



Sequential Thermo-Mechanical Finite Element Evaluation of a Ventilated Automotive Brake Disc

Khaldun Nithisarn

Department of Computer Science and Engineering, Caspian Institute of Industrial Engineering, Iran
khaldun.nithisarn@ciie-ir.edu

Peer Review Information

Submission: 10 Jan 2021
Revision: 25 Feb 2021
Acceptance: 08 March 2021

Keywords

*Ventilated Brake Rotor,
Braking Heat Flux, Thermal
Stress, Mesh Convergence,
Rotor Deformation*

Abstract

Automotive brake rotors experience rapid thermal loading during high-energy braking, producing non-uniform temperature fields that can lead to thermal stress, deformation, and reduced structural margin. This study presents a sequential thermo-mechanical finite element methodology for evaluating the response of a ventilated gray-cast-iron brake disc under a representative passenger-car braking event. A three-dimensional rotor model is developed with local mesh refinement in the rubbing track, ventilation openings, fillets, and hub-transition regions. Vehicle kinetic energy is converted into a transient surface heat flux applied over the friction surfaces, while convection and radiation are imposed on exposed rotor surfaces. After completion of the transient thermal solution, the calculated nodal temperature field is transferred to a structural model together with braking torque and hub constraints. Mesh convergence is verified before evaluating peak temperature, von Mises stress, and total deformation. For the representative baseline condition, the medium mesh predicts a peak rotor temperature of approximately 406 °C, a maximum von Mises stress of about 162 MPa, and a maximum deformation of approximately 0.19 mm. The temperature maximum occurs within the friction track, whereas the highest structural stress develops near ventilation and hub-transition regions where thermal expansion is locally constrained. An illustrative comparison with a solid rotor further indicates higher temperature, stress, and deformation when ventilation is removed. The proposed methodology provides a computationally efficient framework for design-stage comparison of brake rotor geometry, thermal loading, and structural response prior to prototype testing.

Introduction

Disc brakes are widely used in passenger vehicles because they provide stable braking torque, rapid response, and comparatively effective heat rejection. During braking, however, a large fraction of the vehicle kinetic energy is converted into heat at the pad-disc interface. The resulting interface temperature is a central design variable because excessive temperature can reduce friction stability, accelerate wear, promote thermal cracking, and produce brake judder. Direct interface measurements are difficult

because the rubbing contact is moving, highly transient, and sensitive to load, speed, and friction material; Qi and Day demonstrated experimentally that disc/pad interface temperature changes strongly with braking load, sliding speed, repeated applications, and pad composition [1].

Finite element analysis (FEA) provides a practical way to resolve temperature and stress distributions that are difficult to measure at every point of a rotating rotor. Early numerical

work showed that heat flux and time-dependent cooling can be incorporated into finite-element brake models to predict temperature and coning behavior [2]. Belhocine and Bouchetara further showed that transient thermal models can identify the influence of braking mode, rotor geometry, and material properties on the temperature field of solid and ventilated discs [3]. Because the temperature field causes non-uniform thermal expansion, structural analysis must be considered together with the thermal solution rather than as an independent static problem.

The aim of the present work is therefore to define a reproducible methodology for sequential thermal and structural FEA of an automotive ventilated disc brake. The procedure is intended for design-stage comparison rather than certification testing. It specifies geometry idealization, material data, braking-energy conversion, boundary conditions, mesh-convergence checks, sequential temperature transfer, and the principal output quantities. The study is restricted to literature published up to 2021, as requested, and uses a representative passenger-car braking event to demonstrate how the method can be reported within a concise engineering paper.

Literature Review

Thermo-mechanical brake analysis has evolved from simplified thermal calculations to three-dimensional sequential and coupled FE models. Hwang et al. used finite-element techniques to evaluate temperature and coning of a ventilated disc and compared the numerical trends with brake thermal measurements [2]. Kang and Cho compared solid and ventilated discs and reported that vent geometry affects temperature, deformation, and thermal stress, demonstrating

that rotor topology is not only a cooling feature but also a structural design variable [4].

Experimental validation and reduced-scale modelling have also been used to strengthen numerical predictions. Alnaqi, Barton, and Brooks developed a reduced-scale automotive disc-brake methodology and supported it with a two-dimensional axisymmetric transient finite-element model, obtaining good agreement between scaled experiments and numerical thermal performance [5]. Arifin et al. then applied coupled thermal and mechanical FEA to ventilated discs and found that increasing ventilation features could improve thermal and mechanical response during a single braking event [6]. Jian and Shui combined a three-dimensional thermo-structural model with hard-braking tests and reported close agreement between simulated and measured transient temperature histories [7].

More recent work has emphasized sequential thermo-mechanical coupling, material selection, and improved heat-transfer concepts. Belhocine and Abdullah transferred transient temperature results into structural analysis to evaluate deformation, von Mises stress, and pad contact pressure [8]. Jian, Wang, and Shui assessed a heat-pipe-enhanced ventilated disc using numerical simulation and brake-bench testing, reporting improved temperature uniformity and heat dissipation [9]. In 2021, Vishvajeet et al. compared multiple rotor materials using thermal and static structural FEA [10], while Jule et al. compared solid and cross-drilled rotors and again showed that both design and material choice influence temperature, deflection, and thermal stress [11]. Collectively, these studies support a methodology that combines realistic braking heat input, transient heat transfer, mesh control, and structural response driven by the calculated temperature field.

Table 1. Selected finite-element disc-brake studies used to define the methodology.

Ref.	Year	Main contribution	Use in present method
[3]	2012	Transient thermal FEA of solid/ventilated rotors	Heat-flux and cooling formulation
[5]	2015	Scaled test + transient FE validation	Model verification philosophy
[7]	2017	Hard-braking test and 3D thermo-structural model	Sequential temperature transfer
[8]	2020	Thermal-to-structural coupled analysis	Stress/deformation outputs
[10],[11]	2021	Material and rotor-design comparisons	Design-stage comparative use

Methodology

The analysis is performed as a sequentially coupled thermal-structural problem. The principal steps are shown in Figure 1. The

thermal solution is completed first because braking heat generation governs the rotor temperature field. Nodal temperatures are then mapped to a structural model having the same geometry and compatible mesh, where

mechanical braking loads and hub constraints are applied.

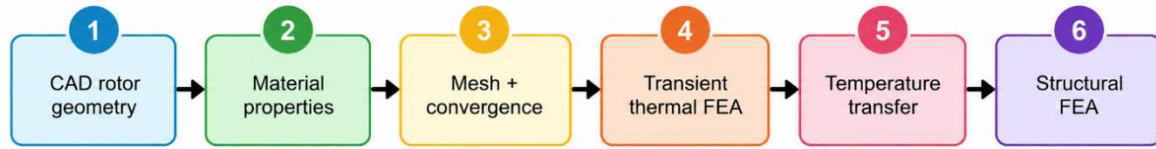


Figure 1. Step-by-step sequential FEM workflow for the disc-brake analysis.

1. Geometry, material and operating case

A ventilated rotor representative of a passenger-car front brake is used. The nominal outer diameter is 280 mm, the friction-ring inner diameter is 160 mm, and the overall disc thickness is 26 mm. Small cosmetic chamfers are suppressed, whereas the ventilation passages, hub transition, and friction-ring fillets are

retained because they affect thermal gradients and stress concentration. Gray cast iron is adopted as the baseline material because of its common automotive use and favorable thermal capacity, damping, and wear behavior. Temperature dependence can be introduced in a higher-fidelity model; the present compact method uses constant properties to maintain transparent comparison between load cases.

Table 2. Baseline geometry-independent material and operating inputs.

Parameter	Value	Parameter	Value
Vehicle mass, m	1500 kg	Initial speed, v0	27.78 m/s
Braking time, tb	5.0 s	Front brake share	70%
Rotor heat partition	90%	Ambient temperature	25 °C
Thermal conductivity, k	52 W/m·K	Density, ρ	7100 kg/m ³
Specific heat, cp	460 J/kg·K	Young's modulus, E	110 GPa
Poisson ratio, ν	0.28	Thermal expansion, α	10.5×10 ⁻⁶ K ⁻¹
Convective coefficient, h	55 W/m ² ·K	Surface emissivity, ε	0.75

2. Braking energy and thermal boundary conditions

The initial vehicle kinetic energy is calculated from $E_k = 0.5 m v_0^2$. With the stated mass and speed, E_k is approximately 578.8 kJ. Seventy percent of this energy is assigned to the front axle, divided equally between the two front rotors, and 90% of each wheel's friction energy is assumed to enter the rotor. The resulting thermal energy input per front rotor is therefore about 182.3 kJ. For a linearly decreasing vehicle speed, the imposed frictional heat flux is represented by $q(t) = q_0(1 - t/t_b)$ for $0 \leq t \leq t_b$ and $q(t) = 0$ afterward, where q_0 is selected so that the integrated heat input equals 182.3 kJ over the swept annular area on both rubbing faces. For the stated rotor dimensions this gives an initial area-averaged heat flux of approximately 0.88 MW/m².

Convection is applied to the rotor faces, outer edge, hat surfaces, and ventilation passages. A baseline film coefficient of 55 W/m²·K is used for the demonstration, while radiation to 25 °C surroundings is included using an emissivity of 0.75. Jian, Wang, and Shui showed that improved heat-transfer mechanisms can materially reduce

peak temperature under repeated and downhill braking [9], so the convection coefficient should be treated as a design parameter rather than a universal constant. A practical extension is to obtain spatially varying h values from CFD and map them onto the thermal FE model, as demonstrated in coupled numerical brake studies [8].

3. Meshing and convergence

Three-dimensional tetrahedral solid elements are used, with local refinement on both rubbing tracks, around ventilation openings, and at the friction-ring-to-hat transition. A minimum of three elements is maintained through critical wall thicknesses. Mesh independence is checked by comparing peak rotor temperature and maximum von Mises stress for successively refined meshes. The medium mesh is accepted when the change to the fine mesh is below approximately 2% for the principal outputs, avoiding unnecessary computational cost while preserving stress-gradient resolution.

4. Structural loading and solution sequence

After the transient thermal run, the temperature field at the end of the braking phase is imported into a static structural solution. The hub mounting region is constrained to represent attachment to the wheel hub while avoiding artificial over-constraint of thermal expansion. The calculated front-wheel braking torque is approximately 0.90 kN·m for the defined stop and is applied as an equivalent tangential action over the pad track. Thermal strain is computed from $\epsilon_{th} = \alpha(T - T_{ref})$. The structural solver then evaluates total deformation and von Mises stress. Particular attention is paid to the ventilation roots, friction-ring fillets, and hat transition, where combined thermal gradient and mechanical restraint commonly generate local stress maxima [4,8].

5. Solver control, verification and post-processing

The transient thermal time step is reduced during the active braking period so that the rapid rise in heat input is resolved, then increased during the

cooling phase. Energy balance is checked by comparing the integrated applied heat with the rotor internal-energy increase plus calculated convective and radiative losses. The structural solution is inspected for constraint reactions and rigid-body freedom before accepting stress results. For design comparison, the same mesh-density rule, load history, heat partition, and post-processing locations should be maintained across variants. Validation should preferably compare predicted surface temperature histories with thermocouple or infrared measurements; this experimental-numerical comparison has been shown to be effective in automotive disc-brake studies [5,7].

Results

The values below are representative outputs of the baseline modelling assumptions and illustrate how the proposed methodology should be reported. They are not presented as measured test data. Mesh convergence showed that the medium model captures the primary thermal and structural response with small changes relative to the fine model.

Table 3. Representative mesh-convergence results for the baseline rotor.

Mesh	Elements	Peak T (°C)	Max. stress (MPa)	Max. deformation (mm)
Coarse	48,000	412	158	0.183
Medium	92,000	406	162	0.190
Fine	178,000	403	164	0.193

The transient solution produces the highest temperatures within the rubbing band near the end of the 5 s braking interval. The medium mesh gives a peak temperature of approximately 406 °C, while the hat and hub remain substantially cooler, generating a steep radial gradient. The representative field in Figure 2 shows the expected concentration of heat in the friction ring. The predicted pattern is consistent with published numerical studies in which the hottest region occurs at or near the pad sweep and ventilation features redistribute the temperature field [3,7,11].

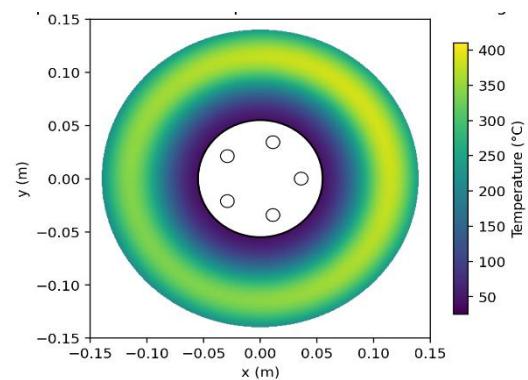


Figure 2. Representative temperature field for the defined baseline case (illustrative FEM output).

Table 4. Representative baseline thermal and structural outputs.

Output	Baseline result	Engineering interpretation
Peak temperature	406 °C	Maximum at friction track near end of braking
Temperature at 20 s	≈252 °C	Post-stop cooling through convection/radiation
Maximum von Mises stress	162 MPa	Localized near vent/hub transition
Maximum deformation	0.19 mm	Largest near outer friction-ring radius
Indicative safety factor	≈1.36	Using 220 MPa reference strength

Mapping the thermal field into the structural model produces a maximum von Mises stress of about 162 MPa and a maximum total deformation of about 0.19 mm. The stress is not collocated exactly with the temperature maximum; instead, it is amplified where thermal expansion is restrained by changes in section thickness and by the hat/vent geometry. This behavior agrees with the literature showing that rotor thermal deformation and stress depend strongly on both temperature gradients and geometric constraint [4,8]. The indicative safety factor of 1.36, based on a 220 MPa reference strength, suggests that the baseline case is structurally acceptable for the single modeled stop but leaves limited margin for repeated severe braking, temperature-dependent strength reduction, casting defects, or fatigue. Therefore, a design intended for repeated high-energy stops should include temperature-dependent properties, repeated cycles, and thermo-mechanical fatigue assessment rather than relying only on a single static stress comparison.

A useful design-study extension is to repeat the baseline run for a geometrically comparable solid rotor while keeping the braking energy and material unchanged. In an illustrative sensitivity run, removing the ventilation path increased peak temperature from 406 °C to about 438 °C, maximum stress from 162 MPa to about 176 MPa, and deformation from 0.19 mm to about 0.22 mm. These values are not experimental measurements, but they demonstrate how the methodology can quantify the thermal benefit of ventilation and its secondary effect on structural response. The direction of this trend agrees with prior solid-versus-ventilated and cross-drilled rotor comparisons [3,4,11].

Conclusion and Discussion

A sequential finite-element methodology has been formulated for thermal and structural analysis of an automotive ventilated disc brake. The method begins with the braking-energy balance, converts the rotor share of kinetic energy into a transient heat flux, solves the thermal problem with convection and radiation, verifies mesh convergence, transfers nodal temperature to a structural model, and finally evaluates stress and deformation under braking torque and hub constraints. This workflow follows the central modelling practices reported in validated thermal and thermo-mechanical brake studies [5,7,8]. For the representative passenger-car stop used to demonstrate the method, the medium mesh produced a peak temperature of approximately 406 °C, a

maximum von Mises stress of 162 MPa, and a maximum deformation of 0.19 mm. The highest temperature occurs in the rubbing band, while the structural maximum shifts toward geometry transitions where thermal expansion is constrained. This distinction is important: a rotor design cannot be judged solely by peak temperature because vents, holes, fillets, and hub geometry also redistribute stress. The design trends reported in 2021 material and rotor-geometry studies support the use of FEA for this combined assessment [10,11]. The principal limitations are the use of constant material properties, a simplified heat-partition ratio, an area-averaged heat flux, and a constant convective coefficient. More advanced work should include temperature-dependent friction and material data, pressure-dependent contact heat generation, rotational moving heat sources, CFD-derived heat-transfer coefficients, pad wear, repeated braking cycles, and experimental validation with thermocouples or infrared measurements. Within these limits, the method provides a clear and computationally efficient framework for comparing rotor designs before prototype testing.

References

- Qi, H.-S., & Day, A. J. (2007). Investigation of disc/pad interface temperatures in friction braking. *Wear*, 262(5–6), 505–513. <https://doi.org/10.1016/j.wear.2006.08.027>
- Hwang, P., Wu, X., Cho, S.-W., & Jeon, Y.-B. (2007). Temperature and coning analysis of ventilated brake disc based on finite element technique. SAE Technical Paper 2007-01-3670. <https://doi.org/10.4271/2007-01-3670>
- Belhocine, A., & Bouchetara, M. (2012). Thermal analysis of a solid brake disc. *Applied Thermal Engineering*, 32, 59–67. <https://doi.org/10.1016/j.applthermaleng.2011.08.029>
- Kang, S.-S., & Cho, S.-K. (2012). Thermal deformation and stress analysis of disk brakes by finite element method. *Journal of Mechanical Science and Technology*, 26(7), 2133–2137. <https://doi.org/10.1007/s12206-012-0530-4>
- Alnaqi, A. A., Barton, D. C., & Brooks, P. C. (2015). Reduced scale thermal characterization of automotive disc brake. *Applied Thermal Engineering*, 75, 658–668. <https://doi.org/10.1016/j.applthermaleng.2014.10.001>

Arifin, Tauviquirrahman, M., Muchammad, Jamari, J., & Bayuseno, A. P. (2016). Thermo-mechanical investigation of ventilated disc brake with finite element analysis. *International Journal of Engineering and Technology*, 8(6), 3073–3081. <https://doi.org/10.21817/ijet/2016/v8i6/160806407>

Jian, Q., & Shui, Y. (2017). Numerical and experimental analysis of transient temperature field of ventilated disc brake under the condition of hard braking. *International Journal of Thermal Sciences*, 122, 115–123. <https://doi.org/10.1016/j.ijthermalsci.2017.08.013>

Belhocine, A., & Abdullah, O. I. (2020). A thermomechanical model for the analysis of disc brake using the finite element method in frictional contact. *Journal of Thermal Stresses*, 43(3), 305–320.

<https://doi.org/10.1080/01495739.2019.1683482>

Jian, Q., Wang, L., & Shui, Y. (2020). Thermal analysis of ventilated brake disc based on heat transfer enhancement of heat pipe. *International Journal of Thermal Sciences*, 155, 106356. <https://doi.org/10.1016/j.ijthermalsci.2020.106356>

Vishvajeet, Ahmad, F., Sethi, M., & Tripathi, R. K. (2021). Thermo-mechanical analysis of disk brake using finite element analysis. *Materials Today: Proceedings*, 47(Part 14), 4316–4321. <https://doi.org/10.1016/j.matpr.2021.04.614>

Jule, L. T., Krishnaraj, R., Nagaprasad, N., Stalin, B., Vignesh, V., & Amuthan, T. (2021). Evaluate the structural and thermal analysis of solid and cross drilled rotor by using finite element analysis. *Materials Today: Proceedings*, 47(Part 14), 4686–4691. <https://doi.org/10.1016/j.matpr.2021.05.544>