

High-Fidelity Transient Thermal Modeling of a Brake Corner

Nimrod Kapas and Ajith Jayasundera
General Motors Corp.

ABSTRACT

There is an increasing interest in transient thermal simulations of automotive brake systems. This paper presents a high-fidelity CFD tool for modeling complete braking cycles including both the deceleration and acceleration phases. During braking, this model applies the frictional heat at the interface on the contacting rotor and pad surfaces. Based on the conductive heat fluxes within the surrounding parts, the solver divides the frictional heat into energy fluxes entering the solid volumes of the rotor and the pad. The convective heat transfer between the surfaces of solid parts and the cooling airflow is simulated through conjugate heat transfer, and the discrete ordinates model captures the radiative heat exchange between solid surfaces. It is found that modeling the rotor rotation using the sliding mesh approach provides more realistic results than those obtained with the Multiple Reference Frames method. Because of the significant computational requirements involved, the scope of this paper is limited to a single braking event of a brake subsystem in an environmental chamber. Corresponding dynamometer tests have also been performed to validate the simulation results. It is shown that the agreement between the measured and simulated data is acceptable. The thermal energy partitioning between rotor and pad predicted by the model confirms that almost all of the frictional heat is directed from the friction interface into the rotor. The uneven temperature gradients in the brake rotor are quantified and the sensitivity of the simulation results with respect to key modeling parameters is discussed. The presented simulation model can be implemented in several commercial CFD codes. While Fluent was used in this study, the comparison with other CFD software will be the subject of future work.

CITATION: Kapas, N. and Jayasundera, A., "High-Fidelity Transient Thermal Modeling of a Brake Corner," *SAE Int. J. Passeng. Cars - Mech. Syst.* 9(3):2016, doi:10.4271/2016-01-1929.

INTRODUCTION

Automotive brakes convert significant amounts of mechanical energy into frictional heat when decelerating a vehicle. Understanding the temperature levels occurring in brake systems during various driving conditions is critical for appropriate brake component sizing and selection of materials. Excessive temperatures may result in a loss of braking performance such as reduced lining life, increased wear and pedal travel, pad fading, brake fluid boiling, thermal stresses and rotor cracking, etc. In order to achieve the durability, performance and safety goals in a cost effective way, the thermal modeling of brake systems is conducted at the early stages of a vehicle program.

The thermal analysis of brake systems has primarily focused on brake rotor cooling in the past. A few decades ago, when the computational resources were not available for detailed CFD simulations, empirical Nusselt numbers and heat transfer coefficients played a major role [1, 2, 3]. As the computing power of modern computers increased, the numerical simulations became a common practice for engineering analysis and several CFD studies were presented that predicted the steady-state convective heat transfer coefficient of brake rotors [4, 5]. Although this approach solved the heat transfer equation in the fluid domain only, it was efficiently used to select an optimized vehicle configuration for enhanced cooling airflow around the brake assembly [6, 7]. Some studies combined the objectives of developing a vehicle design for both reduced aero drag and improved brake

cooling at the same time [8, 9]. Also, the convective heat transfer coefficients predicted by the CFD solvers could be utilized in lumped capacitance models of the rotor for simulating transient braking schedules [10, 11, 12]. An important assumption in these models is that the heat conduction within the rotor occurs much faster than the heat convection from the rotor into the surrounding air, so the temperature of the rotor can be considered uniform during the cooling phase [13]. For passenger cars at moderate vehicle speeds, the Biot number, which is the ratio of the convective and the conductive heat transfer rates, typically falls below the recommended value of 0.1 and the lumped capacitance model is reasonable. However, in case of larger size rotors in high performance or racetrack capable applications, temperature gradients in the rotor may reduce the predictability of the model.

To reduce the computational time of high-fidelity brake cooling simulations, the heat conduction equations could be solved in the solid domain only while the convective and radiative heat transfer are modeled as boundary conditions on the solid surfaces [14]. In other solution strategies, the governing equations are solved sequentially in the fluid and solid domains, so that the convection and radiation variables obtained from a pure CFD analysis are mapped into a subsequent FEM model for simulating transient thermal and structural processes [15]. Solving the conduction and convection equations in both the solid and fluid domains simultaneously is implemented in the conjugate heat transfer modeling approach that

can reveal more accurate and realistic details about the thermal processes in brake systems [16]. Most recently, in addition to conduction and convection, radiation equations also are solved simultaneously either by fully coupled flow and thermal codes [17] or within one single flow solver [18]; these models initialize the variables into a state that represents conditions after a brake apply event and the simulations only include the subsequent transient brake rotor cool-down process during constant speed driving.

The objective of this paper is to advance the state-of-the-art modeling techniques for brake system thermal analysis by developing an analytical method that includes the simulation of both brake apply and subsequent cool-down processes. In this approach, the model applies the frictional heat at the interface on the contacting rotor and pad surfaces during braking. Traditionally, the partitioning of the total frictional heat flowing from the pad-rotor contact interface into the solid volumes of the pad and the rotor was determined based on simplifying assumptions. Considering two semi-infinite contacting solids with equal surface temperatures at their interface, the distribution of the heat flux absorbed by the rotor and the pad can be estimated as [19]:

$$\frac{q_r''}{q_p''} = \sqrt{\frac{\rho_r C_r k_r}{\rho_p C_p k_p}}, \quad (1)$$

where the symbols used in the above equation are explained in the nomenclature. The thermal energy partitioning between the rotor and the pad is sometimes used as a tuning parameter in order to better correlate the simulated brake temperatures to physical test data. A more sophisticated approach iteratively solves the heat transfer equations within the domain surrounding the contact interface while matching the surface temperatures of the contacting bodies at all points of the contact area for a given total frictional heat applied at the friction interface. The resulting heat partition function is space dependent within the contact area and can be based on transform solutions that are inverted to obtain the surface temperature distribution [20] or using numerical algorithms with local heat sources [21]. These methods assume a perfect contact between the bodies with no temperature jump across the contact interface. However, in real situations the surface temperature of the rotor and pad can be significantly different due to the contact thermal resistance at the friction interface that is caused by the formation of surface layers and interfacial wear products [22]. Modern commercial CFD codes allow the users to easily set up the partitioning of frictional heat between contacting bodies with sliding interfaces [23]. Specifically, the coupled thin-wall boundary condition can account for contact thermal resistance and heat generation with automatic partitioning between contacting bodies considering the total thermal resistance of the solid parts on each side of the friction interface. Hereby, the simulations can provide uniquely new information such as the uneven temperature gradients in the brake rotor that are generated while the frictional heat is applied during braking. In addition, contributions of thermal conduction, convection and radiation within the net heat transfer rate into and out of the thermal mass of the brake rotor can also be quantified during both the deceleration and acceleration phases of a braking event. The sample simulation data presented in this paper could be further utilized in

other brake engineering studies, for example, in rotor thermal stress, pad lining life and brake fluid temperature analyses. In this paper, ANSYS Fluent is used for modeling the thermal behavior of automotive brakes, and comparison with other CFD codes will be the subject of a future work.

The rest of the paper is organized into the following sections, starting with a description of the modeling approach through validation of simulation results with test data:

1. Modeling Approach, Assumptions and Solution Techniques
2. Sensitivity Analysis with Key Solver Settings
3. Dynamometer Test and Data Analysis
4. Model Validation
5. Conclusions

1. MODELING APPROACH, ASSUMPTIONS AND SOLUTION TECHNIQUES

The following subsections describe the overall methodology and assumptions applied, along with a comparison between the results obtained by the sliding mesh and MRF modeling options for a test case.

1.1. Governing Equations

For turbulent flows in a stationary reference frame, the basic variables of fluid pressure and flow velocity can be obtained by solving the Reynolds-Averaged Navier-Stokes (RANS) equations in Cartesian tensor form as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0 \quad (2)$$

$$\begin{aligned} \frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = & -\frac{\partial p}{\partial x_i} + \dots \\ & + \frac{\partial}{\partial x_j} \left(\mu \left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right] \right) + \frac{\partial}{\partial x_j} (-\rho \overline{u'_i u'_j}) + \rho g_i + F_i \end{aligned} \quad (3)$$

Equations (2) and (3) have the same general form as the instantaneous Navier-Stokes equations, but the solution variables here are time-averaged values and the momentum equation has additional terms, the Reynolds stresses $-\rho \overline{u'_i u'_j}$, that represent turbulence effects. Assuming that the turbulent viscosity is isotropic, the system of equations can be closed by modeling the Reynolds stresses according to the Boussinesq hypothesis:

$$-\rho \overline{u'_i u'_j} = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \left(\rho k + \mu_t \frac{\partial u_k}{\partial x_k} \right) \delta_{ij} \quad (4)$$

The selected turbulence model in this study is the standard k - ϵ model that introduces two additional transport equations for k and ϵ , and the turbulent viscosity, μ_t , is calculated based on these two variables:

$$\begin{aligned} \frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) &= \dots \\ &= \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \end{aligned} \quad (5)$$

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) &= \dots \\ &= \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon \end{aligned} \quad (6)$$

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (7)$$

The turbulence model constants $C_{1\varepsilon}$, $C_{2\varepsilon}$, C_μ , σ_k , σ_ε have standard values defined in ANSYS Fluent that are established by experiments and are widely accepted for basic turbulent flows. Although they work well for a wide range of engineering applications, it should be noted that their values may need to be tuned for more accurate results in a particular simulation model.

In order to obtain the temperature field in the domain the turbulent energy equation, that is analogous to the turbulent momentum transfer, has to be solved also:

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_i}[u_i(\rho E + p)] = \frac{\partial}{\partial x_j} \left(k_e \frac{\partial T}{\partial x_j} + u_i(\tau_{ij})_e \right) + S_h \quad (8)$$

In Equation (8) the effective thermal conductivity is calculated as:

$$k_e = k + \frac{c_p \mu_t}{Pr_t}, \quad (9)$$

where a default value is used for the turbulent energy Prandtl number, Pr_t . Again, it should be noted that changing the value of Pr_t may be necessary to improve the accuracy of a simulation model in certain cases.

The k - ε model is derived under the assumption of a fully turbulent flow where molecular viscosity has negligible effects. Near the walls, however, the tangential velocity fluctuations are reduced by the no-slip condition on the wall surface and the viscous damping in the fluid, and the normal fluctuations are decreased by the kinematic blocking of the wall. Immediately next to the wall in the viscous sublayer, the flow is almost laminar and molecular viscosity has the dominant role in momentum and heat transfer. In the fully-turbulent outer boundary layer, turbulence has a major role because of the production of turbulence kinetic energy by the large mean velocity gradients. To capture the effects of walls in a brake thermal model, the widely used semi-empirical standard wall function approach is utilized in this paper, which calculates the dimensionless mean velocity, U^* , based on the dimensionless distance from the wall, y^* :

$$U^* = \begin{cases} y^* & (y^* < 11.225) \\ \frac{1}{\kappa} \ln(Ey^*) & (y^* > 11.225) \end{cases} \quad (10)$$

Similarly, according to Reynolds' analogy between momentum and energy transfer, the wall function for mean temperature comprises a linear law in the thermal conduction sublayer and a logarithmic law in the turbulent region. For smooth walls, the law-of-the-wall is given by:

$$T^* = \begin{cases} Pr y^* & (y^* < y_T^*) \\ Pr_w \frac{1}{\kappa} \ln(Ey^*) + \dots \\ + Pr_w 9.24 \left(\left(\frac{Pr}{Pr_w} \right)^{\frac{3}{4}} - 1 \right) \left(1 + 0.28 e^{-\frac{0.007 Pr}{Pr_w}} \right) & (y^* > y_T^*) \end{cases} \quad (11)$$

The turbulent Prandtl number at the wall, Pr_w , is another model constant for which a default value is used. The standard wall functions require a controlled refinement of structured computational mesh in wall normal direction and a sufficient resolution of the wall boundary layer for accurate numerical results.

Besides heat conduction and convection in the fluid zones, Equation (8) can also model the thermal radiation through the volumetric heat source term, S_h . Since the surface-to-surface radiation model would require the view factors to be updated at every time step in a moving mesh simulation, the computationally more efficient discrete ordinates (DO) model is selected in this study. The DO approach solves the transport equation of radiation intensity for a finite number of discrete solid angles that are associated with vector directions $\bar{s}$:

$$\begin{aligned} \nabla \cdot (I(\bar{r}, \bar{s}) \bar{s}) + (a + \sigma_s) I(\bar{r}, \bar{s}) &= \dots \\ &= an^2 \frac{\sigma T^4}{\pi} + \frac{\sigma_s}{4\pi} \int_0^{4\pi} I(\bar{r}, \bar{s}') \Phi(\bar{s} \cdot \bar{s}') d\Omega' \end{aligned} \quad (12)$$

In solid regions, the overall energy equation can be written as:

$$\frac{\partial}{\partial t}(\rho h) + \nabla \cdot (\bar{v} \rho h) = \nabla \cdot (k \nabla T) + S_h, \quad (13)$$

where the term $\nabla \cdot (\bar{v} \rho h)$ represents convective energy transfer caused by the solid zone motion, which is defined by $\bar{v}$.

For modeling the complex flow patterns through the vanes of the rotating brake disc, either the MRF or sliding mesh approach needs to be utilized. According to this, the brake thermal model is divided into volumetric zones that are either stationary or rotating. In case of the MRF approach, both the stationary and rotating zones have static mesh nodes, but the governing equations are solved in stationary and rotating reference frames, respectively. When the MRF model with absolute velocity formulation is used, the conservation of mass, momentum and energy equations for a purely rotating reference frame take the following form:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \bar{v}_r) = 0 \quad (14)$$

$$\frac{\partial}{\partial t} (\rho \bar{v}) + \nabla \cdot (\rho \bar{v}_r \bar{v}) + \rho (\bar{\omega} \times \bar{v}) = -\nabla p + \nabla \cdot \bar{\tau} + F \quad (15)$$

$$\frac{\partial}{\partial t} (\rho E) + \nabla \cdot (\rho \bar{v}_r H + p \bar{u}_r) = \nabla \cdot (k_e \nabla T + \bar{\tau} \cdot \bar{v}) + S_h \quad (16)$$

The absolute velocity formulation was chosen because the majority of the computational domain belongs to a stationary reference frame and so this option reduces the numerical diffusion in the solution. In addition, since in this approach the resulting velocities are always stored in the absolute frame, the velocity data can be transferred between the rotating and stationary reference frames directly.

In the sliding mesh approach, the mesh is updated at every discrete time step during a transient simulation by rotating the mesh nodes rigidly within the specified dynamic zones. The rotating and stationary cell zones are connected through non-conformal interfaces that are also updated based on the new locations of the rotating nodes. The motion of the brake disc model is setup in such a way that the bonding surfaces of the stationary and rotating zones are sliding along the interface boundary, creating a continuous contact without overlaps or separations between the two cell zones. For the sliding mesh model, the conservation equation for a scalar, ϕ , on an arbitrary control volume, V , can be expressed in integral form as:

$$\begin{aligned} \frac{d}{dt} \int_V \rho \phi dV + \int_{\partial V} \rho \phi (\bar{v} - \bar{v}_g) \cdot d\bar{A} &= \dots \\ &= \int_{\partial V} \Gamma \nabla \phi \cdot d\bar{A} + \int_V S_\phi dV \end{aligned} \quad (17)$$

Using the control volume technique the above partial differential equations are integrated in an arbitrary control volume followed by discretization in space and time. The resulting equations represent the conservation laws for each discrete control volumes or cell in the computational mesh. After linearization, the discrete equations are transformed into a linear algebraic equation system with a sparse coefficient matrix. The flow variables at the cell centroids are then solved numerically using a linear equation solver.

1.2. Geometry and Mesh

Because of the excessive computational requirements involved in case a full vehicle model was to be analyzed, the scope of this paper is limited to studying a brake subsystem in an environmental chamber. In addition, the first validation of this new modeling approach can be carried out more thoroughly by running well-controlled and highly repeatable inertia dynamometer tests as compared to using full vehicle data where various disturbances such as side wind, moisture and ambient temperature may affect the results.

The overall model geometry is illustrated in [Figure 1](#). The test section of the environmental chamber that was used for collecting test data for model validation is virtually recreated for the numerical simulations.

The brake assembly selected for analysis is a generic front corner with a front mounted floating caliper. The original CAD model of the brake corner is modified before mesh generation in several steps. Some small geometric features are removed (edge chamfers, rotor balance groove, gap between solids, spline teeth on shafts) and a few parts are deleted (bolts, abutment shim, bearing cage) or simplified (seals, caliper guide pin, bearing balls). The rotor volume is subdivided into inboard cheek, outboard cheek, vanes and hat regions. The splash shield is converted into a zero-thickness sheet.

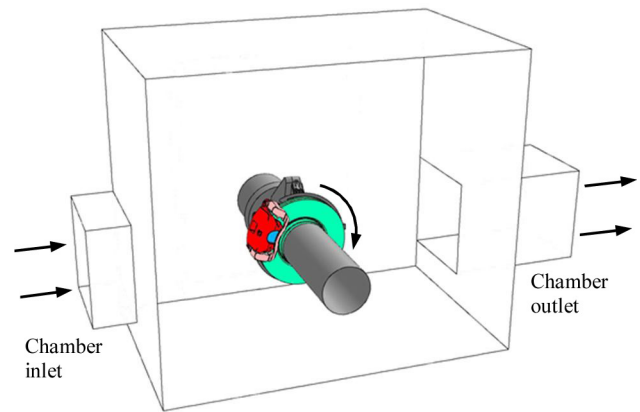


Figure 1. Model geometry.

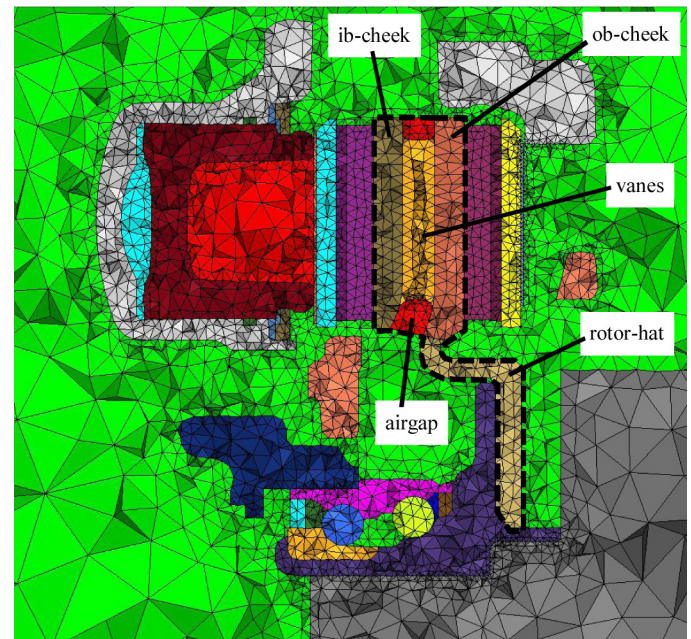


Figure 2. Volume mesh.

[Figure 2](#) depicts a detail from the volume mesh. Here the thick dashed line marks the mesh interfaces that are created between the rotating and stationary zones for the sliding mesh approach. Similarly, in case of the MRF model the regions called ib-cheek, ob-cheek, airgap, and rotor-hat are placed into a rotating reference frame. However, the vanes region has to be in a stationary reference frame since the solution of the energy equation in a moving reference frame is not implemented for solid zones that do not have a shape of a body of revolution.

In Figure 2, the volumes ib-cheek, ob-cheek and vanes represent the thermal working mass of the rotor. An additional interface is created between the ob-cheek and rotor-hat regions to monitor the heat conduction from the working mass of the rotor through this interface. For simplicity, the airgap that is present between the rotor and the pad when the brake is released is not included in the mesh. Instead, the rotor and the pad are always in contact allowing heat conduction to occur between these two parts through their contact interface. The effect of this simplification on the simulation results will be discussed later.

Although the mesh resolution may influence the accuracy of the numerical simulations and special local grid refinements may be needed to improve the agreement between the numerical and test results during model validation, a mesh sensitivity study is beyond the scope of this work.

1.3. Model Settings

Since the effect of gravity and other external body forces are not considered significant in the simulation model, they are not included in the momentum equation. Similarly, the pressure work, kinetic energy and viscous dissipation terms can be ignored in a brake thermal model that involves an incompressible gas model, and thus these terms are omitted in the energy equation. The radiative heat transfer is considered as a gray-diffuse radiation with isotropic scattering and low optical thickness of air (radiative energy exchange between wall surfaces is unaffected by the air that separates them). The following subsections provide more details about the model settings.

1.3.1. Material Properties

The air is modeled as an incompressible ideal gas with its specific heat, thermal conductivity and dynamic viscosity defined as a piecewise linear function of temperature according to dry air property tables at 1 atm pressure. The absorption and scattering coefficients of thermal radiation within air are set to zero, assuming no dispersed particles, droplets or gas species are present that would contribute to absorption or scattering. The air is also considered as a transparent gas and so its refractive index is specified as one.

For solid materials, the density, specific heat, thermal conductivity and emissivity properties are defined as constants or piecewise linear functions of temperature where data are available. The material designations are based on the individual part drawings and include steels, cast iron, aluminum, rubber and plastics. Stationary fluids, such as brake fluid, bearing grease and air that are enclosed inside of a small volume, are also modeled as solids with the appropriate fluid material properties. Only the heat conduction equation is solved within solids and stationary fluids.

It should be noted that although efforts have been made to assign realistic emissivity values to the solid wall surfaces, it is difficult to represent this property accurately since the emissivity depends on surface texture, color and temperature at the time of measurement.

1.3.2. Boundary Conditions

At the model inlet boundary, a constant air velocity and temperature is specified along with the default settings for turbulent intensity and turbulent viscosity ratio. The thermal conditions at the chamber walls are considered to be adiabatic.

The rotational speed within the volumetric zones representing the brake rotor and the airgap around the vanes is prescribed as a function of time. The rotating motion of the rotor boundary walls that are adjacent to a rotating fluid zone is defined with a zero relative velocity in the reference frame of the rotating fluid zone. For the rotor boundary walls that are adjacent to a stationary fluid zone, however, the rotating motion of the wall surfaces is specified using a rotating wall boundary condition as a function of time in the stationary absolute reference frame.

The friction interface on the rotor surface that is in contact with the pad and where the heat input is applied is dynamically re-defined as the rotor is rotated by the sliding mesh model. The transient total heat input at the friction interface is prescribed using a thin wall boundary condition as a function of time. Thin walls are artificially defined by the solver and are not represented in the mesh. They can be used, for example, to model a thin coating on a solid wall, a contact resistance between two solid regions or heat generation through dissipated electrical power in printed circuit boards. In the case of a brake thermal simulation, the artificial thickness, Δx , and the thermal conductivity, k , of the thin wall material has to be specified, and the thermal resistance of the thin wall is then calculated as follows:

$$\Delta x/k \quad (18)$$

Contact resistance may have a significant effect on the temperature levels within the contacting surfaces of the brake rotor and the pad. In this study, a zero contact resistance is considered in the friction interface for simplicity, so that the rotor and pad temperatures can be assumed to be equal at the contacting points. The thin wall thickness is arbitrarily set to 0.001 m in the simulations. Based on a parametric study using the brake thermal model, Figure 3 shows that increasing the value of the thin wall thermal conductivity above 100,000 W/mK does not change the maximum rotor surface temperatures considerably any further. Therefore, it can be concluded that this level of thin wall conductivity is large enough to appropriately represent a zero contact resistance between the contacting surfaces.

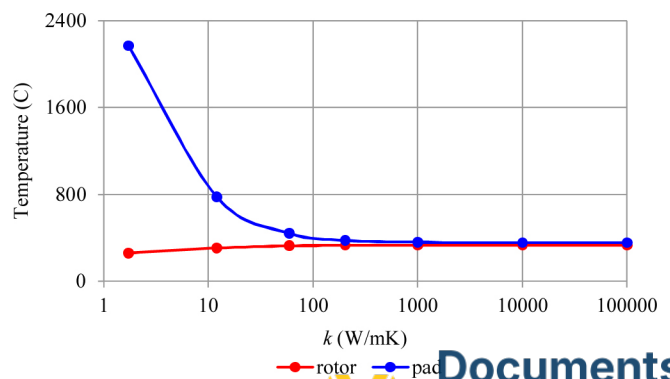


Figure 3. Contact resistance vs. maximum contact surface temperature