OKANAGAN

ENGR 492

Thermal Stress Analysis of a Disc Brake Rotor

Zach Welch (61563755)

Noah Landman (16274011)

Tyson Senger (23955971)

Nicholas Jean (45063195)

Benny Ng (35797729)

School of Engineering Faculty of Applied Science

The University of British Columbia Okanagan

Submission date: April 10th, 2024

1.0 Introduction

Thermal stress analysis is a method to determine an object's reaction to heat flux based on material properties and geometry. Knowing how an object will change the temperature at different points under a given heat flux is an important step in the design process for parts that will be subjected to heat fluxes as a part of their use and operation. Specifically, disk brakes turn the rotational kinetic energy created by a moving vehicle and turn that energy into sound and heat, thus slowing the rotational kinetic energy. In practice, this means that brake pads are pressed against brake rotors to create friction, thus slowing down the vehicle.

When materials increase in temperature their material properties change in kind. For brakes, this means that, generally, the friction of the brake rotor decreases as the temperature increases. This is an important phenomenon to model as if the temperature of the rotor increases enough the frictional properties of the rotor will change enough such that the rotors will cease properly functioning as a braking device. The goal of modeling thermal stress analysis for brakes is to know under what loads the braking system will remain at serviceable temperatures, and under what conditions the braking system will no longer function. This analysis is a vital safety measure for brake system design as the failure of a brake system is extremely dangerous.

To accurately determine the response to thermal stress, we used three methods of analysis to ensure model accuracy. The three methods of analysis are simple hand calculations, MATLAB, and ABAQUS. The overall goal of the project is for us to better understand finite element methods of analysis, therefore our goal is to use hand calculations and MATLAB simulations to confirm ABAQUS results and compare the outcomes of the models.

2.0 Literature Review

Before discussing our simulation and calculations it is important to understand how thermal stress analysis is currently conducted for brake rotors. Given the importance of brakes in motor vehicles, lots of thermal stress analysis studies have been conducted to better understand this mechanism. Numerous research groups primarily use finite element methods (FEM) to model and study how heat is conducted through the vehicle's brake system. When conducting a FEM simulation on rotor brakes, they will often conduct two tests on a ventilated brake disc and a non-ventilated brake disc [1]. Unfortunately, as the education version of ABAQUS is limited in its capabilities, we will only be able to model the non-ventilated system.

To model and solve the correlation between temperature and stress two main methods are used, indirect and direct coupling [2]. This corresponds to how well researchers believe that their model is coupled to the solution of the problem [3]. This can involve researchers modeling the effect of the wear rate of the brake pads on the heat flux generated. Researchers will typically disregard the indirect 3-D FE analysis of brake pads as they have found that the model leads to measured fluctuations in the surface temperature of the brake pad based on the relative displacement of the brake rotor and pad [4]. They have also found in [4] that the direct axisymmetric model of thermal stress analysis reduces computational power and more closely models reality. Researchers also differentiate between the Lagrangian approach and the Eulerian approach. The Lagrangian approach involves the rotor disk moving relative to the brake pad whereas the Eulerian approach does not rotate but the material instead moves through the mesh [5]. The Lagrangian approach is an accurate model of reality but is very computationally intensive [5]. As a result, our MATLAB & Abaqus simulations will make use of this direct model and make use of the Eulerian approach.

Using the heat conduction problem outlined in [4], we can model the heat flux density of our brake disc. This method assumes that the material of the disc is uniform and isotropic. The heating process is also assumed to be a linear heat conduction problem between the contact surfaces of the brake rotor and pad. We will also only analyze one-half of the brake disc due to symmetry on the other side. A detailed explanation of our MATLAB simulation can be found in Section 4.2 of this report. This system is modeled in Figure 1 below.

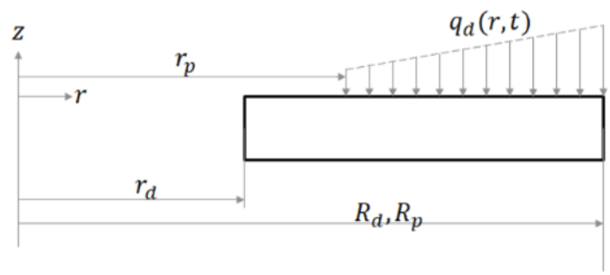


Figure. 1 Model of heat conduction problem [6].

Of particular importance to researchers is the study of hot bands and hoop stresses in brake rotors. Hoop stresses are also known as circumferential stresses as they present themselves in the circumference of a band. This stress is a principal stress that is developed when a cylinder's ends are closed as they are in a brake rotor. Internal stresses arising from the fluctuation in temperatures cause hoop stresses to appear [7]. These internal stresses can lead to cracks and material warping in brake discs. If the hoop stresses reach the maximum yield stress of the brake rotor, catastrophic failure of the brakes will occur resulting in severe damages to property and potential loss of life.

3.0 Methodology

The methodology adopted for the thermal stress analysis of the brake rotor and pads included a systematic and comprehensive approach beginning with modeling. The geometry and material properties of the Brembo rear rotors sourced from a Volvo 740i were replicated from the figure shown below within the finite element model. Acknowledging computational constraints, a decision was made to choose a solid rotor model over its vented counterpart, ensuring adherence to the 1000-node limitation imposed by Abaqus.

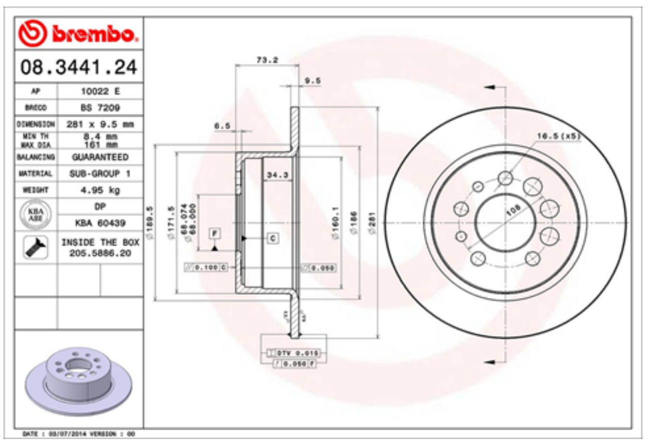


Figure. 2 Brembo rotor used for simulation

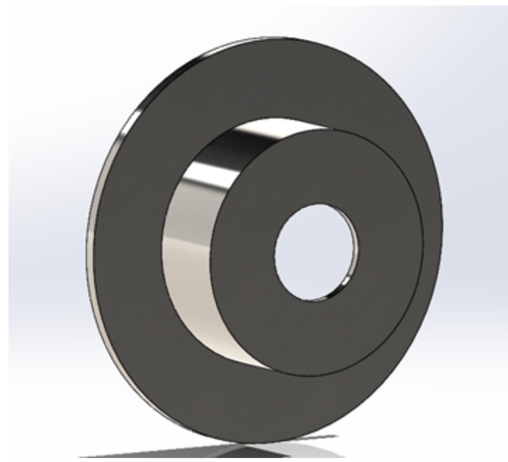


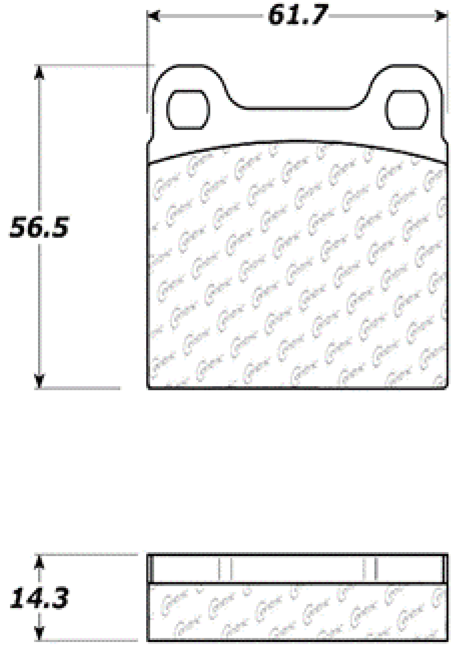
Figure. 3 Rotor modeled in SolidWorks

Additionally, the material of the brake rotor was classified as Gray Cast Iron 250 which is the most common brake rotor material used on vehicles today. The material properties of Grey Cast Iron 250 can be found in Table 1 below.

Property	**Value**
Young's Modulus (GPa)	180
Poisson's Ratio	0.29
Thermal Expansion Coefficient ($\mu m/m - K$)	11
Density (g/cm^3)	7.5
Conductivity ($W/m - K$)	46
Specific Heat ($J/kg - K$)	490
Tensile Strength (MPa)	250 MPa

Table. 1 Material properties of Grey Cast Iron 250

The brake pads, also modeled from Volvo 740i rear brakes, are shown below.



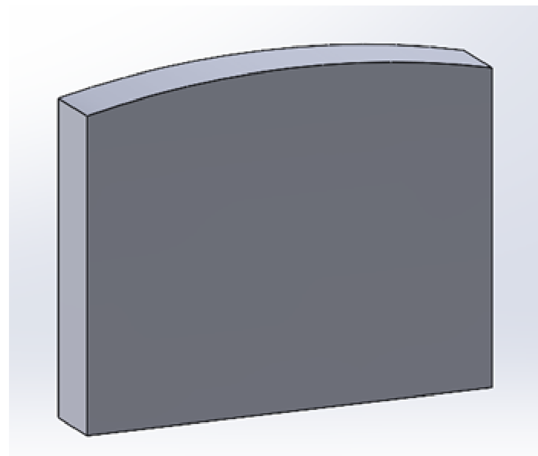


Figure. 4 Brake pad used for Simulation

Figure. 5 Brake pad modeled in SolidWorks

The brake pad material is commonly made up of a combination of materials that are selected to provide optimal frictional performance, durability, and heat dissipation characteristics. Because of this, the composition can vary depending on the intended use, such as regular or high-performance driving. We have taken experimental material property values shown in Table 2 below to model the brake pad material.

Property	**Value**
Young's Modulus (GPa)	28
Poisson's Ratio	0.29
Thermal Expansion Coefficient ($\mu m/m - K$)	11
Density (g/cm^3)	2.7
Conductivity ($W/m - K$)	2.36
Specific Heat ($J/kg - K$)	4000

Table. 2 Brake Pad material properties

Boundary conditions were defined to emulate real-world braking scenarios, simulating a braking force of 1 MPa applied over a 10-second duration while maintaining a constant vehicle speed of 80 km/h. This matches the braking parameters used in the analysis of [2]. Heat transfer phenomena were thoroughly incorporated within the model, including rotor surface convection, forced convection along the rotor's edges, and radiation effects. A friction coefficient of 0.33 between the pads and rotor surfaces was the source of heat generation in this simulation, presumed constant throughout the braking event. To establish consistent initial conditions, a predefined field was introduced to initialize the simulation with a uniform temperature of 60 degrees Celsius across both the rotor and pads. These braking parameters are summarized in Table 3.

Braking Time (sec)	10
Braking Force (MPa)	1.00
Rotor Speed (km/h (rad/s))	80(46.6)
Initial Temperature (C)	60
Friction Coefficient	0.33

Table. 3 Braking parameters defined for simulation

Assumptions in the model, such as uniform temperature distribution and constant heat transfer coefficients, were made to simplify the braking motion into something we can model in Abaqus. Lastly, meshing parameters and settings were optimized to ensure the highest precision in analyzing the thermal stresses induced in the brake rotor system.

4.0 Results & Discussion

4.1 Hand Calculation

In a braking system, the mechanical energy is transformed into calorific energy. This energy is characterized by a total heating of the disc and pads during the braking phase. The heat quantity in the contact area is the result of plastic micro-deformations generated by the friction forces. Generally, the thermal conductivity of the material of the brake pads is smaller than that of the disc. We consider that the heat quantity produced will be completely absorbed by the brake disc. The heat flux evacuated from this surface is equal to the power of friction. The initial heat flux entering the disc can be calculated by first calculating the mechanical energy or kinetic energy of the system.

$$K.E. = \frac{1}{2}M(v \times u)^2 = 271604.9\,J$$

We first decided on using a smaller compact car, using a mass of 1100 kg, with an initial velocity of 80 kilometers per hour. Making sure to convert that to meters per second before calculating, we get a value for the kinetic energy of our vehicle. We then need to convert the kinetic energy into heat power, which is taking the kinetic energy over the time of braking. Our braking time as stated above is 10 seconds. In this step, we also isolate one brake in the system. We choose one of the front rotors, which most cars have a front braking bias of roughly 75%, and we must also divide that in half to isolate one of the two front rotors.

$$H.P. = \frac{K.E.}{time} * (0.75/2) = 10185.2\, W$$

In the final step, we take the value of the heat power over the area of contact between the brake disk and the pad. The larger radius will be the outer radius of the disk, and the smaller radius will be the radius where the brake pad stops contacting the disk.

$$H.F. = \frac{H.P.}{A} = \frac{H.P.}{2\pi(R^2 - r^2)} = 140348.3\, W/m^2$$

4.2 Matlab Simulation

The reason behind using MATLAB as an alternate simulation software for our project was that the system is very complex and hand calculations do not offer much insight into how the simulation will perform. Using MATLAB, we created an approximate model using only half the thickness of the brake disk and one brake pad for simplicity. We simplify the brake disk into two separate rectangles, one where the brake pad and disk make contact, and the second rectangle for the part of the disk that is open to air. Next, the geometry is meshed, and the thermal model is applied to the E6 edge, shown in *Figure 6,* which is the contact surface. Next, we run the model and solve for the temperatures at a few key radii to help us get an idea of what we should expect to see in ABAQUS.

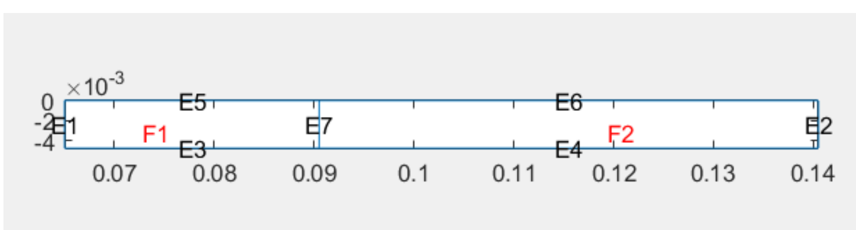


Figure. 6 MATLAB Simplified Thermal Model

To calculate the heat flux function, we input all of the properties of the pad and disk, using all of the same values as the ABAQUS simulation. The results are then plotted out on a single figure for us to compare the temperatures at the different instances.

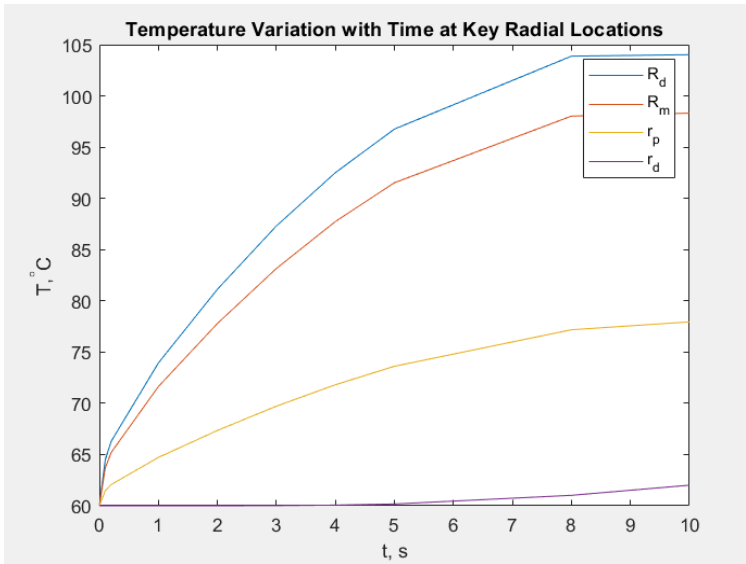


Figure. 7 Temperature Variation with Time at Key Radial Locations

In the graph, we can see that the radius that has the highest value is Rd, which is the outermost radius of the disk, at roughly 105 degrees. In the practical analysis, this is to be expected, as the largest radius will experience a higher rotational force, which will increase the heat flux and temperature. The radius in the middle of the pad, (Rm) is sitting just below it. The value to note in this graph as an outlier would be the inner radius of the pad, rp. This temperature seems to be a lot lower than expected. In the real world, the change in rotational velocity at different radii would not change the heat flux constant by enough. The main driving force of the heat is the friction between the pads, so this value can be disregarded, and we can assume that the temperature under the pad is somewhere closer to 100 and 105 degrees and is fairly constant over the entire brake pad contact. Another piece of information to take away from this graph is that the inner radius of the brake disk does not experience much temperature change. This is most likely due to the heat transfer through the brake disk itself needing more time to fully propagate through the entire disk, in addition to the disk having high heat dissipation properties. As shown in *Figure 7*, right near the end of the ten-second mark, the value of rd begins to rise, which backs up the assumption.

Another visual that we had the opportunity to implement is a cross-sectional heat mapping of the brake disk at some different key times (*Figure 8*). The x-axis is the length of the brake disk, and the y-axis is the thickness of the disk (both in cm). This allows us to visualize the disk heating up along the simulation in a similar way to what the ABAQUS simulation grants, but looks into the cross-section of the brake disk. Since we are using the same data and model as above, the same discrepancies can be seen in these graphs, notably the difference in temperature between the outer and inner edges where the pad and disk are in contact.

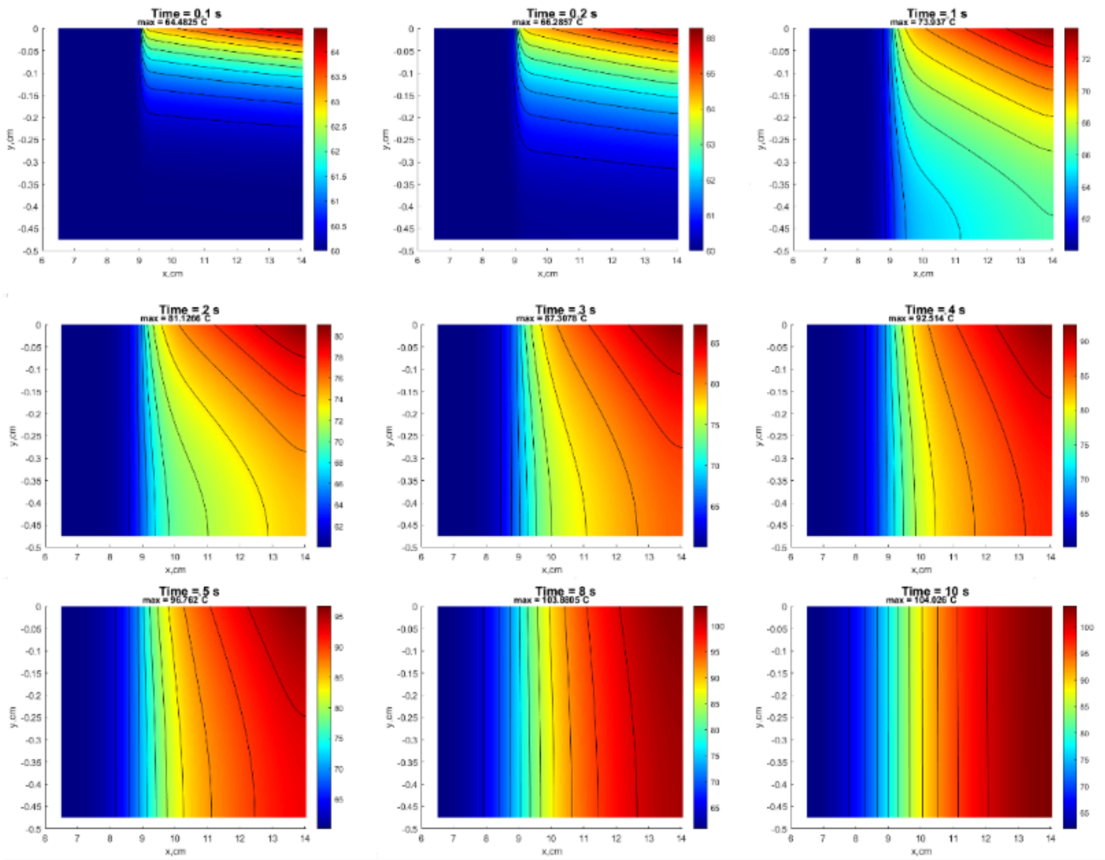


Figure. 8 MATLAB Cross-Sectional Temperature Distributions of Brake Disk

4.3 Abaqus Simulation

The simulation was carried out in Abaqus according to the braking parameters defined above. The most important thing to note is the hot bands formed on the surface of the brake rotor, these hot bands are the rings in the middle of the braking surface that are hotter than the inner and outer surfaces. These hot bands are important to simulate as they are the locations that are most likely for fatigue cracks to form on the surface of the brake rotor. Figure 9 shown below is the temperature distribution of the front and back of the brake rotor at different timesteps throughout the simulation. Overall, both sides show what we would expect to see with the rotor heating more from the middle of the braking surface and forming the hot bands around the rotor. Additionally, both sides show very similar temperatures which was expected. The maximum temperature reached at the end of the simulation was 107 C, which was also compared for validity with the previous Matlab simulation showing only a 3C difference.

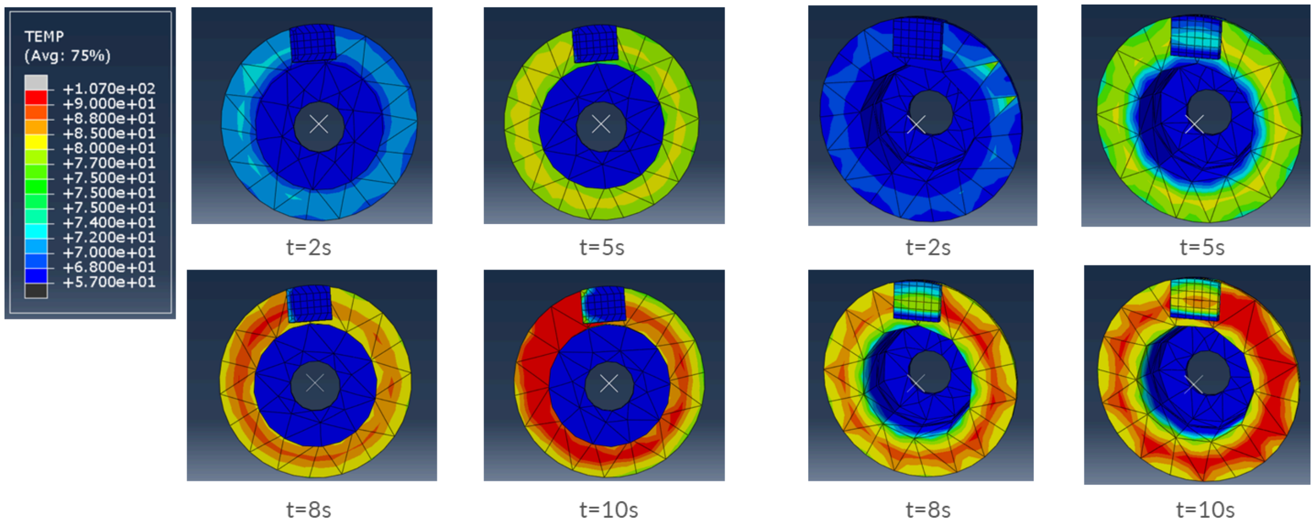


Figure. 9 Temperature distribution of brake rotor

In addition to the temperature distribution, the simulation carried out in Abaqus also modeled the circumferential or "hoop" stress in the brake rotor. As mentioned previously, the hoop stress represents the stress in the circumference of the rotor. It can be seen in Figure 10 below the maximum hoop stress in the rotor is around 20 MPa from the legend on the left, this is far below the tensile stress of the brake rotor material which is 250 MPa as specified above.

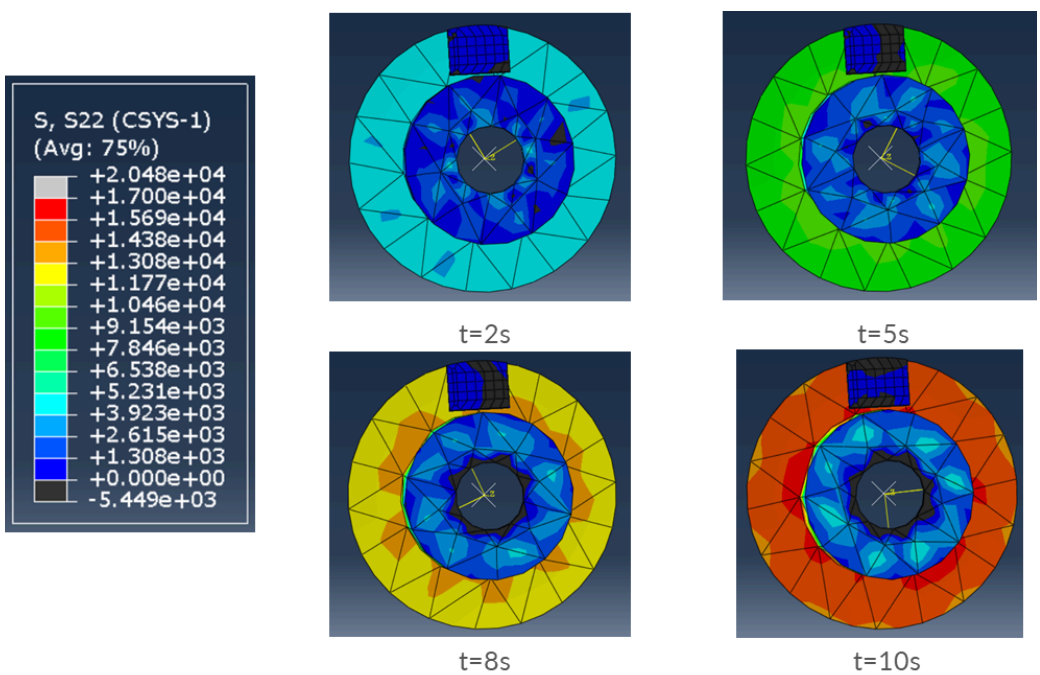


Figure. 10 Hoop stress distribution of brake rotor

However, more crucial than comparing the hoop stress to the material tensile stress is if any variations in stress are present on the surface of the brake rotor, it can be seen in this braking simulation that the hoop stress remains uniform throughout the rotor surface for the whole duration. This is an ideal case as in a harder braking simulation we would expect to see variations start forming and following the locations of the hot bands. Overall, we must model the hoop stress as the variations that can form on the surface of the rotor are a major cause of radial cracking.

5.0 Conclusion

After the simulations, we can conclude that the hot bands will form on the disc rotor surface, particularly in the middle of the braking surface. We have the maximum temperature recorded at 107°C in the ABAQUS simulation which is only 3°C different compared to the MATLAB simulation. On the other hand, the hoop stress in the disc rotor was far under the material's tensile stress which proved that the Brembo disk brakes system can handle our real-world braking scenarios pretty well in an ideal situation.

The results from the simulation support that the middle of the disk rotor is the most critical part due to high thermal stress. When performing an inspection of the disc brake system, we can expect that the fatigue cracks will most likely form in this critical area. When designing or improving the performance of a disk brake system, we should consider how to remove the thermal stress from the critical area quickly to avoid any potential damage to the disc rotor and improve braking performance.

Although the tested hoop stress is way lower than the material's tensile stress in this simulation, it only demonstrated the performance of the brake under our assumptions. We are not confident to state that the brake can handle a more realistic situation. Further refinement of models that include additional factors, such as material nonlinearity and transient effects, may improve the accuracy of the simulation. Moreover, advanced disc brake systems in the market are often equipped with a vented disc rotor. By making holes in the disc surface, the thermal stress can be transferred to the surroundings more efficiently. The higher-end disc brake system also uses ceramics instead of Gray Cast Iron 250 to produce the disc rotor. Further research and simulation can be done to compare the performance of those advanced designs and the model that we used in this project.

Overall, the project successfully demonstrated the thermal stress analysis of a disc brake system by using both MATLAB and ABAQUS. The simulation provides valuable insight to determine the behavior and performance of a simple disc brake system. Although further studies can be done to determine the real-life situation and advance design performance, this project can be a significant starting point for studying the thermal analysis of an engineering problem.

6.0 References

[1] I. T. Jiregna and H. G. Lemu, "Thermal Stress analysis of disc brake using analytical and numerical methods," in IOP Conf. Series: Materials Science and Engineering [Online]. Available: https://iopscience.iop.org/article/10.1088/1757-899X/1201/1/012033/pdf

[2] M. R.Moghanlou and H. S. Googarchin, "Three-dimensional simulation of thermal stress in discs during an automotive braking cycle," in *Trans & Motauto World*, Varna, Bulgaria, July 2018. Available: https://www.researchgate.net/publication/336835787_THREE-DIMENSIONAL_SIMULATION_OF_THERMAL_STRESSES_IN_DISCS_DURING_AN_AUTOMOTIVE_BRAKING_CYCLE

[3] H. Y. Yang, E. Tempero and, R. Berrigan, "Detecting indirect coupling" in *Australian Software Engineering Conf.*, April 2005, doi: 10.1109/ASWEC.2005.22

[4] A. Adamowicz, "Axisymmetric FE model to analysis of thermal stresses in a brake disk," *Journal of Theor. and App. Mech.* vol 53 - 2, pp. 357 - 370. Available: http://ptmts.org.pl/jtam/index.php/jtam/article/view/1965

[5] A. Rashid and N. Stromberg, "Thermomechanical simulation of wear and hot bands in a disc brake by adopting an eulerian approach," in *Eurobrake*, Dresden, DE, June 2013. Available: https://www.researchgate.net/publication/282879634_Thermomechanical_Simulation_of_Wear_and_Hot_Bands_in_a_Disc_Brake_by_adopting_an_Eulerian_approach

[6] "Thermal and structural analysis of a disc brake," Mathworks, https://www.mathworks.com/help/pde/ug/thermal-and-structural-analysis-of-disc-brake.html#ThermalAndStructuralAnalysisOfDiscBrakeExample-6 (accessed, April 6, 2024).

[7] "Hoop Stress" Trenchlesspedia, https://trenchlesspedia.com/definition/2799/hoop-stress (accessed April 6, 2024).

[8] *Iron alloy*. MakeItFrom.com. (2020, May 30). https://www.makeitfrom.com/material-properties/ASTM-Grade-35-or-250-ISO-250-EN-JL-1040-F12401-Grey-Cast-Iron