INVESTIGATION OF STRESS, DEFORMATION, AND CRACKS IN AISI 4140H-A BRAKE DISC OF USING FINITE ELEMENT METHOD

[1]Avinash Gopal Patil, [2]Dr. S. Singh,

[1, 2] Department of Mechanical Engineering

[1, 2] Bundelkhand University, Jhansi, India.

Abstract: Premature failure is the major problem encountered in the operation of brake discs of the high performance motorbike. Because of road conditions, harsh braking and uneven cooling cycles motorbike brake disc are subjected to cyclic temperature & mechanical loading. Thermo-Mechanical fatigue cracks can often be found on the friction surface of brake discs used in vehicles after a period of usage and include crackle, radial and circumferential patterns. These cracks typically exhibit different initiation and propagation behavior under different braking conditions. In this study, the effect of clamping pressure and thermal stresses on fatigue crack evolution is analyzed by using numerical simulations. A FEA analysis was performed using ANSYS Workbench 18 to determine the temperature & stress allocation in disc during braking. It also gives a good prediction of Crack evolution, crack shape prediction. Kinetics of propagation shows that crack in the specimen propagates circumferentially outward through the surface edge having maximum SIF.

Index Terms - **Fatigue Fracture, Crack initiation, Thermal and Mechanical Loading, temperature Gradient, Stress Intensity Factor, FEA.**

I. INTRODUCTION

In modern developments light motor vehicles and heavy motor vehicles are equipped with a disc brake system, which is a component that can ensure safety during driving. The braking system, therefore, becomes important element in a vehicles. A disc brake is a type of braking system that utilizes components, like a disc clamped with two pieces of brake pads serving to retard or stop the rotation of the wheel. Some studies on disc brakes have been conducted to measure brake torque and shaft speed during braking. Also, theoretical analysis and mechanical systems have been developed and solved numerically using finite-difference methods and Matlab software.

During the braking process, wet disc brakes can withstand excessive wear, high temperature, and thermal elastic deformation, which results in deterioration of braking or stability that affects passenger comfort and shortens the life of disc brakes. Simulation analysis of disc brakes is needed to determine the effect of damage caused by friction forces and compressive forces which results in thermal deformation in disc brakes. Until now, most numerical simulations have shown a dominant influence of the pad-disk interface on the stability of the brake system.

The finite element method is a powerful tool to obtain the numerical solution of wide range of engineering problem. The method is general enough to handle any complex shape or geometry, for any material under different boundary and loading conditions. The generality of the finite element method fits the analysis requirement of today’s complex engineering systems and designs where closed form solutions of governing equilibrium equations are usually not available. In addition, it is an efficient design tool by which designers can perform parametric design studies by considering various design cases, (different shapes, materials, loads, etc.) and analyze them to choose the optimum design.

In practical applications, finite element method, boundary element method or a modified J-integral theory were used to study the stress, strain field and stress intensity factor (SIF) induced by thermal fatigue. The computation of SIFs for a cracked body subjected to a thermal transient loading by the finite element method requires step-by-step computation for the entire time range, and this procedure should be repeated for each step of the crack growth.

II. METHEDOLOGY

2.1) Transient Thermal Analysis

2.1.1) FEA Model

A rear disc brake system consists of a rotor that rotates about the axis of a wheel, a caliper–piston assembly where the piston slides inside the caliper, which is mounted to the vehicle suspension system, and a pair of brake pads. When hydraulic pressure is applied, the piston is pushed forward to press the inner pad against the disc, and simultaneously, the outer pad is pressed by the caliper against the disc. In this braking action the kinetic energy due to friction is converted into the heat energy. Numerical simulations using the ANSYS APDL/Workbench 18 finite element analysis software package were performed in this study for a simplified version of a disc brake system.

❖ Model:-

The actual model is prepared in CATIA V5 & used in the ANSYS software in modeling for Transient thermal Analysis code with the following dimensions axis wise. The model is nothing but brake disc.

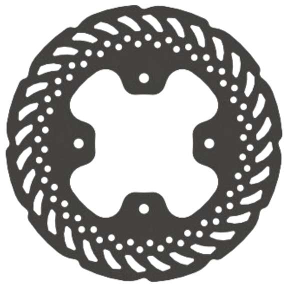


Figure-1: CAD Model of brake disc

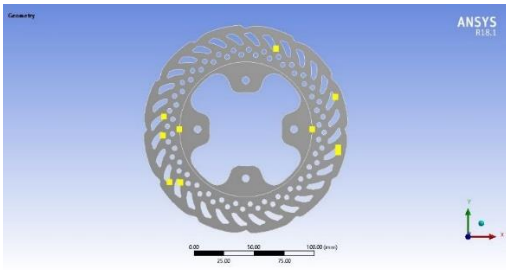


Figure-2: ANSYS Model of brake disc

2.1.2) MESHING

The requirement of meshing is in the area which is in contact with the caliper during braking so this area is coarse meshed. System generated mesh are square mesh but for this analysis its required very coarse mesh models for the better accuracy so that the tetrahedral mesh are used.

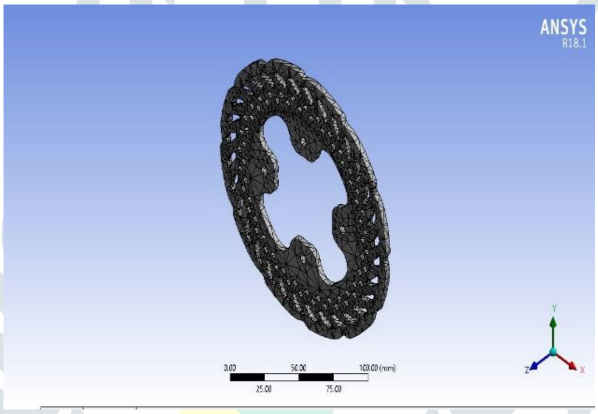


Figure-3: Meshing of brake disc

TABLE 1: Nodes & Elements

Nodes	26758
Elements	13410
Span Angle Center	Coarse
Minimum Edge Length	2.e-002 mm
Mesh Model Type	Tetrahedron

The minimum edge length is found to be 2.e-002 mm for the each Tetrahedron elements. In the above figure 3 it is observed that the outer surface of the periphery of the disc the element size is more. The minimum element size is maintained near to the ventilated holes of the brake disc.

2.1.3) Boundary Conditions

➢ Fix Support

As the brake disc is supposed to fix between the faces of the caliper, the two faces which are in the contact with friction material are supposed to be fixed. So that the fix support is applied to the two opposite faces of the part of the rotor disc which in the caliper. Also the fix support is provided to the bolted part of the brake disc.

➢ Pressure

As the air is continuously flows over the disc which is act as the cooling media for the disc so that consider the atmospheric pressure over the faces of the disc.

- Thermal Load

  The thermal loading conditions are used from the values obtained during the experimentation,

TABLE 2: Thermal loading Conditions

Time (S) Step	0	0.1	0.2	0.3	0.4	0.5	0.6	0.7	0.8	0.9	1
Temperature(0C)	300	335	370	405	445	485	515	555	590	615	650

2.1.4) Results

2.1.4.1) Temperature Distribution

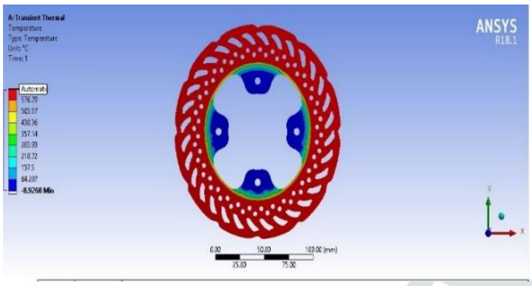


Figure-4: Temperature Distribution of Brake Disc

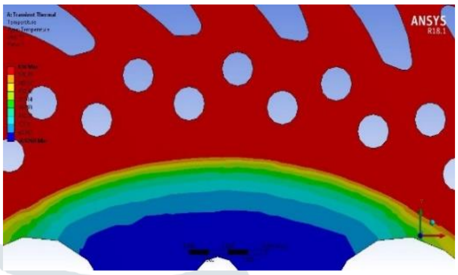
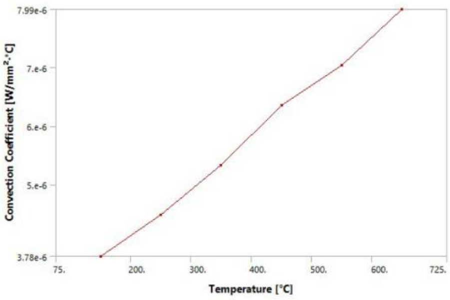


Figure-5: Heat convection rate of Brake Disc

TABLE 3: Temperature for different ranges of convective Heat transfer coefficient.

SR. NO.	Temperature (0C)	Convection Coefficient [W/mm²·°C]
1	150	3.78e-006
2	250	4.49e-006
3	350	5.34e-006
4	450	6.35e-006
5	550	7.03e-006
6	650	7.99e-006

Convection is less at the flange area of the disc and it is more at the bridge. Also its heat convection rate is reducing radially outward from center.



Graph No 1: Convection Coefficient Vs Temperature

From this graph, it is observed that the convective heat transfer coefficient is increases with the rate of increase of temperature.

2.1.4.2) Heat Flux

Brake disc is under the study of the thermal loading, after the input the values of the thermal loading heat flux is then find out. Figure shows the distribution of the heat flux along the surface of the disc.

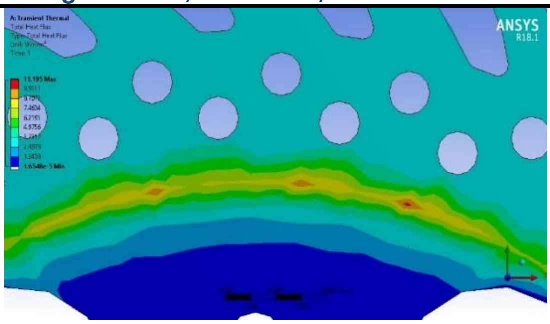


Figure-6: Maximum Heat Flux

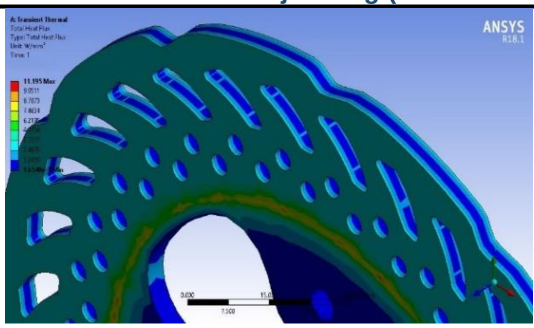


Figure-7: Heat Flux trough thickness of the disc

Figure 6 shows the closed view of the surface heat flux distribution. It is observed that heat flux value is maximum in between the areas of the flange and bridge which located near the ventilated holes of the brake disc. It is then moderate in radial outward direction and low at the bolted area. Figure 7 shows the heat flux distribution along the thickness of the disc. It observed that the heat flux penetration through the thickness of the brake disc then in the surface area it gives us higher value than the depth along thickness of the disc.

Following table indicates the minimum and maximum evaluation of the heat flux during the applied time steps,

TABLE 4: Heat Flux for time step Minimum and Maximum

Time [s]	Minimum [W/mm²]	Maximum [W/mm²]
1.e-002	3.6277e-005	34.292
2.e-002	3.9511e-006	24.198
3.e-002	1.0823e-005	21.282
4.e-002	6.8998e-006	19.6
5.e-002	2.8548e-006	18.518
6.e-002	4.2623e-006	17.764
7.e-002	2.7346e-006	17.203
8.e-002	2.1649e-006	16.763
9.e-002	1.6501e-006	16.4
1.e-001	3.143e-006	16.091

2.2) Crack Analysis by ANSYS 18 Workbench

Fracture Mechanics is the branch of engineering which studies the fatigue failure of the material; it focuses on the crack analysis with its initiation and propagation. In the present study fatigue failure due to crack initiation and propagation is studied, experimentally, these experimental results now simulated using finite element package ANSYS version 18.In the previous analysis by applying thermal loading a transient thermal analysis was done. The results shows that thermal gradient is present after the dissipation of the heat. In this study effect of the thermal gradient on the brake disc need to study for that structural Analysis and further crack analysis by using fracture tool is applied.

2.2.1) Mesh Model

In this analysis the meshing of the brake disc model is need to more focus on the area of the heat flux present. To the identified areas a fine meshing is applied. For the better accuracy tetrahedron element is used. Following table presents the values of nodes & elements.

TABLE 5: Nodes and elements of fracture model

Nodes	246009
Elements	136190
Span Angle Center	fine
Minimum Edge Length	2.e-002 mm
Mesh Model Type	Tetrahedron

The size of the elements is 2.e-002 mm with the fine meshing. The total peripheral of the disc around the ventilated holes is mesh.

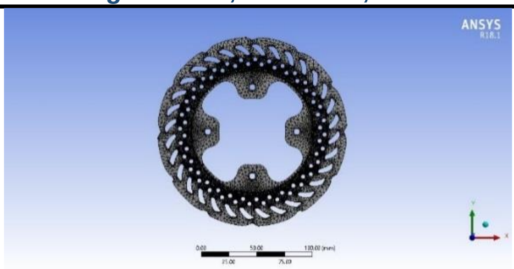


Figure-8: Static Structural Mesh Model

Figure-9: Static Structural Mesh Model

Above figure shows the meshing along the thickness of the brake disc. In this mesh model of structural analysis number of nodes and elements are more than that of transient thermal analysis.

2.2.2) Inputs & Boundary Conditions.

- Fix Support

As the brake disc is supposed to fix between the faces of the caliper, the two faces which are in the contact with friction material are supposed to be fixed. So that the fix support is applied to the two opposite faces of the part of the rotor disc which in the caliper. Also the fix support is provided to the bolted part of the brake disc.

- Clamping Load

A clamping load calculated in the analytical calculation is used as the input in this analysis.

- Thermal Load

a) Temperature

The thermal loading conditions are used from the values obtained during the experimentation,

TABLE 6: Fracture Thermal Loading

Time (S) Step	0	0.2	0.4	0.6	0.8	1
Temperature(0C)	100	200	300	400	500	600

2.2.3) Heat flux

- Material Properties

The material is used for the disc is AISI 4140H.

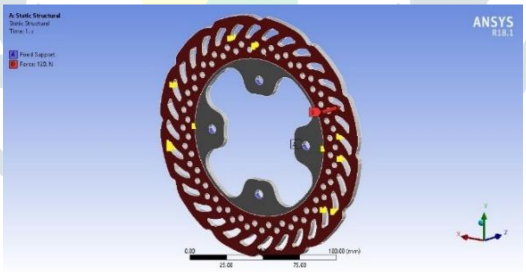


Figure-10: Boundary conditions on the brake disc.

III. RESULTS

3.1) Equivalent Stress

In the static structural analysis Equivalent Van misses stresses are found.

TABLE 7: Equivalent Stress

MAXIMUM	4862.8MPa	MINIMUM	870.01MPa

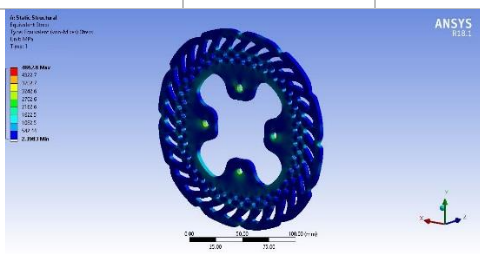


Figure-11: EQUIVALENT (Van Misses) STRESS

It is observed that maximum equivalent stresses found near the ventilated hole. As it is penetrating the depth the values approaches towards the moderate range.

Following table no. 8 shows the time steps minimum and maximum range of equivalent stresses.

TABLE NO.8: Minimum and Maximum equivalent stresses.

Sr. No.	Time [s]	Minimum [MPa]	Maximum [MPa]
1	0.1	0.53389	1080.2
2	0.2	0.74047	1500.4
3	0.3	0.94741	1920.7
4	0.4	1.1545	2341.
5	0.5	1.3617	2761.3
6	0.6	1.569	3181.6
7	0.7	1.7763	3601.9
8	0.8	1.9836	4022.2
9	0.9	2.191	4442.5
10	1	2.3983	4862.8

3.2) Fracture Tool

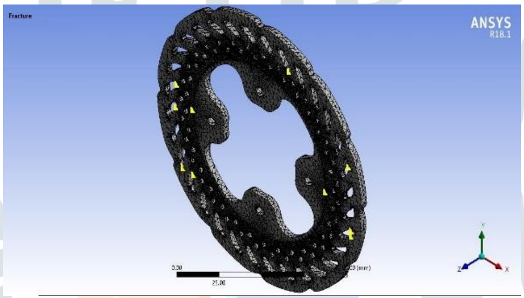


Figure-12: Fracture Mesh

In the facture tooling, mostly, it need to perform the fracture mesh. In this meshing pre-mesh cracks are found. Figure shows the pre- mesh cracks indicated by yellow spots. These spots are also the same one that are picking in the experimentation also with the in the transient thermal analysis.

One of the crack spot is selected for the further analysis purpose. Figure 13 gives us the identified crack. It is surface crack also penetrates the thickness of the brake disc. Figure 14 shows the close view of the crack.

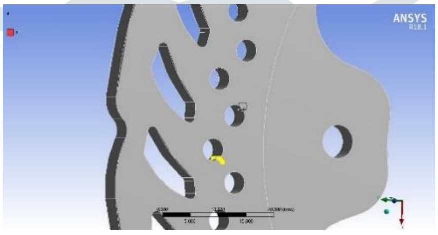
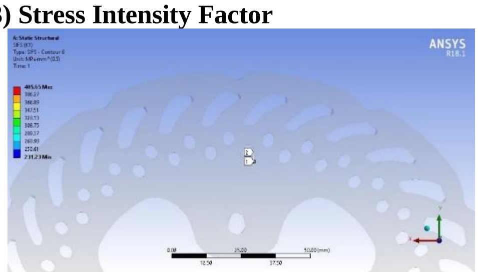


Figure-13: Crack Selection of the Identified Crack

3.3) Stress Intensity Factor



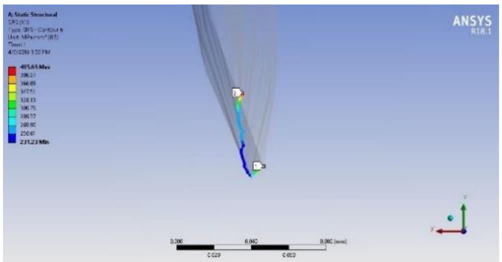


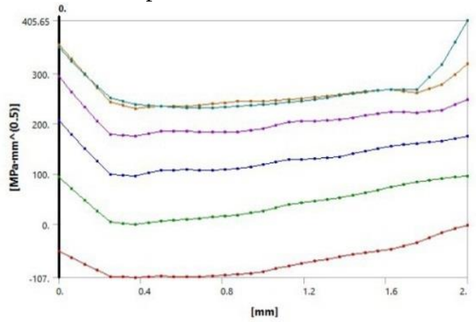
Figure-14: Stress Intensity Factor from 1 to 2 **Figure-15: Close view of Stress Intensity Factor from 1 to 2**

The main purpose to introduce Fracture tool in the simulation process evaluates the Stress Intensity Factor along the edge of the crack. Stress Intensity Factor is evaluates at each counters of the fracture edge which is along the thickness of the rotor disc having t=1.5mm.Following table shows the values of the stress intensity factor at each counter from 1 to 2.

TABLE NO.9: SIF at each counters of crack

Length [mm]	SIFS (K1) Contour 1 [MPa·mm^(0.5)]	SIFS (K1) Contour 2 [MPa·mm^(0.5)]	SIFS (K1) Contour 3 [MPa·mm^(0.5)]	SIFS (K1) Contour 4 [MPa·mm^(0.5)]	SIFS (K1) Contour 5 [MPa·mm^(0.5)]	SIFS (K1) Contour 6 [MPa·mm^(0.5)]
0	-54.32	94.396	206.47	294.7	357.51	352.63
0.25	-105.24	3.8052	99.391	178.16	242.27	251.07
0.5	-103.9	5.4714	107.15	185.04	235.07	234.96
0.75	-104.39	13.315	107.12	183.18	239.05	231.23
1	-94.96	25.841	118.17	190.21	244.69	238.39
1.25	-74.134	45.531	130.54	205.68	252.32	248.25
1.5	-56.869	61.61	144.55	215.74	264.78	262.69
2	-1.787	95.517	175.45	247.94	319.34	405.65

Graph shows the Stress intensity factor at the each counters along the edges of crack. These edges are the depth penetrating. From this graph it is observed that maximum value of SIF at the tip of the crack.

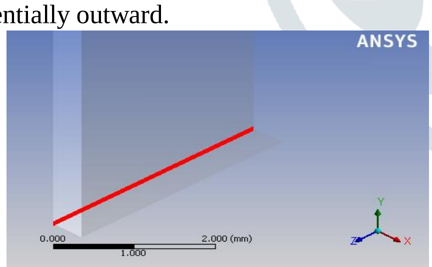


Graph No 2: SIF Vs Thickness of Crack

All the values of SIF are positive expect in counter 1, so that tensile as well as compressive stresses were acting on the disc. Due to these alternating stresses fatigue is occurs in the brake disc material.

3.4) Kinetics of Propagation of Crack

When the value of the stress intensity factor is crosses threshold limit then crack starts to propagate. Kinetics of propagation shows that crack is propagates in the direction radially outwards from the crack edge. Also it can be observed that crack edge is travel circumferentially outward.



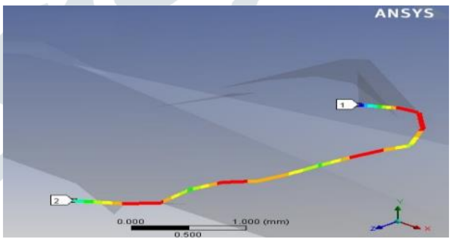


Figure-16: Initial Position of Crack at the Edge **Figure-17: Final Position of Crack at the Edge**

Let select the one Stress intensity factor counter having maximum value.Fig.-17 & Fig.-18 show that initial & final position of crack at edge in the thickness in which it is propagates circumferentially outward direction when SIF cross its threshold value. Maximum value of SIF indicates in figure by red color, other colors indicate intermediate values of SIF in between maximum SIF & minimum SIF.

IV. CONCLUTION

Fatigue failure of disc made of AISI 4140H was focusing on crack initiation & propagation under thermal loading, numerical simulation by FEA coding was studied.

In the numerical simulation process it is notice that the SIF is maximum near the ventilated holes of tip of crack, so that this point leads to the crack propagation. The numerical approach by FEA coding technique gives a fine prediction in terms, Crack evolution, crack shape prediction, and kinetics of crack growth. In addition, it is much less time consuming than experimental. This advantage allows using this approach in parametric study and in industrial applications.

During braking, the exterior of the friction surface is subjected to compressive circumferential stress which leads to fatigue crack initiation and propagation.

V. References

[1] Yildiz, Y. and Duzgun, M., (2010), Stress analysis of ventilated brake discs using the finite element method.Int. J. Automotive Technology 11, 1, 133−138.

[2] A. A. Agbeleye, D. E. Esezobor, S. A. Balogun, J. O, Agunsoye, J. Solis, and A. Neville (2017) Journal of King Saud University

[3] G. Babukanth & M. Vimal Teja, (2012)" Transient Analysis of Disk Brake By using Ansys Software" International Journal of Mechanical and Industrial Engineering (IJMIE), ISSN No. 2231 –6477, Vol-2, Issue-1.

[4] A. Belhocine dan M. Bouchetara, (2013), Journal of Applied Research and technology 11, 674-682, 2013.

[5] S. Courtin , C. Gardin, (2005) "Advantages of the J-integral approach for calculating stress intensity factors when using the commercial finite element software ABAQUS" Engineering Fracture Mechanics 72, 2174–2185.

[6] Zhiyong Yang, Jianmin Han, Weijing Li, ZhiqiangLi,LikePan,Xiaoling Shi, (2013) "Analyzing the mechanisms of fatigue crack initiation and propagation in CRH EMU brake discs", Engineering Failure Analysis 34, 121–128.

[7] Kennedy, F. E., Colin, F. Floquet, A. And Glovsky, R. (1984), Improved Techniques for Finite Element Analysis of sliding Surface Temperatures. Westbury House page 138-150.

[8] Amol,A.,Ravi,R.,FE, (2008), " Prediction of Thermal Performance and Stresses in a Disc Brake System", , Journal of SAE ,2572,01,pp 5-8.

[9] Adam Adamowicz, Piotr Grzes, (2011), "Influence of convective cooling on a disc brake temperature distribution during repetitive braking", Appl. Therm. Eng. 31, 2177e2185.

[10] J.H. Choi, I. Lee, (2004)," Finite element analysis of transient thermo elastic behaviors in disk brakes", Wear 257 47e58.