E-ISSN 2581 - 7957
P-ISSN 2277 - 3916

DOI: 10.32377/cvrjst 1918

Modelling and Analysis of Brake Drum using ANSYS Workbench

C. Sai Kiran[1], M. Ravi Kumar[2], P. Srinivas Reddy[3] and T. Venkatesh[4]
[1]Asst. Professor, CVR College of Engineering/Mechanical Engg. Department, Hyderabad, India
Email: csaikiran001@gmail.com
[2]Asst. Professor, CVR College of Engineering/Mechanical Engg. Department, Hyderabad, India
Email: marripallyravikumar@gmail.com
[3]Asst. Professor, CVR College of Engineering/Mechanical Engg. Department, Hyderabad, India
Email: srinivasrdd87@gmail.com
[4]Asst. Professor, CVR College of Engineering/Mechanical Engg. Department, Hyderabad, India
Email: venkatesh2711991@gmail.com

Abstract: **In this paper, a brake drum was modelled and analyzed which is useful for slowing or completely stopping a moving vehicle. Brake drum is a critical part of the safety system, which is used to prevent the motion of vehicle. The purpose of the work is to find a suitable brake drum material which will be able to stop the vehicle within a minimum distance in case of emergency. Brake drum is modelled in CREO software and analysis of brake drum was performed on three different materials i.e., Grey cast iron, Aluminum alloy and Carbon ceramics by using ANSYS software. Total deformation and Von-mises stress analysis help to predict the brake drum failure when an internal pressure acts on it.**

Index Terms: **Brake drum, CREO, ANSYS, Von-mises stress.**

I. INTRODUCTION

A brake is a device which inhibits the motion of a vehicle. Brake absorbs kinetic energy of the vehicle and decelerates the vehicle motion. Brake drum is connected to the wheels of an automobile for slowing or stopping the vehicle. A brake drum is one type of a brake, which uses friction caused by the set of brake shoes, pressed against a rotating drum. Capacity of any brake device depends on the unit pressure between the braking surface and coefficient of friction.

Sankara et al. [1] has modelled a brake drum design to withstand high temperature and thermal stresses when brakes are applied. The brake drum is modelled in Solidworks2016 software. Structural and thermal analysis is performed in ANSYS workbench software. After modelling the brake drum, pressure of 1.5 MPa and 90 degrees temperature is applied for carbon steel and grey cast iron material. Von-mises stress and total deformation values are observed in the results.

Meenakshi and suman [2] has designed a brake drum for a two-wheeler through reverse engineering approach. Six different materials for brake drum are considered and different stresses, deformation values, rise in temperature on different braking time and heat transfer rate values are obtained. The brake drum is modelled by using CAD software and analyzed in ANSYS workbench 14.5 software. In the results, deflection and stress induced in the brake drum with gradual and sudden braking is observed.

Raju et al. [3] has designed a 3D model of brake drum by using AutoCAD software. In brake drum, kinetic energy is converted into friction and the heat is to be dissipated into the atmosphere. Steady state and transient analysis are performed on brake drum by considering three different materials by using ANSYS software. Steady state analysis was performed to validate the temperature distribution of the brake drum. To determine the thermal flux for different time intervals, transient analysis is performed.

Sagar and dabade [4] have modelled a brake drum using UG-NX 8.5 software. By using analysis software, transient thermal analysis has performed on the brake drum and brake drum has been modified based on the results.

Venkataramana [5] has been modelled and analyzed the brake drum by using ANSYS APDL 10 software. Brake drum of tata indica car and tata nano car is considered for performing thermal and structural analysis.

Anup and sabarish [6] have modelled a brake drum using Pro/ENGINEER software. Structural and thermal analysis is performed by considering the shoe pressure on the brake drum.

Gowthami and balaji [7] has designed a 3D model of brake drum by using AutoCAD software. Thermal flux and steady state conditions are applied for three different materials in ANSYS by applying one Newton pressure. After observing the results for three different materials, aluminium is proved as a better material than other brake drum materials.

Yang et al. [8] has modelled a brake drum using Solidworks software. Modal analysis of brake drum is performed by using ANSYS workbench software. Modal analysis is performed for first 12 natural frequencies to understand deformations for twelve different mode shapes.

Rambabu et al. [9] has focused on designing a standard brake drum by using weight reduction approach. All the different parts of brake drums are designed and assembled in 3D CAD modelling software. Hexahedral finite element model of brake drum is prepared by using CAE processing software. The design iterations are solved by using non-linear/linear finite element solver.

Chiranjeevi and sreenivasulu [10] have modelled a brake drum by using AutoCAD software. Steady state and transient analyses were performed by using Ansys 10.0

software. In analysis, four cyclic braking conditions to determine the thermal deformations and peak temperature developed in the brake drum.

Sinha and gurumeet [11] have created a brake drum model with fins by using Creo parametric software. CFD analyses are performed by considering different materials for brake drum. CFD is performed to determine heat flux, heat transfer coefficient and heat transfer rate by using Ansys Fluent software.

Bakoet al. [12] has developed a modified brake drum model which has extended fins along its circumference by using Solidworks 2013 software. These fins on the brake drum help to improve the heat dissipation rate from the surface of brake drums. The results of stress plot, displacement plot, strain plot and thermal plot are compared with modified finned brake drum and normal brake drum.

Abdul et al. [13] has designed the brake drum by using CATIA V5 software. By considering different loading conditions, structural analysis is performed by using ANSYS software.

A. Different Types of Brake Drum

Brake drum is connected to either rear wheel of a vehicle or to every wheel of a vehicle. A brake drum is coupled with a spring, which is to be internally expanded or contract the brake shoe.

Figure 1. Brake Drum and its Parts

Brake drum is represented in Fig. 1. The main parts of a brake drum are brake shoes, wheel cylinder and shoe holding spring. Brake drums are mainly divided into three types depending on the brake shoes pressing the brake drum:

1. Twin leading shoe type.
2. Leading/trailing shoe type.
3. Duo-servo type.

Twin leading shoe type consists of two leading shoe and two wheel cylinder. In twin leading shoe type, the piston is expanded in both directions and applies equal pressure. This type is mainly used for front brakes of small and medium sized trucks.

In leading/trailing shoe type, two different brake shoes are used. The brake shoes are pressed in the same direction as the rotation of the brake drum, which applies greater pressure on the brake drum. This type is mainly used for the rear brakes of the passenger cars.

In duo-servo type, the primary and secondary brake shoe is linked together by an adjuster. Strong pressure from servo of the primary brake shoe is transmitted to the secondary brake shoe by the link and large braking force is generated. This type is mainly used for parking brakes on the passenger cars.

B. Working Principle of a Brake Drum

Whenever a brake pedal is pressed by the driver, then the brake booster or servo system helps the master cylinder to send the brake oil to the wheel cylinder. Piston is expanded in the wheel cylinder and it will apply pressure on the two curved brake shoes. These two curved brake shoes will be pressed against the inner surface of a rotating brake drum. This contact pressure will create the necessary friction, which enables to slow down or stop the moving vehicle. Therefore, it is very important to design all the components of the brake drum such a way that it can efficiently dissipate the heat to the atmosphere. The working principle of a brake drum is shown in Fig.2.

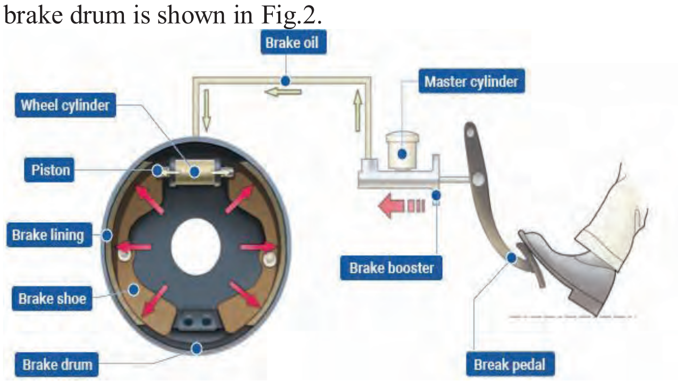

Figure 2. Working Principle of a Brake Drum

C. Brake Drum Material

Brake drums are generally made of materials which are strong enough to withstanding pressure on it and to dissipate heat effectively to atmosphere. Grey cast iron, aluminum alloy and carbon ceramics are the materials selected for brake drum as shown in Table I.

TABLE I.
MATERIAL PROPERTIES FOR DIFFERENT MATERIALS OF BRAKE DRUM

S. No	Properties	Grey Cast Iron	Aluminum Alloy	Carbon Ceramics
1.	Density (Kg/m^3)	7,200	2,800	2,450
2.	Elastic Modulus (GPa)	110	68	30
3.	Poisson's Ratio	0.28	0.33	0.27
4.	Thermal conductivity (w/mK)	54	170	40
5.	Coefficient of thermal expansion (10^{-6}/K)	10.5	23	2.8

II. MODELLING AND FINITE ELEMENT ANALYSIS

A. Brake Drum Model

All the different parts of the brake drum are modelled separately as per the standard dimensions using CREO parametric 2.0 software and all the individual parts of the brake drum are assembled in the CREO parametric 2.0 software as shown in Fig. 3

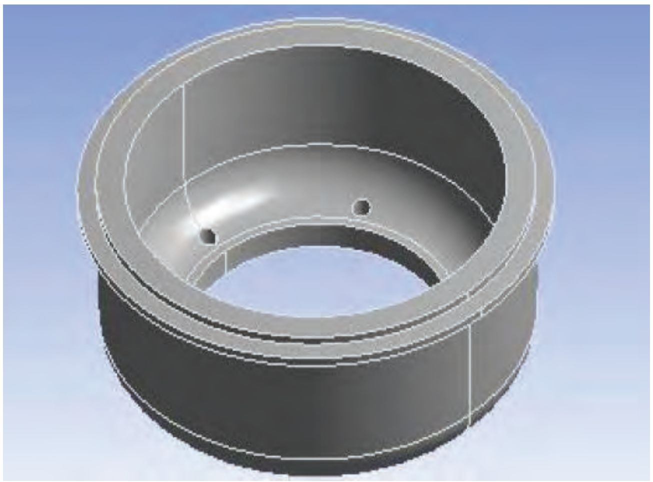

Figure 3. Assembled Model of Brake Drum in CREO

B. Elememt Type

The element type selected for brake drum is SOLID 186. It is a 20-node higher order solid element which has three degrees of freedom per node. The three degrees of freedom are nodal x, y and z translations.

It exhibits quadratic displacement behavior and have spatial orientation. SOLID 186 support large strain capabilities, plasticity, large deflection, hyper elasticity, stress stiffening and creep. SOLID 186 have capability for simulating deformations of incompressible elastoplastic materials and incompressible hyper-elastic materials.

C. Meshing

In meshing, the created 3D model is divided into the certain number of divisions or elements for accurate analysis result. By applying meshing on the model, we can determine the effectiveness and efficiency of any analysis. An automated mesh is generated on the created model which is shown in Fig. 4.

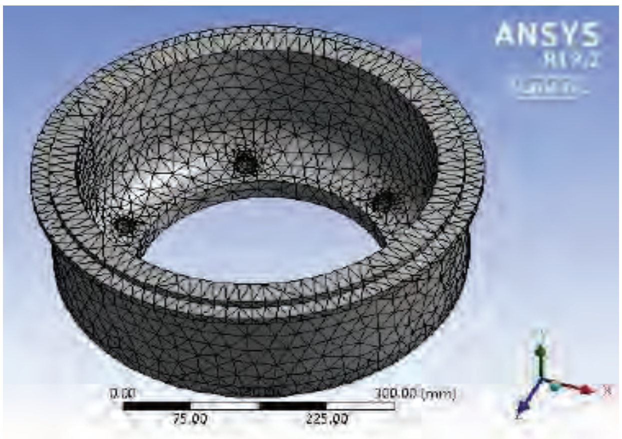

Figure 4. Meshing of Brake Drum

In the automatic mesh, the mesh applied was a fine mesh for achieving precise and accurate results. Instead of using a fine mesh on all the components of the model, coarse mesh was applied on larger area and fine mesh was applied only on the area of higher stress concentration.

D. Applying Loads

On bottom of the brake drum, a fixed support is assigned to withstand the pressure acting on the brake drum as shown in Fig. 5.

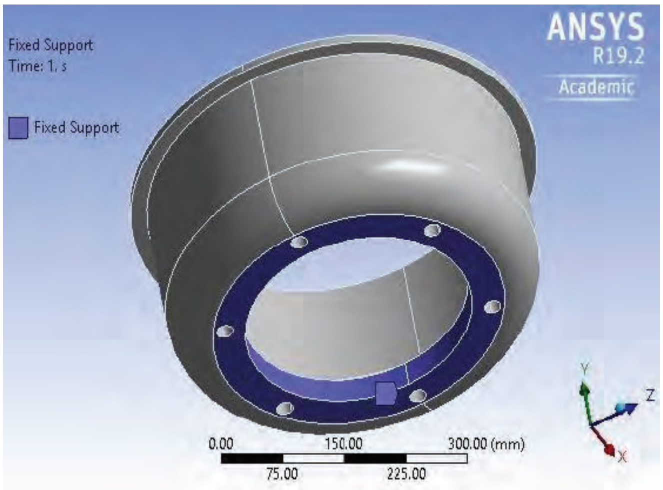

Figure 5. Fixed Support on Bottom of Brake Drum

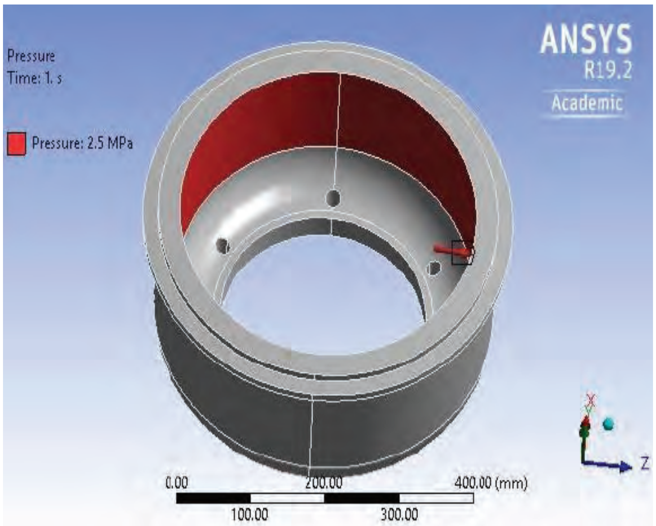

Figure 6. Applying Internal Pressure on Brake Drum

Brake shoe applies internal pressure of 2.5 MPa on the brake drum. For applying the load on the brake drum, the maximum pressure a driver can apply is calculated applied on the brake drum as shown in Fig. 6.

III. Results and Discussions

After assigning fixed support on bottom and applying internal pressure on the brake drum, structural analysis is performed on the brake drum for three different materials in ANSYS workbench to determine the total deformation and von-mises stress. The following results were observed in the analysis.

CVR Journal of Science and Technology, Volume 19, December 2020
DOI: 10.32377/cvrjst 1918

A. Total Deformation of Brake Drum

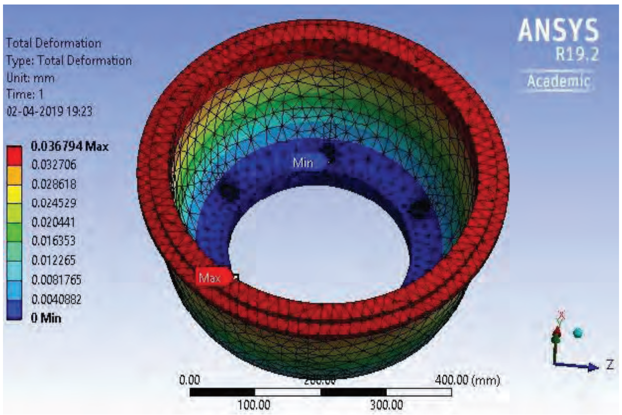

Figure 7. Total Deformation of Grey Cast Iron Material

After performing structural analysis by applying the load on brake drum for grey cast iron material, a maximum total deformation of 0.036 mm is observed from the Fig. 7.

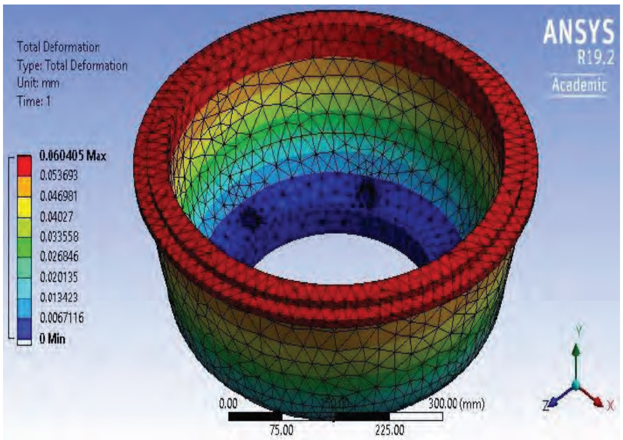

Figure 8. Total Deformation of Aluminum Alloy Material

After performing structural analysis by applying the load on brake drum for aluminum alloy material, a maximum total deformation of 0.06 mm is observed from the Fig. 8.

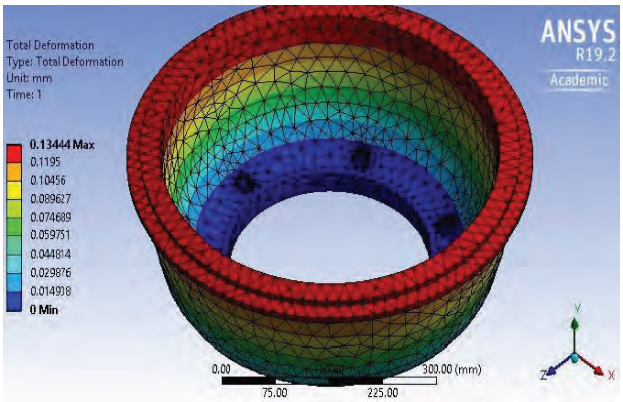

Figure 9. Total Deformation of Carbon Ceramics Material

After applying the load on brake drum, structural analysis is performed on carbon ceramics material. A maximum total deformation of 0.133 mm is observed from the Fig. 9.

B. Von-mises Stress of Brake Drum

After performing structural analysis by applying the load on brake drum for grey cast iron material, maximum von-mises stress of 20.2 MPa and minimum von-mises stress of 0.002 MPa are observed from the Fig. 10.

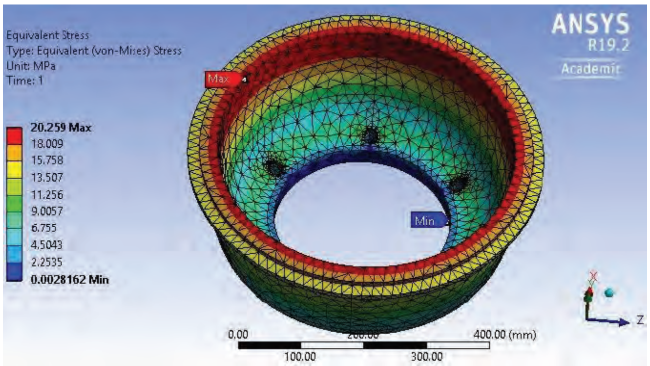

Figure 10. Von-mises Stress of Grey Cast Iron Material

After applying the load on brake drum for aluminum alloy material, maximum von-mises stress of 20.36 MPa and minimum von-mises stress of 0.001 MPa are observed from the Fig. 11.

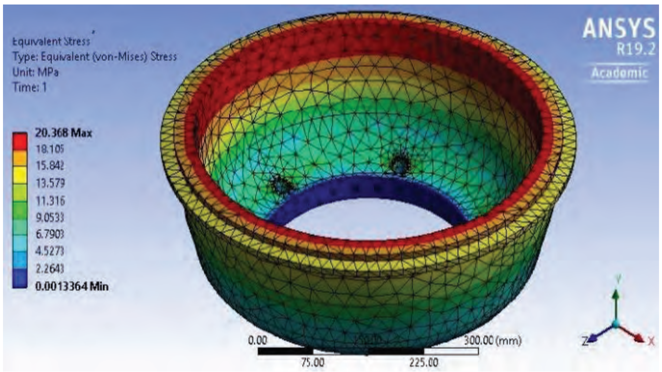

Figure 11. Von-mises Stress of Aluminum Alloy Material

After applying the load on brake drum for carbon ceramics material, maximum von-mises stress of 20.236 MPa and minimum von-mises stress of 0.003 MPa is observed from the Fig. 12.

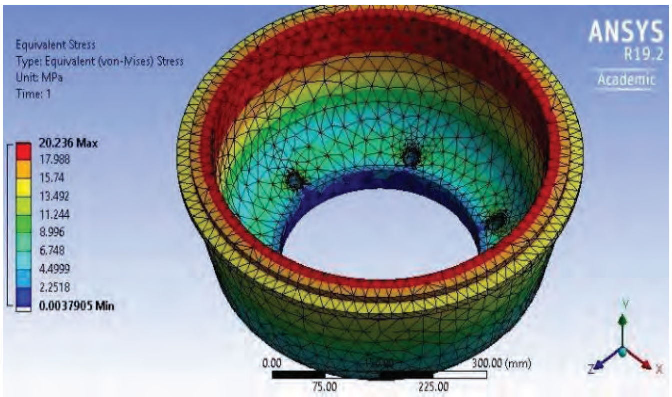

Figure 12. Von-mises Stress of Carbon Ceramics Material

The graph of total deformation for three different materials is observed from the Fig. 13.

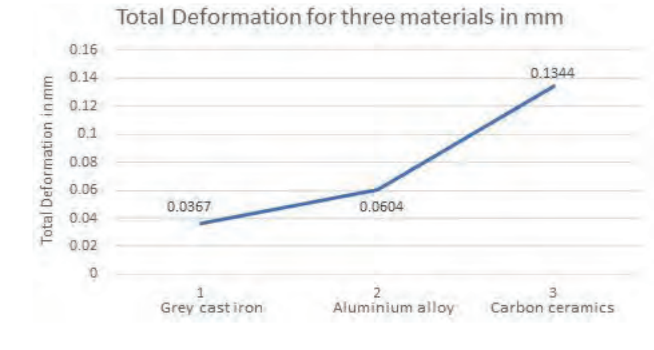

Figure 13. Total Deformation Graph for Different Materials

The graph of von-mises stress for three different materials is observed from the Fig. 14.

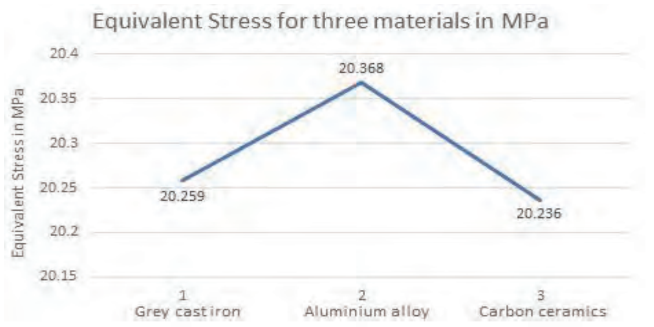

Figure 14. Equivalent Stress Graph for Different Materials

The results of total deformation in mm and Equivalent stress in MPa for three different brake drum materials are represented in Table II.

TABLE II.
RESULTS COMPARISON FOR BRAKE DRUM MATERIAL

Material	Total Deformation (mm)	Equivalent Stress(MPa)
Grey Cast Iron	0.0367	20.259
Aluminum Alloy	0.0604	20.368
Carbon Ceramics	0.1344	20.236

IV. CONCLUSIONS

In this paper, Creo parametric 2.0 software is used for modelling the brake drum and analysis of brake drum with an internal pressure of 2.5MPa is analyzed by using ANSYS workbench software.

From the results, it is observed that the maximum total deformation is observed in carbon ceramics because of presence of carbon and the minimum total deformation is observed in grey cast iron material. Maximum von-mises stress is observed in aluminium alloy and the minimum von-mises stress is observed in carbon ceramics material. The values of von-mises stress and total deformation of the brake drum obtained are within their allowable values. Therefore, modelled brake drum design is safe based on the strength and rigidity criteria.

REFERENCES

[1] K. Sankara Narayanan, J. L. Miren Kishan, Ben Mathew Augustine and D. Vijayaganapathy, “Parametric modelling of brake drum and dynamic analysis for different materials,” *International Journal of Pure and Applied Mathematics*, vol. 120, Issue 6, ISSN 1314-3395,pp. 3919-3931, June 2018.

[2] Meenakshi Kushal and Suman Sharma, “Optimization of design of brake drum of two wheeler through approach of reverse engineering by using ansys software,” *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*, vol. 12, Issue 4, e-ISSN 2278-1684, p-ISSN: 2320-334X.pp. 70-75, August 2015.

[3] P Lakshminarayana Raju, P Hussain, A Ramanjaneya Reddy, B Sidda Reddy and S Sudhakar Babu, “Design and analysis of a brake drum by using finite element method,” *International Journal of Mechanical Engineering Research and Technology*, vol. 2, Issue 2,ISSN 2454-535X, pp. 51-72, May 2016.

[4] Sagar N. Taleand B. M. Dabade, “Customization & analysis of brake drum,” *Journal of Basic and Applied Engineering Research*, vol. 3, Issue 7, p-ISSN 2350-0077, e-ISSN 2350-0255,pp. 575–579, June 2016.

[5] P Venkataramana, “Design of a brake drum using finite element method,” *International Journal of Mechanical Engineering and Robotics Research*, vol. 2, Issue 2, pp. 101–105, April 2013.

[6] Anup Kumar, and R. Sabarish, “Structural and thermal analysis of brake drum,” *Middle-East Journal of Scientific Research*, vol. 20, Issue 8, ISSN 1190-9233, pp. 1012–1016, 2014.

[7] K. Gowthamiand K. Balaji, “Designing and analysis of brake drum,” *International Journal for Research in Applied Science & Engineering Technology*, vol. 4, Issue 9, ISSN 2321-9653,pp. 135–142, September 2016.

[8] Dan Yang,Zhen Yu, Leilei Zhangand Wentao Cheng, “Modal analysis of automobile brake drum based on ansys workbench,” *Advances in Computer Science Research*, vol. 75, pp. 608–612, 2017.

[9] D. Rambabu, R. Gopinath, U. Senthil Rajan and G. B. Bhaskar, “Weight reduction of a standard brake drum: a design approach,” *International Journal of Engineering & Technology*, vol. 3, Issue 2, pp. 201–207, 2014.

[10] M. Chiranjeevi and B. Sreenivasulu, “Thermal analysis of brake drums at various temperatures,”*Iconic Research and Engineering Journals*, vol. 2, Issue 5, ISSN 2456-8880, pp. 79–82, November 2018.

[11] Yogeshwar Sinha and Gurmeet Singh Gahir, “Modeling and analysis of brake drum with extended fins on the circumference of drum to improve heat dissipation: a CFD approach,”*Research Journal of Engineering Sciences,* vol. 7, Issue 5, ISSN 2278 – 9472, pp. 1–6, May 2018.

[12] Bako Sunday, Usman Aminu, Paul O. Yahaya and Mohammed B. Ndaliman, “Development and analysis of finned brake drum model using solidworks simulation,” *International Journal of Innovative Research in Science, Engineering and Technology*, vol. 4, Issue 5, ISSN2319 - 8753, pp. 3651–3658, May 2015.

[13] Mohammed Abdul Aleem, Asrar Ahmed Saad, Abdul Sohail, Shaik Mohammed Siraj and M. Raghunath Reddy, “Design and analysis of drum brakes,” *International Journal of Advanced Technology and Innovative Research*, vol. 10, Issue 4, ISSN2348-2370, pp. 348–354, April 2018.