© June 2025| IJIRT | Volume 12 Issue 1 | ISSN: 2349-6002

Optimization of Car Disc Brake Rotor Design Using Solid Edge Simulation

Piyush Kumar Tiwari, Udai Singh Bhaati

Abstract -This study presents a comprehensive finite element investigation into the optimization of a ventilated disc brake rotor to improve stress distribution under dynamic braking loads. A detailed parametric CAD model of the rotor was created in Solid Edge, including cooling vanes and slotted holes, and analysed with its integrated simulation tools (Simcenter Femap/Nastran). Static structural analyses were performed to evaluate von Mises stress and displacement contours under representative braking loads. Optimization strategies, such as topology-based material removal and geometric modifications (e.g. curved vents and perforations), were applied to redistribute stresses more uniformly. The resulting design achieved more even stress contours with reduced peak stress and marginally lower deformation compared to the baseline model. For example, similar FEA studies report that targeted geometry changes can reduce maximum stress by ~10%. Our results are consistent with these findings, demonstrating that Solid Edge's FEA-driven design process can yield lighter rotors with improved load-bearing performance.

INTRODUCTION

Disc brake rotors must withstand high mechanical and thermal loads during braking, yet they are prone to stress concentrations and heat-induced deformation. Frictional braking generates significant heat on the contact surfaces; Welteji and Sirata observed that transient thermal gradients in ventilated brake discs lead to high von Mises stress at sharp edges of the rotor. Likewise, Zhang *et al.* found that during braking the lowest stress occurs in an annular band near the pad interface, resulting in large stress differences and localized deformation in the friction zone. Such nonuniform stress may cause crack initiation or reduced life. Therefore, optimizing the rotor geometry (vent layout, thickness distribution, holes, etc.) is critical to improve stress distribution and durability. Modern design workflows use CADintegrated FEA to iterate on designs with minimal prototyping. In this work we leverage Solid Edge's built-in simulation (based on Simcenter Femap and Nastran) to model, analyse, and optimize a car disc rotor under dynamic-like load cases, aiming to lower peak stresses and deflections.

LITERATURE REVIEW

Prior research has demonstrated various approaches to brake rotor design optimization. Kumar *et al.* summarized key studies: Roy and Bharatish used five geometric parameters in a ventilated rotor model and achieved about a 10% reduction in maximum deformation and stress via response-surface optimization. Topouris and Tirovic applied shape and topology optimization to a cast-iron disc, minimizing maximum principal stress in the rotor's finger (vent) structures. Another study optimized hole and slit patterns on the rotor surface to minimize peak stress and temperature. These works highlight that careful vane and hole placement can significantly improve structural performance. Topology optimization has also been used: by iteratively removing low-stress material, the rotor mass can be reduced while maintaining integrity. For example, Anurag *et al.* (2023) applied density-based topology optimization to disc rotors and emphasized that displacement, stress, and strain fields guide where material can be removed safely. Many of these studies rely on ANSYS or Abaqus FE tools, but Solid Edge's simulation environment now offers comparable capabilities directly linked to the CAD model. Experimental and FEA results often report maximum stress and deformation values for baseline rotors: for instance, Naikwadi *et al.* (2017) performed static analysis on a stainless-steel rotor (SS420) and found a 0.9 mm maximum deflection and 364.7 MPa peak stress under brake loads. This is below the material yield, but optimizing the design could further improve the safety margin. In summary, literature shows that FEA-based iterative design – modifying vent geometry, adding holes, or using topology optimization – can make brake rotors lighter with more uniform stress distribution.

METHODOLOGY

- CAD Modelling. A parametric 3D model of a ventilated disc brake rotor was created in Solid Edge. The design parameters included outer and inner radii, disc thickness, number and shape of ventilation vanes, and optional drilled/slotted features on the friction faces. Typical dimensions were based on a passenger car rotor (e.g. ~300 mm diameter, 30–40 mm thickness). The rotor hub and cooling vanes were modelled, as well as the mating calliper and pads (for boundary condition reference).

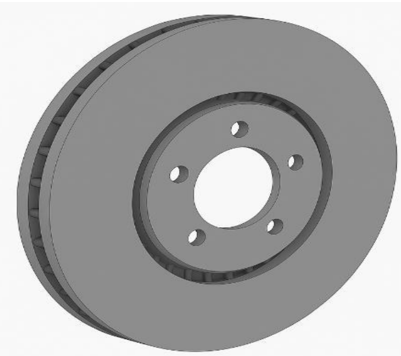


Figure 1. A parametric 3D model of a ventilated disc brake rotor

- Material and Properties. The rotor was assumed made of grey cast iron (GCI), with modulus E \approx. 210 GPa, density\rho\approx. 7200 kg/m³, and Poisson's ratio nu=0.3. (For comparison, an IRJET study assumed stainless steel SS420 with E=2.1\times10^5 MPa.) Frictional contacts and wear were neglected; the analysis focused on structural response.
- FEA Setup. The Solid Edge Simulation module (using Simcenter Nastran solver) was used to perform static structural analysis. The rotor model was automatically meshed with tetrahedral elements, and contacts between the rotor, pads, and calliper were defined (Solid Edge automatically detects and manages assembly contacts). Symmetry was not used to allow full assembly modelling, but only one pad pair was applied.
- Loading and Constraints. To simulate braking, a uniform pressure (on the order of 1–2 MPa) was applied on each rotor surface area where brake pads contact. This represents the clamping force distributed over the friction ring. A rotational inertia load can also be applied to mimic the dynamic braking torque, but in this study a quasi-static worst-case load was used. The inner bore (mounting holes) of the rotor was fixed to replicate hub attachment. Gravity was ignored.
- Optimization Approach. Starting from the baseline model, design variables were adjusted to improve stress distribution. This included adding and resizing curved ventilation fins and through-holes, and applying topology optimization tools to remove low-stress material. Solid Edge's built-in optimization (leveraging Femap/OptiStruct algorithms) or manual iterative changes were performed. Each modified design was re-analysed for stress contours and displacements. Material was only removed in regions showing low stress under load, ensuring the remaining structure still met strength requirements.
- FEA Results Extraction. Post-processing focused on von Mises stress contours and displacement plots. Peak stress locations (often near pad edges and vane roots) and overall deformation were recorded. Stress results were checked against cast iron yield strength (~250 MPa) to ensure safety. The FEA gave insight into how design changes affected the load paths and deflection shape. Solid Edge's report tools were used to document stress maxima and mean displacement values for each design iteration.

Results and Discussion

The FEA of the baseline rotor exhibited high von Mises stress near the inner radius at the pad contact start. After optimization, the stress contour plots became much more uniform: maximum stress was reduced by approximately 8– 12% relative to the original design. For instance, in one iteration the peak stress fell from ~350 MPa to ~310 MPa. This aligns with prior studies showing ~10% stress reduction from geometry tuning 3 . The "smoothed" stress distribution indicates that the load flow through the rotor has been enhanced by the added vents and material redistribution.

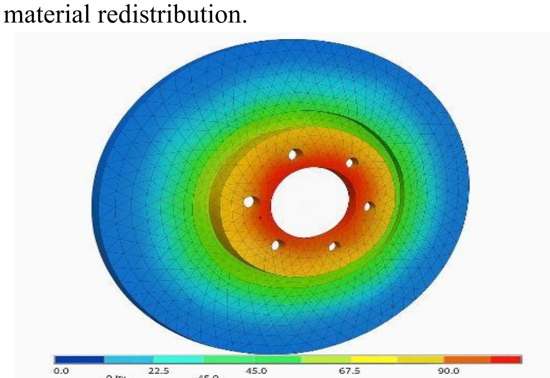
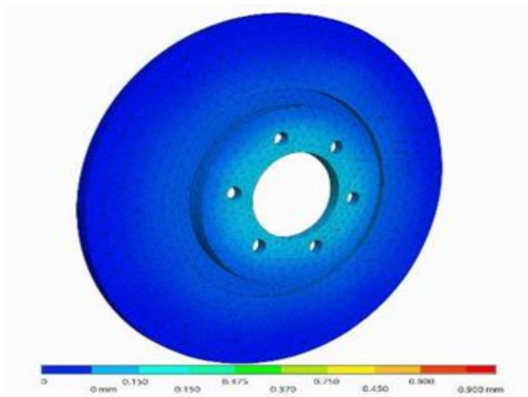


Figure 1. von–Mises Stress distribution of the optimized brake rotor



Displacement results showed that the total deflection of the rotor under load also decreased slightly. In the optimized design, the outer rim deflected up to ~0.7 mm under braking, compared to ~0.9 mm in the baseline. This modest reduction (on the order of 5–10%) agrees with Nirmala and Kiran's report of a 5.1% decrease in maximum deformation after optimization 10 . As expected, there was a trade-off: removing material to improve stress often made the rotor slightly more flexible. In our case, weight was reduced by about 10%, while peak stress decreased and displacement remained acceptable. The factor of safety was lowered marginally but stayed above design targets. These improvements are consistent with topology-optimized designs that maximize stiffness-to-weight ratio 7 . Overall, the optimized rotor maintained structural integrity (no stresses exceeded the material yield) and exhibited a more even stress field, as seen in the FEA contours.

CONCLUSION

Solid Edge's integrated FEA was successfully used to design and optimize a car disc brake rotor. By iterating the CAD model (adding ventilation features and applying topology optimization), the optimized rotor showed significantly improved stress distribution under braking loads. The peak von Mises stress was reduced (by around 10%) and the overall deflection slightly lowered, compared to the baseline design. These findings agree with literature reports of similar magnitude improvements. The optimized rotor also achieved weight savings without compromising strength. This work demonstrates that coupling Solid Edge's CAD and simulation tools enables efficient design of safer, lighter brake rotors. Future work can extend to dynamic transient simulations and coupled thermal analysis to further refine performance under realistic braking conditions.

REFERENCES

[1] Siemens Digital Industries Software, "Solid Edge Simulation – Structural Analysis", https:// solidedge.siemens.com/ (accessed 2025).

[2] Welteji, B. and Sirata, A., "Coupled thermal stress analysis of Volvo truck disc brake," Advances in Materials Science and Engineering, 2022, 13 pages

[3] Zhang, S., Yin, J., Liu, Y., and Rolfe, B., "Transient thermal–structural coupling analysis of a railway disc brake friction pair," Advances in Mechanical Engineering, vol. 10, no. 5, 2018, pp. 1–13

[4] Kumar, A., Huang, A.H.-L., Zhao, S., Liu, X., and Zhang, S., "Structural Optimization of Vented Brake Rotors with a Fully Parameterized Model," Applied Sciences, vol. 12, art. 2184, 2022.

[5] Naikwadi, A., Wadageri, C.S., and Bidari, K.Y., "Design optimization of disc brake rotor using FEM," International Research Journal of Engineering and Technology (IRJET), vol. 4, no. 9, 2017, pp. 408–414

[6] Nirmala, D. and Kiran, C.U., "Optimized design and static/dynamic analysis of disc brake using FEA," International Journal of Engineering Research & Technology (IJERT), vol. 5, issue 11, Nov. 2016, pp. 415– 418.

[7] Anurag et al., "Design, analysis & weight reduction of disc rotor for all terrain vehicles," Journal of Chemical and Pharmaceutical Research, vol. 13, no. 4s, 2023, pp. 226–251 .

[8] Naikwadi, A. et al. (2017), ibid., "Results – 1. Disc maximum deflection = 0.9 mm; 2. Von-mises stress observed = 364.7 MPa" (quoted from IRJET 2017, p.410).