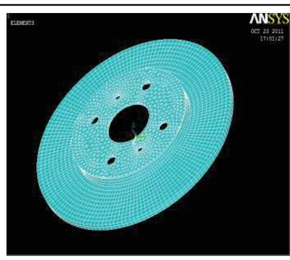
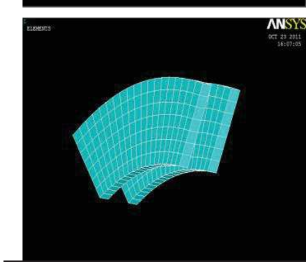


(a) Finite element model (b) Proton Wira brake assembly

Figure 1: Brake rotor assembly

The brake rotor model assembly utilizes up to 9053 solid elements with the rotor element comprising of 8787 and the pads 133 elements. The SOLID 164 element type is used for the three-dimensional modeling of the brake rotor solid structures. The element is defined by eight nodes having the following degrees of freedom at each node: translations, velocities, and accelerations in the nodal x, y, and z directions. It gives a reduced one point integration which saves computer time and robustness in cases of large deformations. The description of the brake rotor model is given in Table 1.

Table 1: Description of the brake rotor assembly components

Components		Type of Elements	Number of Elements	Number of nodes
	Rotor	Solid 164	8787	7638
	Pads	Solid 164	133	640

The FE model structure is imported into the LS-prepost software in preparation for the implicit dynamic solution. The contact type is defined as automatic surface to surface thermal friction for the model which defined the mechanical static and dynamic friction coefficient as a function of temperature. It also defined the thermal contact conductance as a function of temperature, pressure parameters and contact stiffness. This is to ensure that the temperature distributions on the rotor/pad interface is more significant compared to other contact interfaces. The rotor is chosen as the master surface due to its stiffness, while the friction materials were chosen as the slave surface.

2.1 Boundary Conditions

For structural and thermal analysis of the brake rotor model, boundary condition is specified;

2.1.1 Structural boundary condition

It is specified by imposing nodal motion on the set of nodes and the motion is prescribed with respect to the local coordinate system of the brake rotor. The degree of freedom (DOF) for the boundary prescribed motion specifies that x/y DOF for node rotating about the z axis is at a location specified in the x-y plane. The SPC set specifies the constraints at the nodal single points.

2.1.2 Thermal boundary condition

Boundary temperature condition for the set of nodes is specified for coupled thermal/ structural analysis of the brake rotor by the load curve ID for temperature versus time interval.

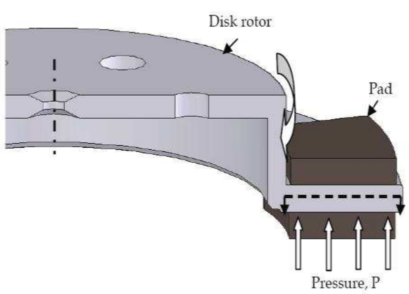


Figure 2: Thermal boundary condition

2.2 Load Application

Load is applied to the rotor model structure in preparation for explicit dynamic solution. The pressure is defined using load segment keyword which is applied to the faces of the model, on top of the appropriate solid elements (as rigid body). The faces are defined with segments and the load is defined with the load curve number. The load curve is specified with a well defined load direction before it is then applied on the brake rotor model as
shown in the Figure 3.

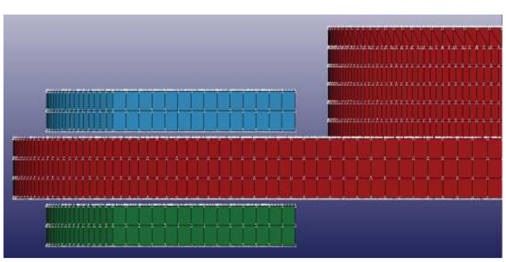


Figure 3: Load application on brake rotor and pad

3.0 THERMAL DISTRIBUTION ANALYSIS

The dissipated energy converted into heat is specified as all the mechanical energy is converted into thermal energy. Energy dissipated as heat between the surfaces and the distributions are equal between the two interacting surfaces. Heat is generated on the surfaces between the rotor and pad when the rotor rotates. This could be expressed as (Al-Bahkali and Barber, 2006).

$$q = \mu V p \quad \text{..............................} \ (1)$$

where μ is the friction coefficient, V is the sliding velocity of the rotor and p is the contact pressure at the interface, q is the amount of heat generated by friction. For other regions on the rotor and pad exposed to the environment, it is assumed that the heat exchange is transferred through convection process. Therefore, convection surface boundary condition is applied. This can be expressed as:

$$-k\frac{\partial T}{\partial x} = h\left[T_\infty - T\left(0,t\right)\right] \quad \text{.............} \ (2)$$

where h is convection heat transfer coefficient, T_∞ is atmosphere temperature and $T\left(0,t\right)$ is the current temperature of the node.

4.0 SIMULATION AND EXPERIMENTATION

In the present study, a proton wira with vehicle curb weight of 1250kg is utilised, the friction and drag coefficient of the contact pair is 0.35 and 0.30 respectively with an initial temperature of 35C. The rotor material for the study is 20 wt% MPS-SiC AMC. The dimension and material properties of the brake rotor and pads are listed in Table 2.

Table 2
Material property and dimension of brake rotor and pad

	Rotor	Pad
Inner radius (mm)	135	155
Outer radius (mm)	230	221
Thickness (mm)	15	10
Density (kg/m^3)	2.903	2.595
Specific heat (Nm/kgK)	845	1465
Thermal conductivity (Nm/s°Cm)	170	1.212
Young`s modulus (GPa)	113	22
Poisson`s ratio	0.24	0.25
Tensile strength (MPa)	178	-

The vehicle speed is 60km/h during the static running test carried out in the automotive laboratory for varied brake pressure application. Brake pressure of 0.5, 1.0 MPa is applied on the pad through the caliper piston to generate the pressure which is monitored with a pressure gauge from the caliper valve (nipple). The model is symmetrical about the work surface of the friction contact pair which is defined to carry out simulation for the temperature distribution profile. Based on the 3D thermo-mechanical coupling technique, the analysis generated for the braking process was presented for temperature versus time interval. To verify the simulation results, an experimental investigation was carried out for the AMC brake rotor temperature distribution and also compared with the conventional cast iron brake disc rotor.

5.0 RESULTS AND DISCUSSION

Several assumptions were taken into consideration when performing the thermal analysis. The applied brake pressure is assumed to be uniformly distributed on the brake pads during operation. The coefficient of friction is assumed to remain constant throughout the analysis. The material and thermal properties are homogeneous and invariant with the temperature. The wear affect is also neglected.

Brake pressure is applied directly on the pads through the caliper piston until it makes contact with the brake rotor where it becomes constant. The rotational speed of the rotor during contact with the pad develops frictional heat until the temperature gradually increases. After which the rotation of the rotor becomes constant and the thermal analysis continue until the end of the simulation. The mid distance region of contact between the rotor and pad is analysed for temperature distribution profile on the rotor surface.

Figure 4 shows a temperature profile for pressure application of 0.5MPa and figure 5 gives the corresponding mid radial temperature distribution plot, the temperature gradually increases to 78°C for a time period of 20 ms. Figure 6 shows the temperature profile for pressure of 1MPa with a temperature rise of 147.7°C for 20 ms and figure 7 gives the mid radial temperature distribution plot respectively. From the surface temperature profile plots, it shows that the temperature increases and decreases at certain region at the same time interval. The increase in temperature results from the rotor contact with the pad, and when the rotor slides away from the pad the temperature will slightly drop. The reason for this is as a result of the cooling effect through heat transfer process (conduction) which also depends on the material properties of the rotor.

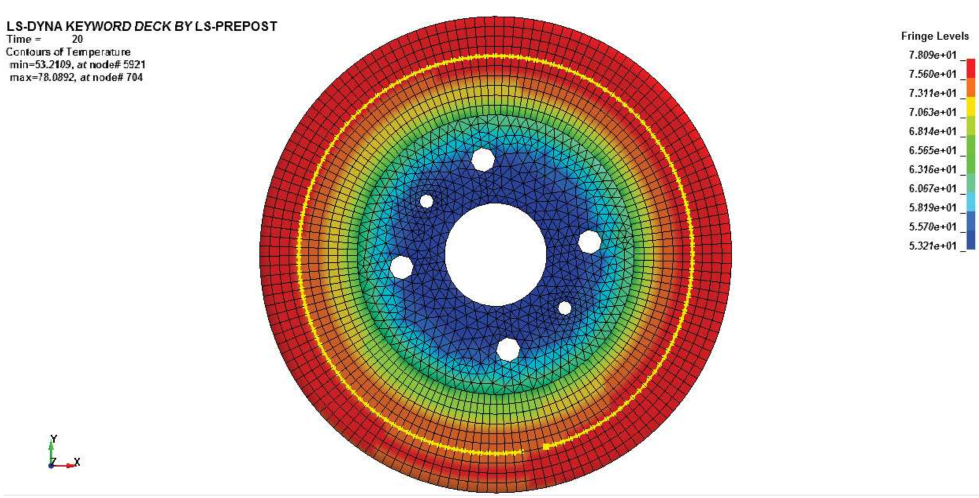


Figure 4: Surface temperature distribution profile for 0.5MPa

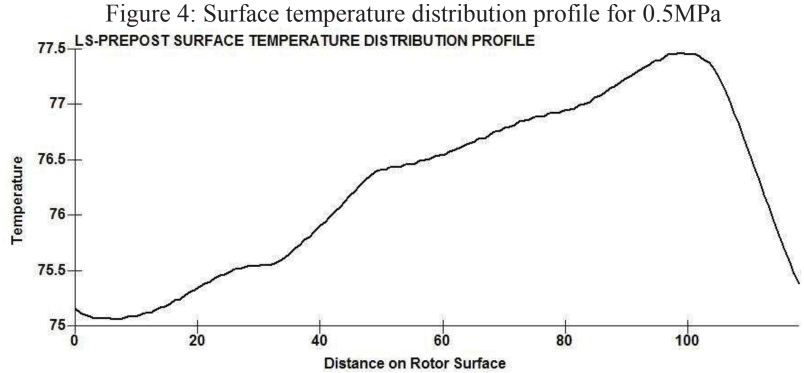


Figure 5: Mid radial temperature distribution plot for 0.5MPa

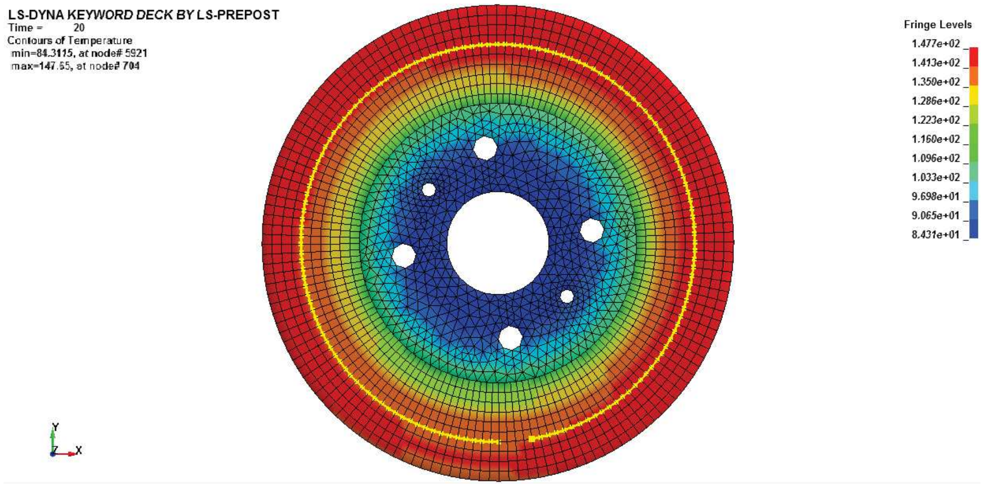


Figure 6: Surface temperature distribution profile for 1.0MPa

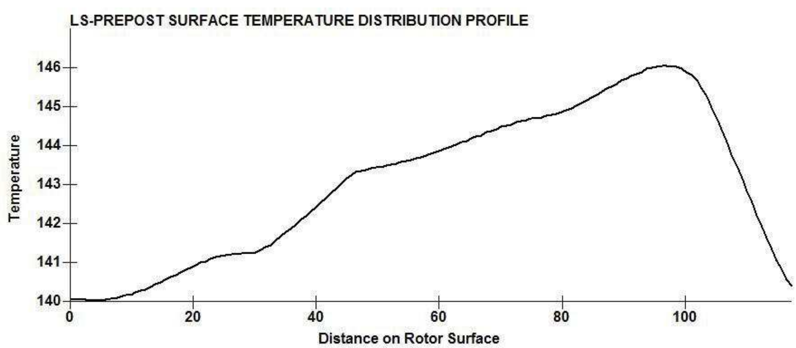
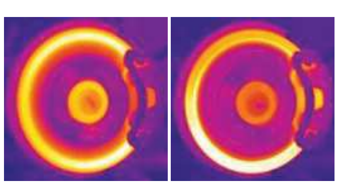


Figure 7: Mid radial temperature distribution plot for 1.0MPa

From the result, the time history curve for the rotor surface shows that temperature increases linearly before dropping which indicates that increase in brake pressure increases the surface temperature of the rotor. The study also found that higher temperature occurs at the center of the rotor surface and it spreads to the circumferential direction. The inner portion of the rotor remains the warmest section. The AMC brake rotor properties exhibited better distribution of temperature which reduced the localization of heat generation thereby influencing the hot spot and thermal elastic instability (Khai, et al., 2007). Experimental test was conducted for both AMC and cast iron brake rotors. Figure 8 shows the profile analysis for both rotors.

P = 0.5MPa P = 1MPa
T = 20 secs T = 20 secs

AMC Rotor



Cast Iron Rotor

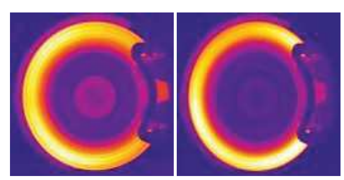


Figure 8: Thermograph analysis for average surface temperature profile for AMC and cast iron brake rotor.

At a pressure of 0.5MPa the average surface temperature for both rotors is slightly different, but as the pressure applied is increased to 1 MPa the temperature gap increases, this is due to the material properties application. The surface temperature profile of the rotor measured in the experimentation test for 0.5 and 1MPa is shown in Figure 8. This shows that the simulation results for the AMC brake rotor are in good agreement with the experimental values.

CONCLUSIONS

The present study investigated the surface temperature distribution analysis of the AMC brake rotor. The LS-prepost (LS-Dyna) finite element software package is utilized to predict the temperature distribution on the rotor surface. Long duration investigation is limited due to hardware limitation and time constraint, although results generated from the investigation is adequate to observe some relevant characteristics of temperature distribution profile. Moreover, the following conclusions can be made from the study;

1. Successful development of AMC brake rotor through experimental and FE model analysis.
2. The AMC brake rotor exhibited better distribution of temperature which reduces the localization of heat generation thereby influences thermoelastic instability TEI, premature wear and thermally excited vibrations TEV.
3. Properties of the AMC rotor shows improved cooling effect due to its high thermal conductivity when compared to conventional cast iron properties.
4. Both the experimental and simulated results for the AMC rotor are in good agreement.

ACKNOWLEDGEMENT

The authors acknowledge the support of the department of Manufacturing and Materials Engineering, International Islamic University Malaysia and also grateful to the Research Management Centre, International Islamic University Malaysia (IIUM), for financial support to conduct this research work under project EDW B 0906-332.

REFERENCES

Al-Bahkali, E. A. and Barber, J. R. (2006). Nonlinear Steady State Solution for a Thermoelastic Sliding System Using Finite Element Method. Journal of Thermal Stresses, 29 (2): 153-168

Chandrupatla, T.R. and Belegundu, A. D. (2002) Introduction to finite elements in engineering. 3rd edition Prentice Hall.

Day, A. (1988). An analysis of speed, temperature and performance characteristics of automotive drum brakes. Journal of tribology, 110: 298.

Gao, C. and Lin, X. (2002). Transient temperature field analysis of a brake in a non-axisymmetric three-dimensional model. Journal of materials processing technology, 129 (1-3): 513-517.

Hwang, P. and Wu, X. (2010) Investigation of temperature and thermal stress in ventilated disc brake based on 3D thermo-mechanical coupling model. Journal of mechanical science and technology, 24(1): 81-84.

Kao, T., Richmond, J. and Douarre, A. (2000) Brake disc hot spotting and thermal judder: an experimental and finite element study. International Journal of Vehicle Design, 23(3): 276-296.

Khai, L. C., and Abu Bakar, A. and Abdullah, M. S. (2007). Prediction of temperature distributions in a disc assembly using the finite element method. 5th Malaysian Friction Materials.

Qi, H.S. and Day, A. J. (2007). Investigation of disc/pad interface temperatures in friction braking. Wear, 262 (5-6): 505-513.

Qi, H., Fan, Y. and Ding, Z. (2001). Application of the particular reinforced aluminum composite to automobile brake rotors. Journal of the Hebei Academy of Sciences, **2**.

Sangkook, L. and Taein, Y. (2000). Temperature and coning analysis of brake rotor using an axisymmetric finite element technique. Science and Technology, 17-22.

Valvano, T., Lee, K. and Systems, D. A. (2000). An analytical method to predict thermal distortion of a brake rotor. SAE transactions, 109(6): 566-571.