Research Journal of Engineering Sciences ISSN 2278 – 9472
Vol. **7(5),** 1-6, May **(2018)** *Res. J. Engineering Sci.*

Modeling and analysis of brake drum with extended fins on the circumference of drum to improve heat dissipation: a CFD approach

Yogeshwar Sinha[*] and Gurmeet Singh Gahir
Department of Mechanical Engineering, Chhattisgarh Swami Vivekanand Technical University, Bhilai, CG, India
yogeshwar.sinha07@gmail.com

Available online at: www.isca.in, www.isca.me
Received 17th March 2018, revised 15th May 2018, accepted 24th May 2018

Abstract

This work involves study of different factors responsible for the failure of brake in vehicles. Drum brake is more prevalent to the brake failure or brake fade in compare to the disc brake as in the case of later one heat is easily dissipated away from the rotors and the pads. But in the case of drum brake system due to its configuration the equipments are not exposed directly to the air because of which the heat convection process is very slow and less amount of heat is dissipated. So there is need to equipped the drum brake with such provision so that it will results in improvement of heat dissipation by convection. To increase the heat dissipation from the drum, design changes have been done. Basically, fins like features are added on the circumference of the drum, three different models with different number of fins have been analyzed. CFD analysis is performed by using ANSYS Fluent software to evaluate the result. Several simulations are carried out to find heat flux, temperature difference between solid fluid, heat transfer co-efficient and heat transfer rate for all three models. Results comes for different cases are then compared to understand the effect of design changes on the heat transfer rate, in this way we can obtain the model which is best suited for our purpose. Finally, we come to the conclusion that when we are increasing the numbers of fins of specific shape and size we can obtaining the positive results i.e. the heat transfer rate is increasing by providing fins on the circumference of the drum brake. Hence, the heat will be dissipated at much faster rate from modified drum model which will reduce the risk of brake fade or brake failure.

Keywords: Heat dissipation, CFD (computational fluid dynamics), FEM (finite element method), ANSYS, CREO parametric 2.0, drum brake, meshing.

Introduction

Basically, this work involves simulation of the heat convection process take place in Drum Brake System. Drum fade is one of the most common problems with the drum brakes. Drum fade usually occurs due to the high temperature generated during friction between shoe lining and the inner surface of the drum, as the system configuration is compact and completely closed unit very less heat convection takes place through air because of which sometimes the temperature of the drum increases to great extent which ultimately results in brake failure. To increase the heat dissipation from the drum we can increase the temperature difference between solid and fluid or increase the surface area. Most of the times, to control the temperature difference is not feasible and increase of heat transfer coefficient may require installation of a pump or a fan or replacing the existing one with a new one having higher capacity, the alternative is to increase the effective surface area by extended surfaces or fins. Fins are the extended surface protruding from a surface or body and they are meant for increasing the heat transfer rate between the surface and the surrounding fluid by increasing heat transfer area[1].

To increase the heat dissipation from the drum we have increased the surface area by providing fins over the circumferential surface of the drum[2]. We performed computational fluid dynamics (CFD) analysis to evaluate the heat transfer rate from the drum surface.

To simulate the process we have used ANSYS workbench software. The modeling is done in workbench module itself. CFD analysis is performed on fluent module of the ANSYS software. Several models with different number of fins have been created and simulated to compare their results. Basically average heat flux, heat transfer co-efficient, temperature difference between inlet-outlet and convective heat transfer rate of all the models have been evaluated and then compared with each other to obtain the best model. Basic model (case 1) result is compared and validated from mathematical calculation.

Methodology

Drum brakes, like most other brakes, convert kinetic energy into heat by friction. This heat should dissipate into the surrounding air, but due to conduction this heat can easily be transfer to other braking system components. Brake drums must be large to bear massive forces involved, and must be able to absorb and dissipate a lot of heat. Heat transfer to air can be aided by incorporating cooling fins onto the drum[2]. So the end aim of our project is to increase the heat transfer rate (Q) by varying the dimension and by increasing the numbers of fins.

So to fulfill this aim we have created three different models. Before starting the analysis process first of all we have done mathematical calculation for several required outputs. Flow chart for the methodology is given below:

According to our literature survey we found that drum brakes might get failed due to high temperature generated by applying brake, the term used to define failure of braking system is drum fade. Drum fade is more prevalent in drum brakes due to their configuration in comparison to disc brake as in the disc brake system heat can be easily vented away from the rotor and pad more easily. Moreover, when the drums are heated by hard braking, the diameter of the drum increases slightly due to thermal expansion, so the shoes must move further for that the driver must press the brake pedal further to apply brake. Due to lack of convection through air the temperature of the drum increases and the heat get transferred to the shoe by conduction. On overheating the properties of the friction material might get changed, which results in reduction of friction. This happens mostly with drum brakes in compare to the disc brakes, since the shoes are inside the drum and not exposed to cooling through ambient air. From the above mentioned problems occurs in braking system we have selected Brake fade problem for our work. Brake fade problem can be significantly reduced by appropriate equipment and materials design and selection, as well as good cooling[3].

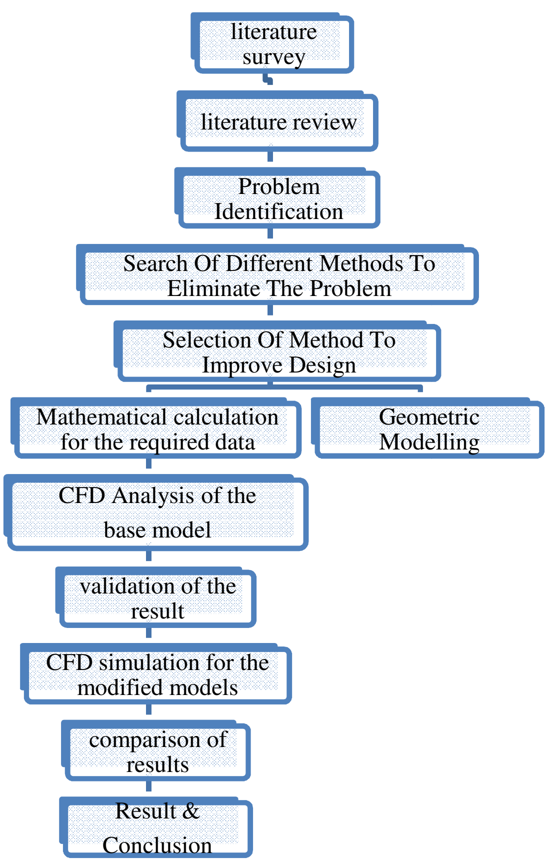

Figure-1: Flow chart of methodology.

So we decide to improve the heat dissipation of drum brake by improving its design.

To increase the heat convection from the drum we can approach by three methods. Convection heat transfer can be increased by either of the following ways: i. Increasing the temperature difference between the surface and the fluid. ii. Increasing the convection heat transfer coefficient by enhancing the fluid flow or flow velocity over the body. iii. Increasing the area of contact or exposure between the surface and the fluid. From these we have chosen the last one, i.e. increasing the area of contact or exposure between the surface and the fluid. To increase the surface area of contact we have added fins like structure on the periphery of the drum, hence in this way we can increase the heat convection. So to increase the heat dissipation we have created drums with fins on the circumferential surface to increase surface area.

Mathematical calculation for the required data: Primarily, we have focused to calculate Convective Heat Transfer Rate (Q) to compare between difference cases.

$Q = h*A*\Delta T$

Where: Q is the Convective Heat Transfer Rate (W), h is the heat transfer co-efficient $W/(m^2K)$, A is the surface area of the model m^2, ΔT is the temperature difference between solid and the fluid around the solid.

As we know our model is rotating with a given rpm as well as it is moving forward with a given speed, so the heat flux (q), temperature difference ΔT and the heat transfer coefficient (h) will varies from point to point. So we have to take the average values of all the required data. To calculate heat transfer co-efficient W/(m2K) (h) we have used the given formula:

$h = \frac{q}{\Delta T}$

All the dimensions of the base model were as shown in figure. In the second model we have just increased the surface area by increasing the number of fins with greater surface area. But according to the base paper the mass of brake drum should not be increased. So we have done a calculation. We measured the volume of base model in CREO parametric 2.0 i.e. 9939831 mm^3. After adding one fin of cross section $10x20mm^2$ the volume measured was 10235142 mm^3. It means the volume to be removed from inner surface of the drum is 10235142 – 9939831 = 295311 mm^2. Therefore, $V = \pi\ (R^2 - r^2).h$

Where: V is volume to be removed, R and r is the bigger and smaller radius of drum respectively in which R is to be calculated and r is 200mm, h is height of drum without rounded base i.e. 180mm.

$295311 = \pi\ (R^2 - 200^2).180$

R= 201.5mm

Hence the inner radius of the second model has been calculated. By following this process the volume and mass of the drum will be same as base model[2].

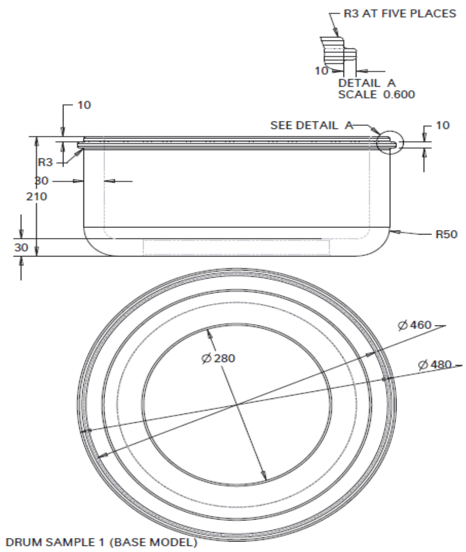

Figure-2: Case 1 Drawing with Dimensions.

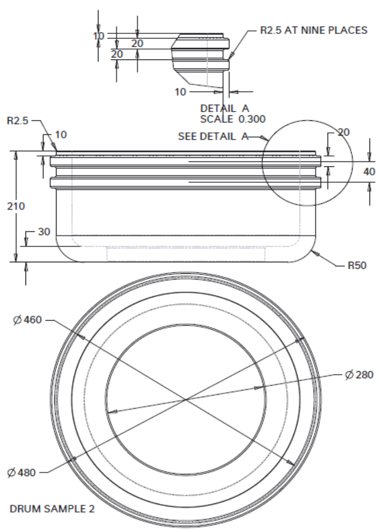

Figure-3: Case 2 Drawing with Dimensions.

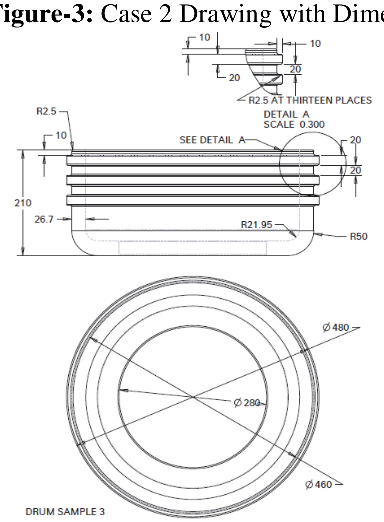

Figure-4: Case 3 Drawing with Dimensions.

In third model, we have added three fins. By applying the same calculation we have got the value of radius as 203.3mm.

Importing the model in to ANSYS Fluent: Models created in CREO Parametric 2.0 is saved in IGES format, so it can import in ANSYS fluent for analysis purpose.

Modeling (Creating Fluid and Solid Domains): As we know that this is a drum brake of a vehicle. When we apply brake the shoe lining makes contact with the inner wall of the drum and it tends to stop the rotating drum, during this process heat generation takes place due to which the temperature of the drum increases, but when the drum moves through the air much of the heat gets dissipated which results in reduction of drum temperature. So first of all in ANSYS we have to create a fluid domain (air) around the solid domain (drum). In our case we have selected a rectangular domain for the fluid, so that we consider two opposite faces as INLET and OULET and rest of the surfaces as side wall.

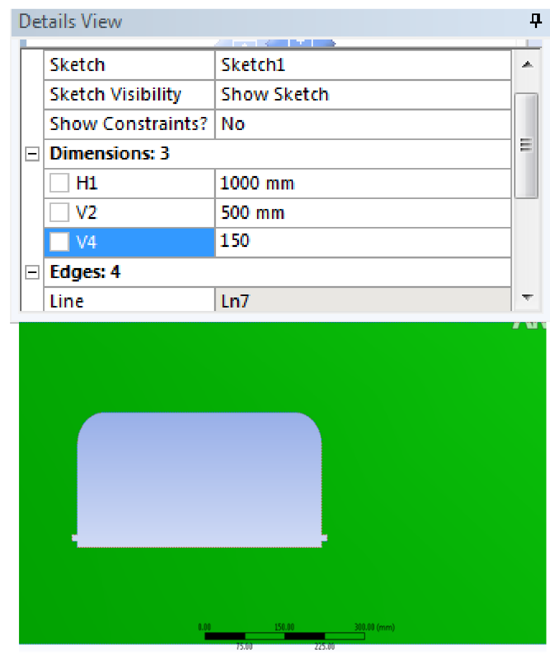

Figure-5: Modeling of fluid and solid domains.

Meshing: After creating domain we have invoked in to the mesh menu were we have to set the size of and type of meshing required for the domains.

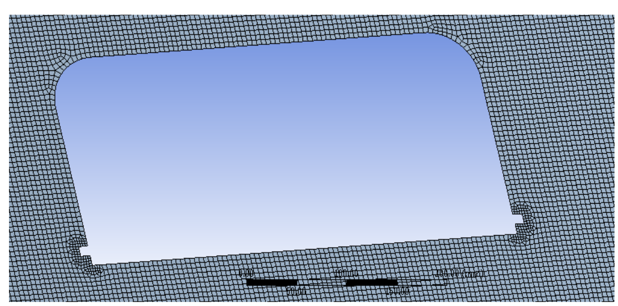

Figure-6: Structured meshing of the domain.

SETUP: Now, in setup mode we have to turned on the energy equation option so that the energy can be taken into account during simulation (heat and kinetic energy).

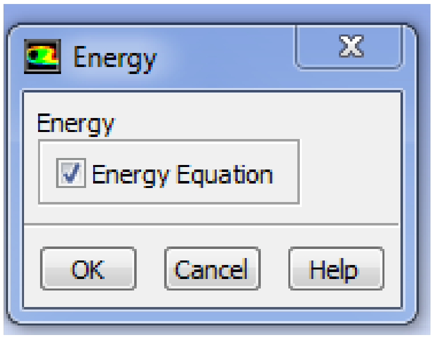

Figure-7: Equation Check Box Option.

For our model we have selected k-epsilon (2 equation) turbulent model to perform analysis and rest of the setting is remain as standard.

Material Selection: For fluid domain we have selected air as incompressible gas and for solid domain we have selected aluminium material with default properties.

Fluid properties: i. Density→ incompressible-ideal–gas, ii. Cp (Specific heat) (j/kg-k)→1006.43, iii. Thermal conductivity (w/m-k)→ 0.0242, iv. Viscosity (kg/m-s)→ 1.789 E-05.

Solid properties: i. Density (kg/m3)→ 2719, ii. Cp (Specific heat) (j/kg-k)→ 871, iii. Thermal conductivity(w/m-k)→ 202.4
.

Boundary conditions given: Inlet wall →: i. Velocity inlet is provided at inlet location, ii. Velocity Magnitude is 12m/s, iii. Temperature at inlet is 300^0K (ambient temperature).

(Actually by setting the air velocity we assume that the vehicle is moving with the given velocity)

Top and bottom wall →: i. Velocity Magnitude given is 12 m/s.

This define that the surrounding is moving in the negative direction of the vehicle.

Outlet wall→ : i. Gauge pressure (pascal) is set to be 0(zero) so that it will act as outlet and the air will flow towards it. ii. Temperature at oulet is 300^0K (ambient temperature).

Drum wall →Rotation is provided about the x axis to 32.4 (rad/sec): i. Temperature of wall is set to 410^0K. The temperature developed on applying brake due to the friction occurs b/w shoe lining and drum brake is calculated to be 410^0K.

Solution initialization and run calculation: Hybrid initialization is selected for simulation purpose. Simulation is started with 650 number of iteration for the first case, 1000 iterations for second case and 1200 iteration for third case to get the converged result.

Results and discussion

On simulation we get results of heat flux (q), heat transfer co-efficient (h), Temperature difference (ΔT) and these values are used to calculate convective heat transfer rate (Q) which is later on compared for each cases.

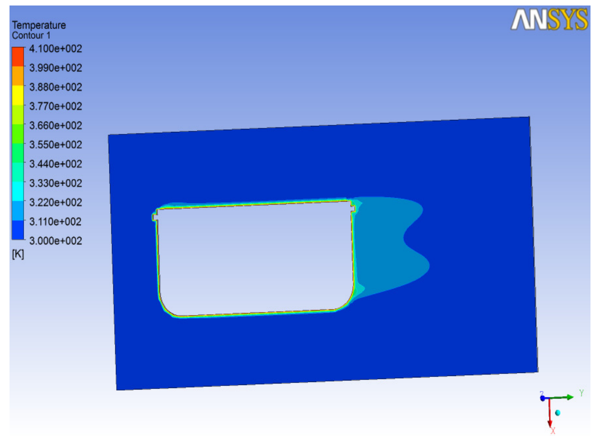

Figure-8: Temperature contour of Case 1.

Case-1: Wall temperature, wall heat flux, fluid temperature adjacent to the wall and heat transfer coefficient value for different co-ordinates are evaluated by the software, by using that values we have calculated the following values :-
Average heat transfer co-efficient (h) = 1.79E+02 W/(m2K)
Average ΔT = 6.14E+01 K
Surface Area of drum sample 1 = 0.74 m^2
Therefore,
Q= h*A* ΔT = 1.79E+02 * 0.74* 6.14E+01
=> Q = 8.12E+03

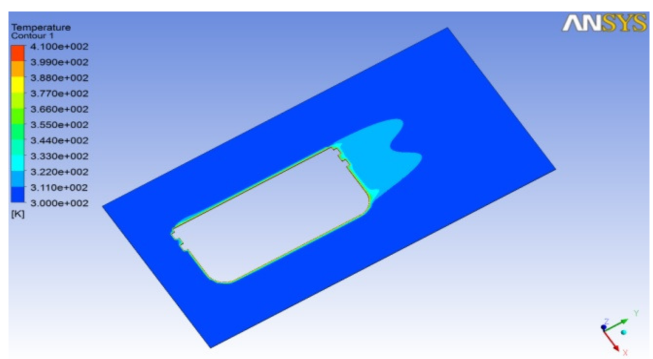

Figure-9: Temperature contour of Case 2.

Case-2:Averageheattransferco-efficient (h)=1.63E+02 W/(m2K)
Average ΔT = 6.51E+01 K
Surface Area of drum sample 1 = 0.77 m2
Therefore,
Q= h*A* ΔT = 1.63E+02 * 0.77 * 6.51E+01
=> Q = 8.19E+03

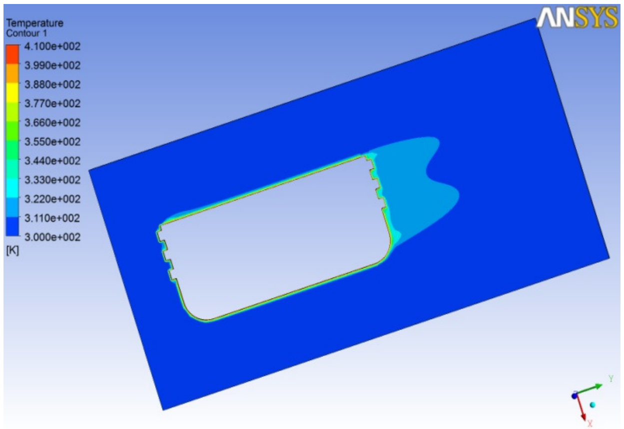

Figure-10: Temperature contour of Case 3.

Case3:- Average heat transfer co-efficient (h) = 1.53E+02 W/(m^{2}K)
Average ΔT = 6.83E+01 K
Surface area of drum sample 1 = 0.80 m^2
Therefore,
Q= h*A* ΔT = 1.53E+02 * 0.80 * 6.83E+01
=> Q = 8.34E+03

Table-1: Average HTC(h), Surface area of models, average temp. Diff., Heat transfer.

Models	Avg. HTC (*h*) W/m^{2}K	Surface area (*A*) m^2	Avg. tem. Difference (Δ*T*) K	Heat transfer (*Q*) W
Model 1	1.79E+02	0.74	6.14E+01	8.12E+03
Model 2	1.63E+02	0.77	6.51E+01	8.19E+03
Model 3	1.53E+02	0.80	6.83E+01	8.34E+03

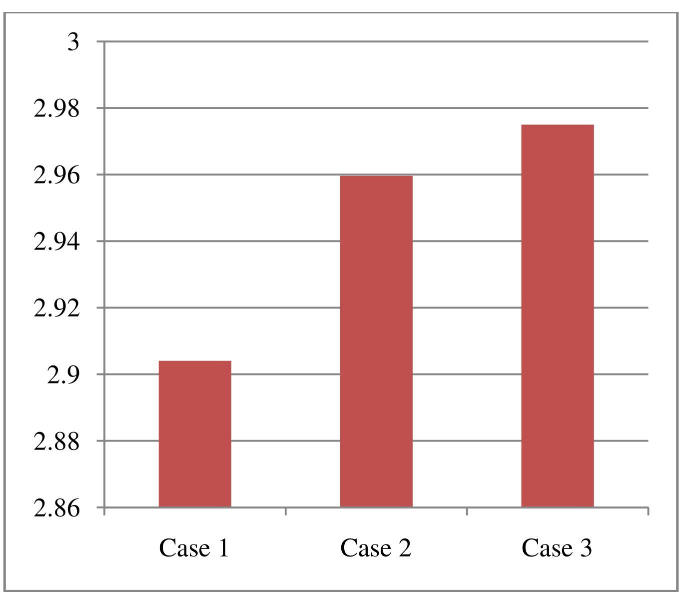

Figure-11: Average heat transfer Co-efficient.

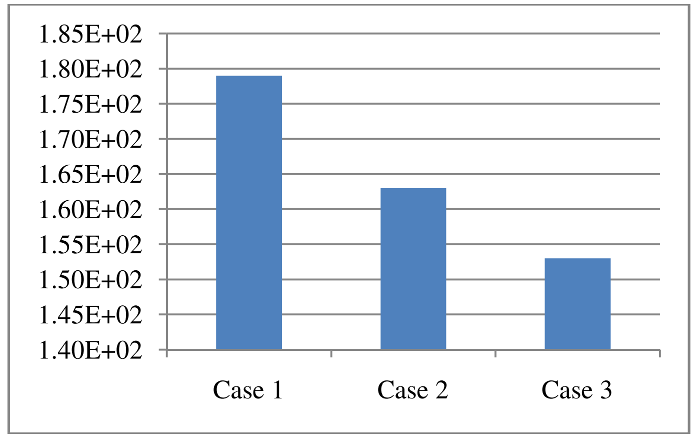

Figure-12: Outlet to Inlet temperature difference.

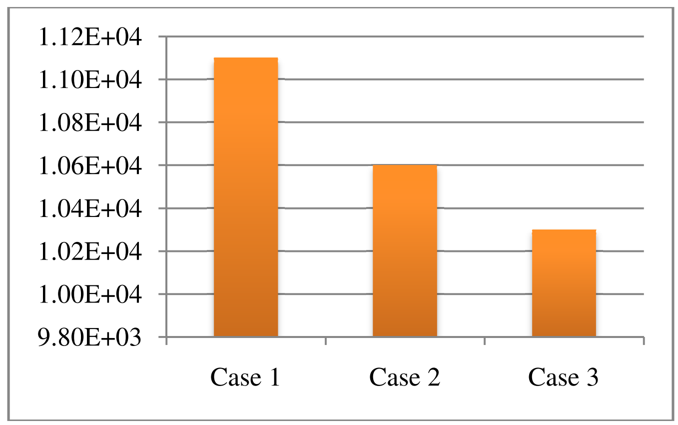

Figure-13: Average heat flux.

Figure-14: Convective Heat Transfer Rate.

Conclusion

By interpretation of the above result we can conclude that on increasing the surface area by providing projections on the external surface of the drum brake we can increase the heat dissipation by convection as we can see the area average temperature difference from outlet to the inlet is highest for the Case 3 which has more surface area. And the area average temperature difference decreased for 2nd and 1st case as their surface area is less.

Higher temperature difference shows that more heat is transferred and dissipated from the brake drum surface to the air by convection.

Secondly, Q (convective heat transfer rate) is highest in third case which shows us that by providing projection on the drum surface we can increase the heat transfer rate from drum to the air in respect time. That means the drum will be cooled faster in Case 3. This analysis can assist Automotive Engineers to design a more effective and reliable brake drums.

References

1. Lee S.R., Li Z.G., Wang B.G. and Chiou H.S. (2005). An Application of the Fractal Theory in the Design of Heat Sink for Precision Measurement Instrument. *Key Engineering Materials*, 295, 717-722.

2. Sunday B., Aminu U., Yahaya P.O. and Ndaliman M.B. (2015). Development and Analysis of Finned Brake Drum Model Using Solidworks Simulation. *International Journal of Innovative Research in Science, Engineering & Technology*, 4(5), 3651-3658.

3. Watson C. and Newcomb T.P. (1991). Prediction of Temperature Distributions in a Drum Brake Rubbing Path. *Elsevier-Tribology Series*, 18, 175-182.

4. Raj K.T.R., Ramsai R., Mathew J. and Soniya G. (2014). Numerical investigation of fluid flow and heat transfer characteristics on the aerodynamics of ventilated disc brake rotor using CFD. *Thermal Science*, 18(2), 667-675.

5. Limpert R. (1975). The Thermal Performance of Automotive Disc Brakes. *Journal of SAE*, 750873(1), 4-7.

6. Narang G.B.S. (1977). Automobile Engineering. First Edition. Kahanna Publisher, India, 275-277. ISBN:13: 1234567144518

7. How Drum Brakes Works (2015). Available: http://www.howstuffworks.com/autoparts/brake/brake-types/drum-brake1.html, accessed March 2015.

8. Kicreyco Brake Drum Catalog. Kic Holding Inc (2015). Available at www.kicgroup.com, accessed April 2015.

9. Hohmann C., Schiffner K., Oerter K. and Reese H. (1999). Contact analysis for drum brakes and disk brakes using ADINA. *Computers & structures*, 72(1-3), 185-198.

10. Rosario Lanzafame, Stefano Mauro and Michele Messina (2014). 2D CFD Modeling of H-Darrieus Wind Turbines using a Transition Turbulence Model. *Elsevier-Energy Procedia*, 45, 131-140.

11. Versteeg H.K. and Malalasekera W. (1995). An Introduction to Computational Fluid Dynamics. The Finite Element Method, 134-176. ISBN 0- 470-23515-2.