DESIGN AND ANALYSIS OF DRUM BRAKE

Dr. Kodathalapalli Sudheer [1]

[1] MS (CAD/CAM),Ass. Prof.,Dept of MECH, Farah Institute of Engineering and Technology,

Chevella, Rangareddy, Telangana, India. Pin: 501503

Mail Id: kodathalapalli@rediffmail.com

Abstract

A brake is a mechanical device which inhibits motion. A drum brake is a brake that uses friction caused by a set of shoes or pads that press against a rotating drum-shaped part called a brake drum.

The brake drum is a critical component that experiences high temperatures and develop thermal stresses during application of brakes. In addition, the application of shoe pressure gives rise to mechanical loads. So the analysis takes into account both the thermal stresses and mechanical stresses together.

Brakes in cars and trucks are safety parts. Requirements not only in performance but also in comfort, serviceability and working lifetime are high and rising. i.e. the brake pad with the friction material, the counter body and caliper, can be modeled.

So in this project we design the model of drum brake (drum, liners, springs etc.,) And perform the structural and thermal analysis in solid works premium 2016.

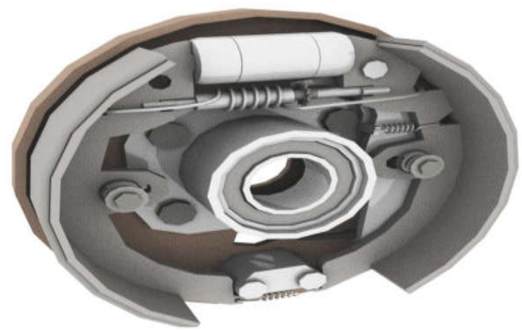


Fig: Drum brake

1. Introduction:

A brake is a device which is used to bring to rest or slow down a moving body. Safe operation of vehicle demands dependable brakes is required to absorb the kinetic energy of the moving parts or the potential energy of the object being lowered by host when the rate of descent is controlled. The energy absorbed by brakes is dissipated in the form of heat. This heat is dissipated in the surrounding atmosphere to stop the vehicle, so the brake system should have following requirements:

- The brakes must be strong enough to stop the vehicle with in a minimum distance in an emergency.
- The driver must have proper control over the vehicle during braking and vehicle must not skid.
- The brakes must have well anti fade characteristics i.e. their effectiveness should not decrease with is constant prolonged application.

- The brakes should have well anti wear properties.
- The important requirements of the brake drum are following:
- It should provide a surface having well anti wear qualities.
- It should allow the optimum rate of heat transfer.
- Heat is generated during each brake application and it must be dissipated to the atmosphere immediately, because the next brake application would again produce more heat. Any excess heating of brakes would cause the drum to expand resulting in loss of effective pedal travel and fading of brake lining.
- It should have sufficient strength but minimum weight.
- It should be able to be accommodated within the wheel space available.

LITERATURE SURVEY

C. H. Neeraja a C. R. Sireesha and D. Jawaharlal [1] have modelled a suspension frame used in two-wheeler. Modelling is done in Pro/Engineer. They have done structural and modal analysis on suspension frame using four materials Steel, Aluminium Alloy A360, Magnesium and carbon fiber reinforced polymer to validate our design. By observing the results, for all the materials the stress values are less than their respective permissible yield stress values. So the design was safe, by conclusion. By comparing the results for four materials, stress obtained is same and displacement is less for carbon fiber reinforced polymer than other three materials. So for design considered, CFRP is better material for suspension frame.

Cicek Karaoglu and N. Sefa Kuralay [2] did the finite element analysis of a truck chassis. The analysis showed that INTERNATIONAL JOURNAL FOR RESEARCH IN EMERGING SCIENCE AND TECHNOLOGY, VOLUME-2, ISSUE-1, JANUARY-2015 E-ISSN: 2349-7610 VOLUME-2, ISSUE-1, JANUARY-2015 COPYRIGHT © 2015 IJREST, ALL RIGHT RESERVED 43 increasing the side member thickness can reduce stresses on the joint areas, but it is important to realize that the overall weight of the chassis frame increases. Using local plates only in the joint area can also increase side member thickness. Therefore, excessive weight of the chassis frame is prevented. In November 2008 Mohamad Tarmizi Bin Arbain uses 3D model for finite element analysis issues regarding the experimental analysis of car chassis is addressed. The modeling approach is investigated extensively using both of computational and compared it to experimental modal analysis. A comparison of the modal parameters from both experiment and simulation shows the validity of the proposed approach. Then perform the computational stress analysis with linear material type analysis to find the stress concentration point in the car chassis.

Karaoglu and Kuralay[3] investigated stress analysis of a truck chassis with riveted joints using FEM. Numerical results showed that stresses on the side member can be reduced by increasing the side member thickness locally.

Fermer et al investigated the fatigue life of Volvo S80 Bi-Fuel using MSC/Fatigue Conle and Chu [4] did research about fatigue analysis and the local stress –strain approach in complex vehicular structures. Structural optimization of automotive components applied to durability problems has been investigated by Ferreira et al

Filho Et. al. [5] have investigated and optimized a chassis design for an off road vehicle with the appropriate dynamic and structural behavior.

2. TYPES OF BRAKES

2.1 basis of method of actuation

(a) Foot brake

(b) Hand brake

2.2 basis of mode of operation

(a) Mechanical breaks

(b) Hydraulic brakes

(c) Air brakes

(d) Vacuum brakes

(e) Electric brakes.

2.3 basis of action on front or rear wheels

(a) Front-wheel breaks

(b) Rear-wheel brakes.

2.4 basis of method of braking contact

(a) Internally – expanding brakes

(b) Externally – contracting brakes.

3. Drum brakes

Drum brake is one of the most commonly used brakes in vehicle design; it can be categorized into leading- and trailing-shoe brake, two-leading-shoe brake, two-trailing-shoe brake, and servo brake concerning the arrangement of the brake shoes. The optimization design object in this paper is the leading- and trailing-shoe brake, which is shown in Figure

Below Figure shows the structural parameters and force diagram of the leading- and trailing-drum brake only left part of the structure is presented. With the effect of braking force P, the two brake shoes at both sides rub the drum brake to generate frictional resisting torque and thus to brake. In Figure 1, $F1$ is the pressing force of the brake shoe, while Rf is the action radius of the frictional force. δ represents the angle between x axis and the active line of $F1$. The main structural parameters are the starting angle of brake friction plate $\theta 0$, wrap angle of friction plate θ, the distance from braking force P to the brake center a, the central location of the brake shoe pin c, c, and so forth.

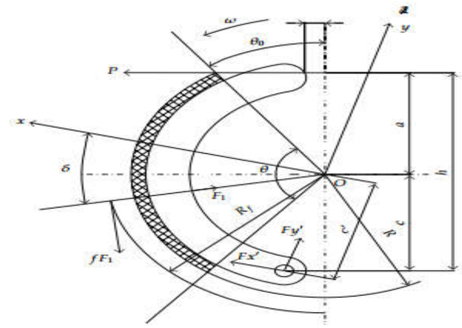


Fig: dimensions of shoe and liners

3.1 Components of drum brakes :

- Backing plate:
- Brake drum:
- Brake shoe:

4. DESIGN OF DRUM BRAKES

Rear drum brakes are typically of a leading design (for non-servo systems), or primary/secondary (for duo servo systems) the shoes being moved by a single double-acting hydraulic brakes and hinged at the same point. In this design, one of the brake shoes always experiences the self-applying effect, irrespective of whether the vehicle is moving forwards or backwards. This is particularly useful on the rear brakes, where the parking brake (handbrake or footbrake) must exert enough force to stop the vehicle from travelling backwards and hold it on a slope. Provided the contact area of the brake shoes is large enough, which isn't always the case; the self-applying effect can securely hold a vehicle when the weight is transferred to the rear brakes due to the incline of a slope or the reverse direction of motion. A further advantage of using a single hydraulic cylinder on the rear is that the opposite pivot may be made in the form of a double-lobed cam that is rotated by the action of the parking brake system.

Front drum brakes may be of either design in practice, but the twin braking design is more effective. This design uses two actuating cylinders arranged so that both shoes use the self-applying characteristic when the vehicle is moving forwards. The brake shoes pivot at opposite points to each other. This gives the maximum possible braking when moving forwards, but is not so effective when the vehicle is traveling in reverse.

The optimum arrangement of twin braking front brakes with leading brakes on the rear allows more braking force at the front of the vehicle when it is moving forwards, with less at the rear. This helps prevent the rear wheels from locking up, but still provides adequate braking at the rear.

The brake drum itself is frequently made of cast iron, though some vehicles have used aluminum drums, particularly for front-wheel applications. Aluminum conducts heat better than cast iron, which improves heat dissipation and reduces fade. Aluminum drums are also lighter than iron drums, which reduces un spring weight. Because aluminum wears more easily than iron, aluminum drums frequently have an iron or steel liner on the inner surface of the drum, bonded or riveted to the aluminum outer shell.

4.1 MATERIAL SELECTION FOR DRUM BRAKES

Traditional material for automotive brake rotor is the cast iron. The specific gravity or density of cast iron is higher which consumes much fuel due to high inertia. Following section will describe the potential candidate materials those can be used for brake rotor application.

Based on the properties, potential candidate materials for automotive brake disc were selected as :

- Gray cast iron (GCI)
- Ti-alloy (Ti-6Al-4V)
- 7.5 wt% WC and 7.5 wt% TiC reinforced
- Ti-composite (TMC) 20% SiC reinforced Al-composite (AMC 1)
- 20% SiC reinforced Al-Cu alloy (AMC 2)

5. STRUCTURE AND FUNCTIONING OF DRUM BRAKES :

Performing detailed analysis of the causes and failure modes of the observed object requires knowledge of the structure, functioning modes and the relationship among the constituent elements. Only a full understanding of the functioning of the system and its elements, as well as knowledge of their mutual relationships, allows the implementation of logical analysis that defines the necessary and sufficient conditions for the appearance of the object's failure.

Drum brake is composed of mobile and immobile elements. Immobile elements are via backing plate (1) attached to the supporting structure of the vehicle, while moving parts (drum (2)) are connected to wheel hub. Friction elements of drum brakes are two symmetrically placed brake shoes and drum. During brake activation, brake shoes snuggle up with the drum and thus the car's kinetic energy is converted into heat, i.e. braking of the vehicle is performed. Drum brakes' shoes are composed of a metal carrier part (3) and friction lining (4). The connection between the metal part and friction lining can be achieved by riveting

Main parts of rear drum brakes: 1 – Baking plate, 2 - Drum, 3 – Brake shoe, 4 – Shoe lining, 5 - Rivet, 6 – Brake adjuster, 7 - Elements for holding the shoes, 8 - Shorter return spring, 9 - Longer return spring, 10 - Lever mechanism of the parking brake 11 – The return spring.

Drum brakes can be activated hydraulically or mechanically. If the hydraulic transmission mechanism is used for the service brake on motor vehicles, activation of shoes is performed by hydraulic cylinders. Braking torque is proportional to the activation force of the brake, i.e. the operation pressure and to the diameter of the brake cylinder. The brake cylinder is a screw-connected to the baking plate (1), which is usually made in sheet metal forming processes with a relatively strong relief, resulting in a higher stiffness of the element. Rear brakes of the vehicles represent the executive mechanisms of the service and parking brake of the vehicle. Activation of shoes for the parking brake is done via the lever mechanism

6. FAULT TREE OF DRUM BRAKES

Forming of the fault tree is done by using the symbols for events, logic gates and transmission. A number of different symbols are used for events that indicate whether it is a complex or basic initiating events. The rectangle is used for complex events. Among the symbols used for basic events most commonly used is the circuit, which signifies the state of an element of the system conditioned by its characteristics, and rhomb, which indicates an undeveloped event. Logical symbols in the fault tree signify mutual conditionality and correlation of lower and higher levels events. For example, the "OR" logic gate produces output if one or more input events are happened. In contrast, "AND" logic gate produces output only if all input events occur.

The peak event in the fault tree, "failure of the drum brake", implies a complete or partial brake's failures. The manner of defining the peak event enables recording of the highest number of potential failure modes of drum brake's elements. Total failures of drum brake occur when the braking torque cannot be achieved on the brake, and they rarely happen. In the case of partial failures, the operating characteristics of brakes are significantly deteriorated. This manifests through the reduction of achieved braking torque on the brake, uneven braking or delayed response of brakes. Failures that lead to reduction of the braking torque are usually called friction failure. They can be permanent and transient. Permanent friction failures could occur due to wear of the friction surfaces of drums and linings, inhomogeneity of the lining material (presence of hard inclusions), occurrence of cracks and falling out of lining's pieces, dirty or greased friction surfaces etc.

SOLIDWORKS:

Solid Works is mechanical design automation software that takes advantage of the familiar Microsoft Windows graphical user interface.

It is an easy-to-learn tool which makes it possible for mechanical designers to quickly sketch ideas, experiment with features and dimensions, and produce models and detailed drawings.

A Solid Works model consists of parts, assemblies, and drawings.

- Typically, we begin with a sketch, create a base feature, and then add more features to the model. (One can also begin with an imported surface or solid geometry).
- We are free to refine our design by adding, changing, or reordering features.
- Associatively between parts, assemblies, and drawings assures that changes made to one view are automatically made to all other views.
- We can generate drawings or assemblies at any time in the design process.
- The Solid Works software lets us customize functionality to suit our needs.

INTRODUCTION TO SOLIDWORKS:

Solidworks mechanical design automation software is a feature-based, parametric solid modeling design tool which advantage of the easy to learn windows ™

graphical user interface. We can create fully associate 3-D solid models with or without while utilizing automatic or user defined relations to capture design intent.

Parameters refer to constraints whose values determine the shape or geometry of the model or assembly. Parameters can be either numeric parameters, such as line lengths or circle diameters, or geometric parameters, such as tangent, parallel, concentric, horizontal or vertical, etc. Numeric parameters can be associated with each other through the use of relations, which allow them to capture design intent.

Design intent is how the creator of the part wants it to respond to changes and updates. For example, you would want the hole at the top of a beverage can to stay at the top surface, regardless of the height or size of the can. Solid Works allows you to specify that the hole is a feature on the top surface, and will then honor your design intent no matter what the height you later gave to the can. Several factors contribute to how we capture design intent are Automatic relations, Equations, added relations and dimensioning.

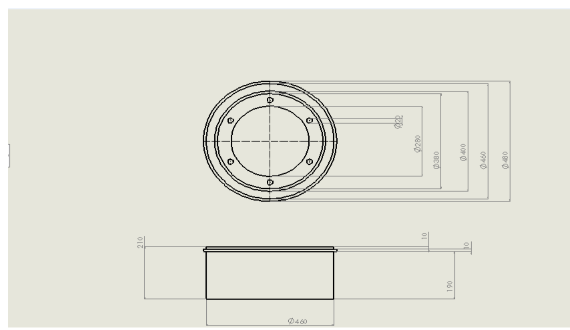
Features refer to the building blocks of the part. They are the shapes and operations that construct the part. Shape-based features typically begin with a 2D or 3D sketch of shapes such as bosses, holes, slots, etc. This shape is then extruded or cut to add or remove material from the part. Operation-based features are not sketch-based, and include features such as fillets, chamfers, shells, applying draft to the faces of a part, etc.

Building a model in Solid Works usually starts with a 2D sketch (although 3D sketches are available for power users). The sketch consists of geometry such as points, lines, arcs, conics (except the hyperbola), and spines. Dimensions are added to the sketch to define the size and location of the geometry. Relations are used to define attributes such as tangency, parallelism, perpendicularity, and concentricity. The parametric nature of Solid Works means that the dimensions and relations drive the geometry, not the other way around. The dimensions in the sketch can be controlled independently, or by relationships to other parameters inside or outside of the sketch.

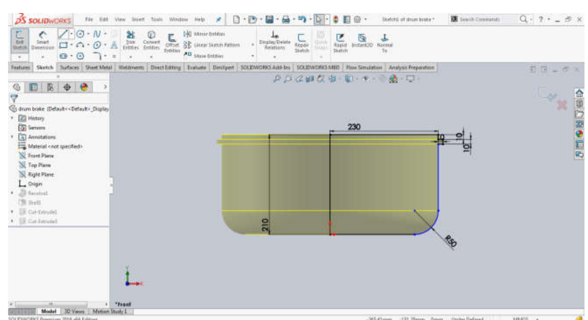
Design procedure of drum brakes

For designing the drum brakes, the following procedure has to be follow

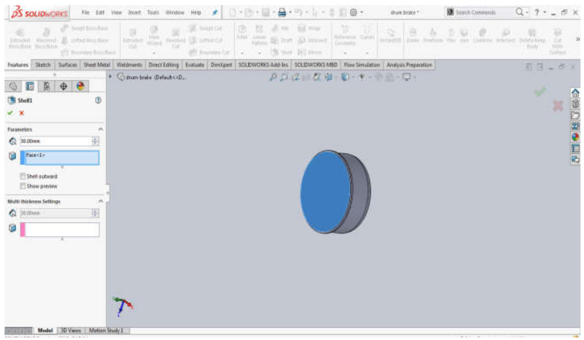
- 2 dimensional sketch of a drum brake



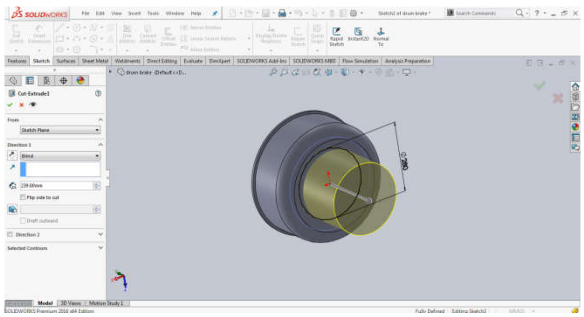
By using Revolve boss tool



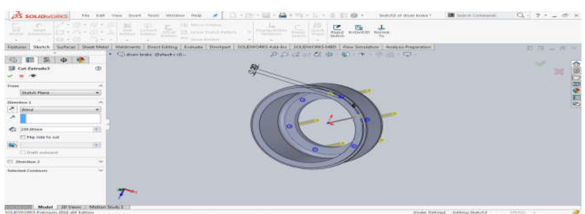
Make shell for the following face of 30mm



Draw a sketch as follows and make extrude cut



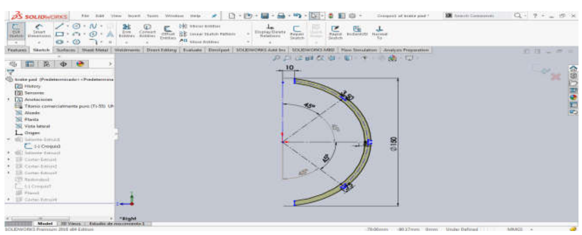
Now draw a circle 20mm diameter and make circular pattern of 6. And make extrude cut in features



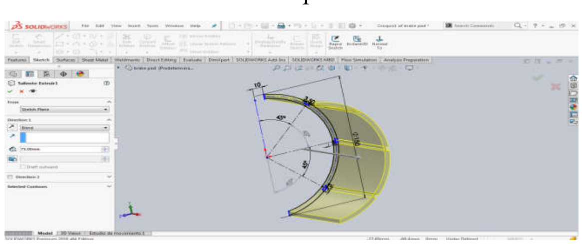
Thus the modeling of drum brake is done

Now modeling of drum brake pad

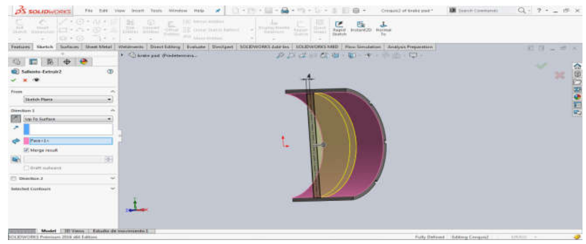
Brake shoe lining



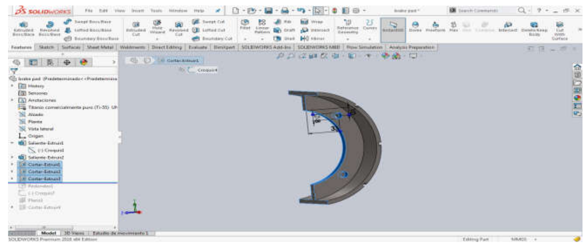
Go to features and extrude upto 75mm

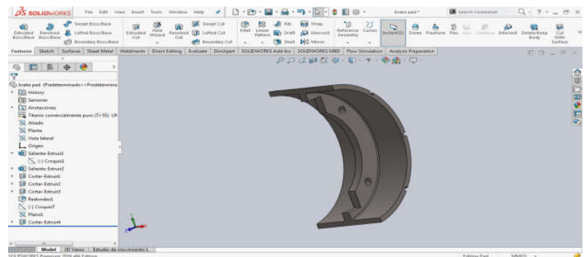


Draw the sketch and extrude it

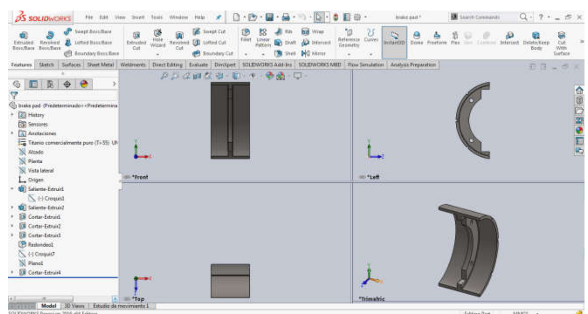


Make individual sketches and make extrude cut





Four views of drum brake pad



Finite Element Analysis

Introduction

Finite Element Analysis (FEA) is a computer-based numerical technique for calculating the strength and behavior of engineering structures. It can be used to calculate deflection, stress, vibration, buckling behavior and many other phenomena. It also can be used to analyze either small or large scale deflection under loading or applied displacement. It uses a numerical technique called the finite element method (FEM). In finite element method, the actual continuum is represented by the finite elements. These elements are considered to be joined at specified joints called nodes or nodal points. As the actual variation of the field variable (like displacement, temperature and pressure or velocity) inside the continuum is not known, the variation of the field variable inside a finite element is approximated by a simple function. The approximating functions are also called as interpolation models and are defined in terms of field variable at the nodes. When the equilibrium equations for the whole continuum are known, the unknowns will be the nodal values of the field variable.

Introduction to Simulation

Simulation is a design analysis system. Simulation provides simulation solutions for linear and nonlinear static, frequency, buckling, thermal, fatigue, pressure vessel, drop test, linear and nonlinear dynamic, and optimization analyses.

Powered by fast and accurate solvers, Simulation enables you to solve large problems intuitively while you design. Simulation comes in two bundles: Simulation Professional and Simulation Premium to satisfy your analysis needs. Simulation shortens time to market by saving time and effort in searching for the optimum design.

Figure: simulation example

Benefits of Simulation:

After building your model, you need to make sure that it performs efficiently in the field. In the absence of analysis tools, this task can only be answered by performing expensive and time-consuming product development cycles. A product development cycle typically includes the following steps:

1. Building your model.
2. Building a prototype of the design.
3. Testing the prototype in the field.
4. Evaluating the results of the field tests.
5. Modifying the design based on the field test results.

This process continues until a satisfactory solution is reached. Analysis can help you accomplish the following tasks:

- Reduce cost by simulating the testing of your model on the computer instead of expensive field tests.
- Reduce time to market by reducing the number of product development cycles.
- Improve products by quickly testing many concepts and scenarios before making a final decision, giving you more time to think of new designs.

Ansys

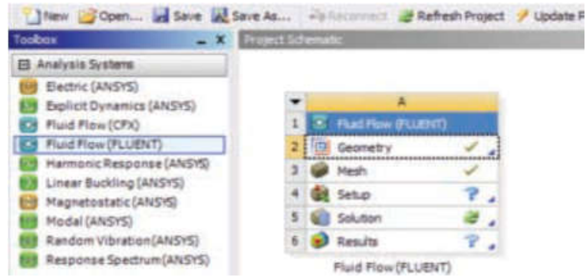
Introductions to Ansys

ANSYS 14.5 delivers innovative, dramatic simulation technology advances in every major Physics discipline, along with improvements in computing speed and enhancements to enabling technologies such as geometry handling, meshing and post-processing. These advancements alone represent a major step ahead on the path forward in Simulation Driven Product Development. But ANSYS has reached even further by delivering all this technology in an innovative simulation framework, ANSYS Workbench14.5The ANSYS Workbench environment is the glue that binds the simulation process; this has not changed with version.14.5 In the original ANSYS Workbench, the user interacted with the analysis as a whole using

The platform's project page: launching the various applications and tracking the resulting files employed in the process of creating an analysis. Tight integration between the component applications yielded unprecedented ease of use for setup and solution of even complex multi physics simulations.

In ANSYS 14.5, while the core applications may seem familiar, they are bound together via the innovative project page that introduces the concept of the project. This expands on the project page concept. Rather than offer a simple list of files, the project schematic presents a comprehensive view of the entire analysis project in flowchart form in which explicit data relationships are readily apparent. Building and interacting with these flowcharts is straightforward. A toolbox contains a selection of systems that form the building blocks of the project. To perform a typical simulation, such as static structural analysis, the user locates the appropriate analysis system in the toolbox and, using drag and-drop, introduces it into the project schematic. That individual system consists of multiple cells, each of which represents a particular phase or step in the analysis. Working through the system from the top down, the user completes the analysis, starting with a parametric connection to the original CAD geometry and continuing through to post-processing of the analysis result. As each step is completed, progress is shown clearly at the project level. (A green check mark in a cell indicates that an analysis step has been completed.)

Work bench:



The ANSYS Workbench environment tracks dependencies among the various types of data in the project. If something changes in an upstream cell, the project schematic shows that downstream cells need to be updated to reflect these changes. A project level update mechanism allows these changes to be propagated through all dependent cells and downstream systems in batch mode, dramatically reducing the effort required to repeat variations on a previously completed analysis. Parameters are managed at the project level, where it is possible to change CAD and geometry parameters, material properties and boundary condition values. Multiple parametric cases can be defined in advance and managed as a set of design points, summarized in tabular form on the ANSYS Workbench project page. Design Exploration systems can be connected to these same project- level parameters to drive automated design investigations, such as Design of Experiments, goal-driven optimization or Design for Six Sigma.

Analysis Types

The different type of analysis that can be performed in ANSYS

Structural static analysis:

1. Structural dynamic analysis
2. Structural buckling analysis
   - Linear buckling
   - Non linear buckling
3. Structural non linearity

4. Static and dynamic kinematics analysis
5. Thermal analysis
6. Electromagnetic field analysis
7. Electric field analysis
8. Fluid flow analysis
   - Computational fluid dynamics
   - Pipe flow
9. Coupled-field analysis

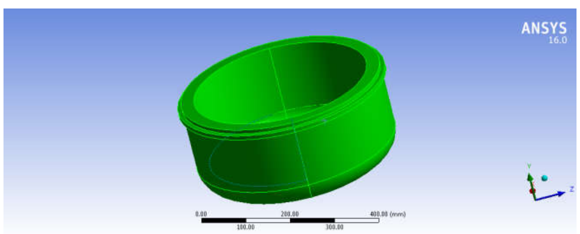
Materials Used In Project and There Properties:

material	Density (kg/m^3)	Young's modulus (MPa)	Poisons ratio	Thermal conductivity(w/mm^0c)
Aluminum oxide	3720	3e005	0.21	2.5e-002
AlSiC	2880	1.15e011	0.27	0.2256
Aluminum alloy	2770	7.1e003	0.33	0.175
Grey cast iron	7200	1.1e005	0.28	5.2e-002

11.1 ANALYSIS ON DRUM BRAKE:

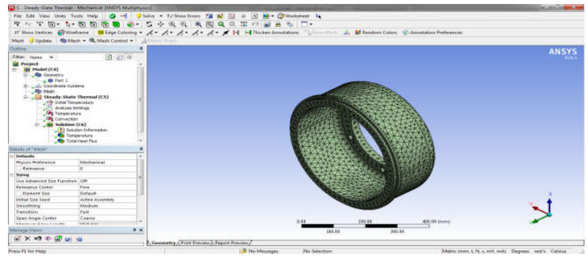
11.1.1 Structural

Model

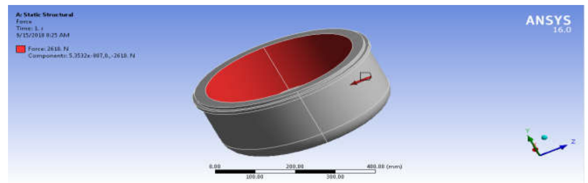


Meshing

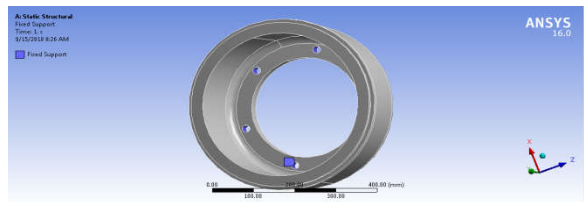
Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Fine
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	10.0 mm



Force applied: 2610N

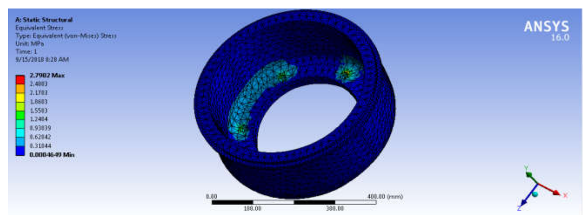


Fixed support:

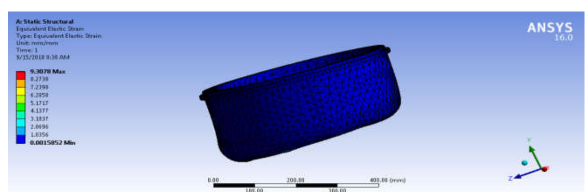


For Aluminum oxide

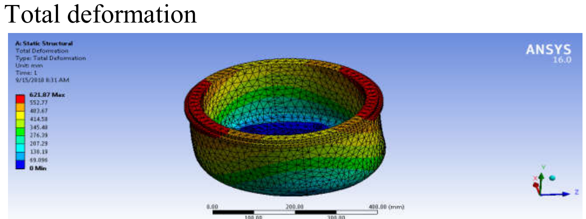
Stress



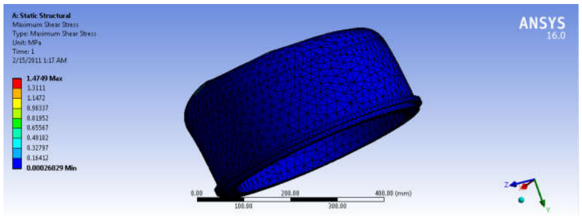
Strain



Total deformation

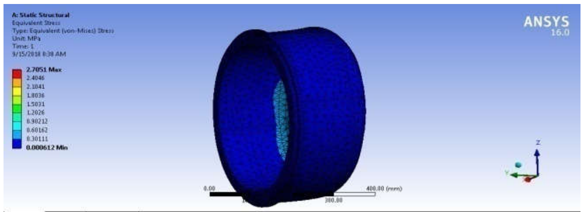


Max Shear stress

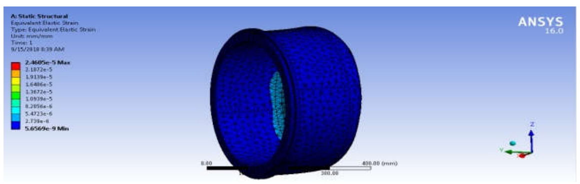


For Grey cast iron

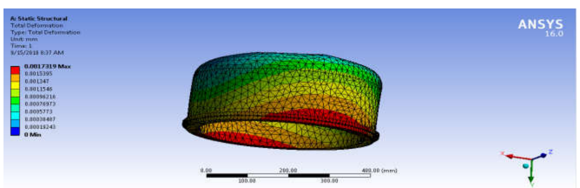
Stress



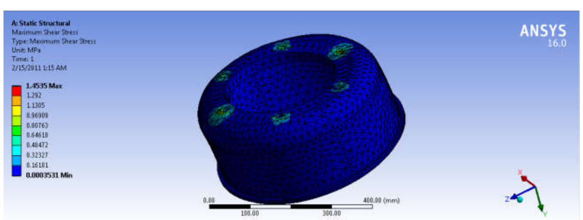
Strain



Total deformation

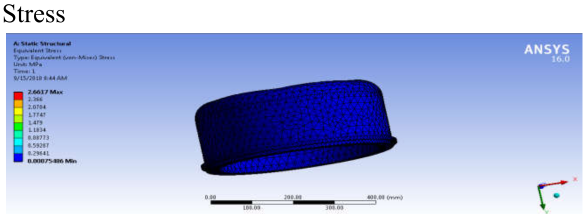


Max Shear stress

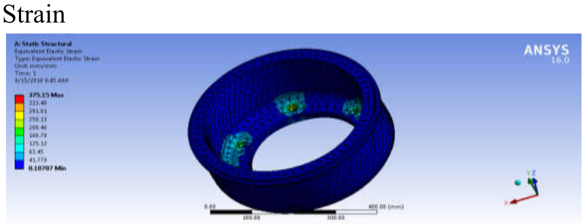


For Aluminum alloy

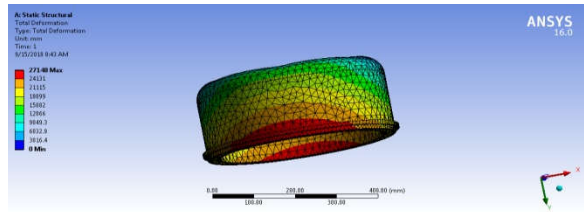
Stress



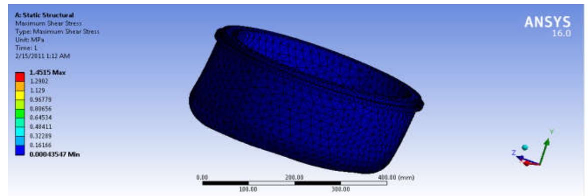
Strain



Total deformation

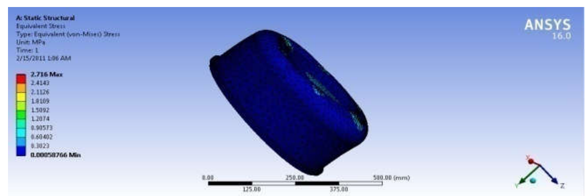


Max Shear stress

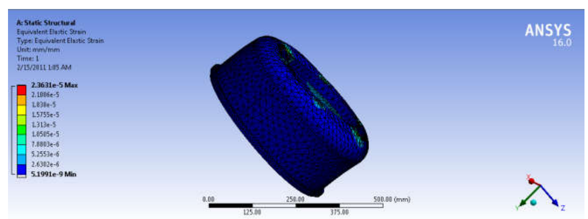


For AlSic

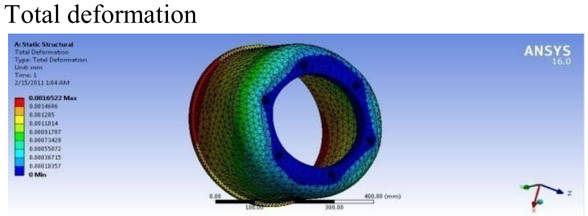
Stress



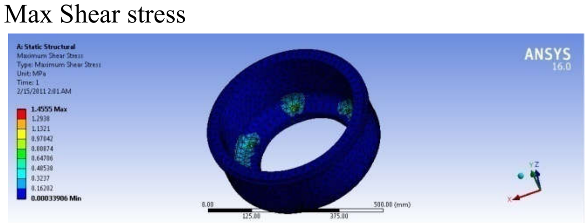
Strain



Total deformation

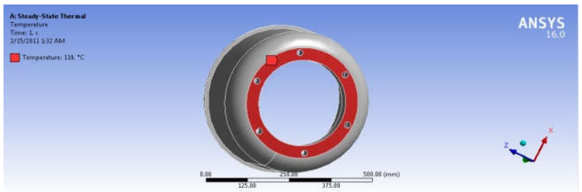


Max Shear stress

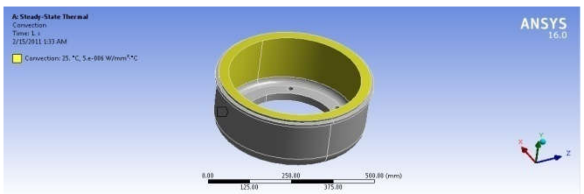


THERMAL ANALYSIS:

Temperature

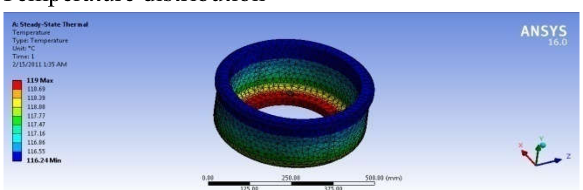


Convection

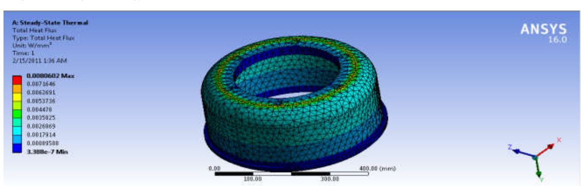


Material: Aluminum Alloy

Temperature distribution

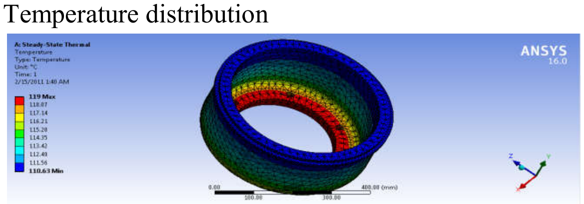


Total heat flux

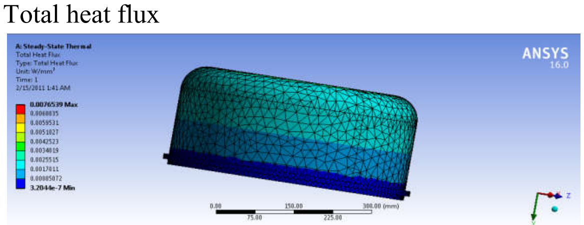


For **Grey cast iron**

Temperature distribution

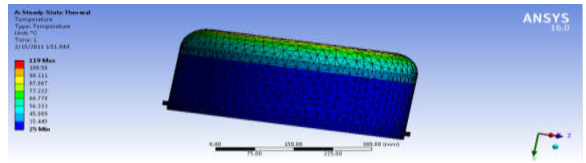


Total heat flux

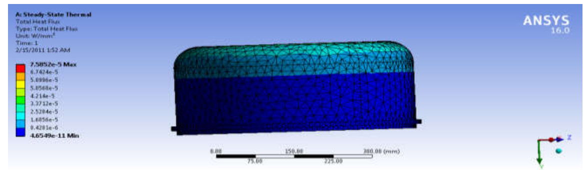


For Aluminum oxide

Temperature distribution

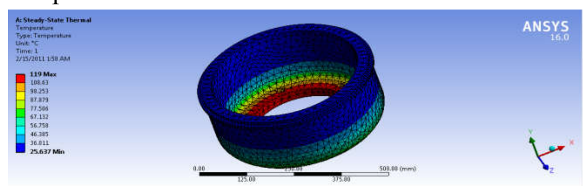


Total heat flux

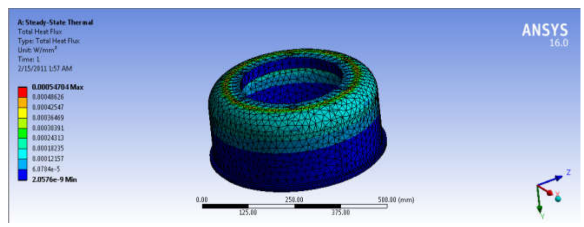


For AlSic

Temperature distribution

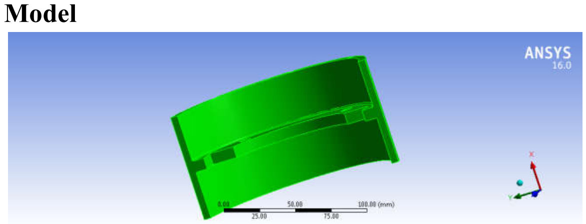


Total heat flux

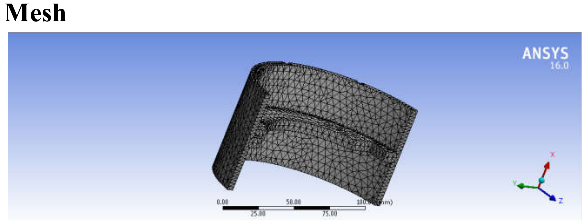


On Brake pad

Model

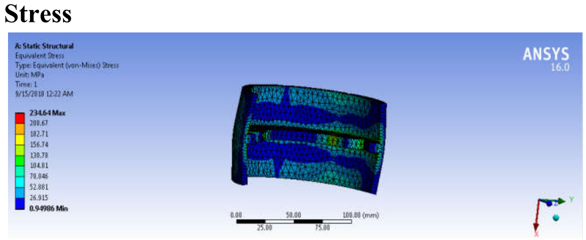


Mesh

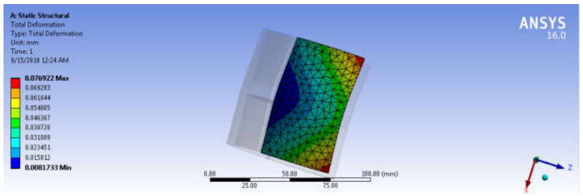


Material: Al2O3

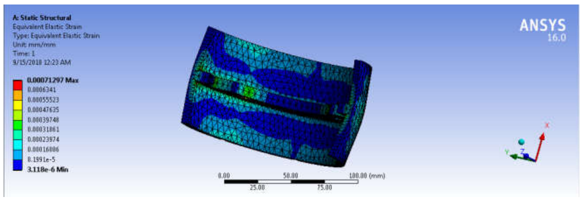
Stress



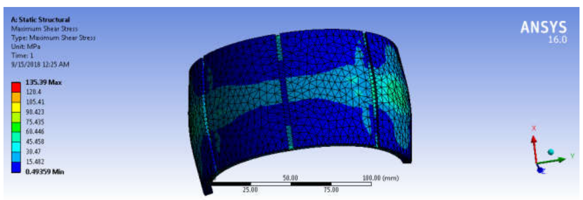
Deformation



Strain

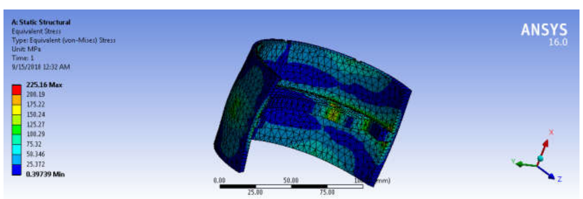


Max Shear stress

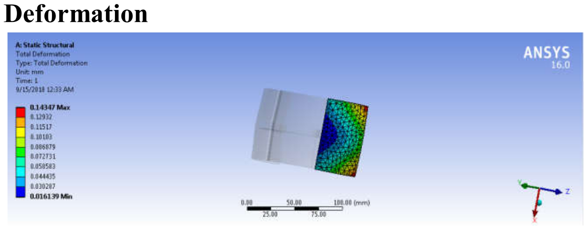


Material: ZIRCONIA

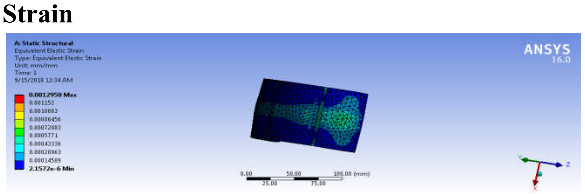
Stress



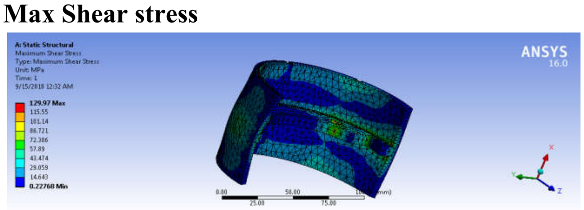
Deformation



Strain

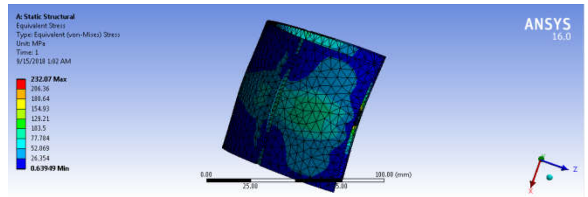


Max Shear stress

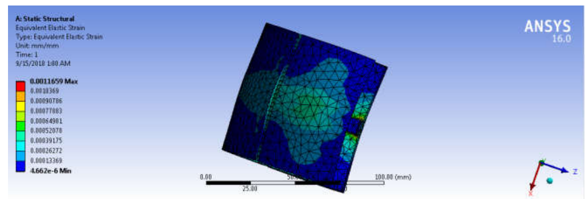


For Cfrp material

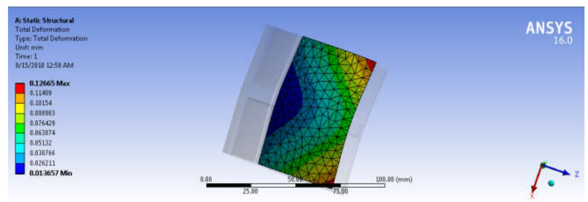
Stress



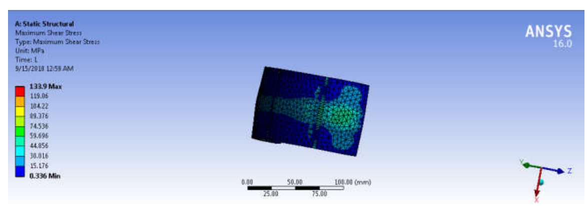
Strain



Deformation



Max Shear stress



Results

STATIC ANALYSIS:

On drum brake

Material	Stress	Strain	Total Deformation	Maximum Shear stress
AlSic	2.716	2.3631e^-5	0.0016522	1.4555
Grey cast iron	2.7051	2.4605e^-5	0.0017319	1.4535
Aluminum alloy	2.6617	375.15	27148	1.4515
Aluminum oxide	2.7902	9.3078	621.87	1.4749

Thermal analysis :

Materials	Temperature distribution		Total heat flux
AlSic	119	25.637	0.0005470
Grey cast iron	119	116.24	0.0076539
Aluminum alloy	119	110.63	0.0080602
Aluminum oxide	119	25	7.5852e^-5

On brake pad
Static analysis

Material	Stress	Strain	Total deformation	Shear stress
Alumina	234.64	0.00071217	0.076922	135.39
Zirconia	225.16	0.001298	0.14347	129.97
Cfrp	232.07	0.0011659	0.12665	133.9

Conclusion:

The use of finite element analysis in product design Validation has been explored in this research work. Computer Aided Analysis was employed to determine the Structural integrity and thermal effects using Finite Element method on motorcycle drum brake and brake pad. The result of the analysis using ANSYS software showed.

- Modeling and analysis of drum brake and brake pad is done.
- Using solid works software with various commands. These components are modeled
- Then the file is saved as IGS to import in Ansys software.
- The drum brake model is imported to Ansys 16.0 work bench software.
- Thermal analysis at 119°C and Structural analysis at 2610N force with four different materials such as Aluminum alloy, Grey cast iron, aluminum oxide, Aluminum silicon carbide is done.
- Structural deformations such as stress, deformation, strain and Thermal calculations of temperature distribution and total heat flux are found and tabulated.
- Similar to that Static analysis of brake pad with three different materials such as Alumina, Zirconia, Cfrp materials is done.
- Structural deformations such as stress, deformation and strain are found and tabulated.
- From the final results we found the Aluminum silicon carbide has less deformations and better heat flux of brake drum as well as Zirconia has less deformations and stress value of brake pad.

References:

[1] Andrew . J. Day. (2014). Braking Of Road Vehicles. Butterworth-
Heinemann Walter, USA PP115-140.

[2] Aduloju S.C., Mgbemena C.O, Maduike J., Abdul rahman J.(2014)
Pivot Hole Thickness Optimization And Material Design For Functional
Requirement Of Front Axle Support. American Journal of Material
science and Application 2(5), PP.63-68

[3] Kang, S.-S., & Cho, S.-K. (2012). Thermal deformation and stress
analysis of disk brakes by finite element method. Journal of Mechanical
science & Technology, Volume 26(Issue 7), PP.1

[4] Ripin, Z. M., & Faudi, Z. (2005). Analysis of Design Parameter
Effects on Vibration mode of a Motorcycle Drum Brake and Brake Shoe.
The Institute of Engineers Malaysia, 66(1), PP.1-2

[5] Madenci, E., & Guven, I. (2006). *The Finite* Element Method and
Application in Engineering Using ANSYS. New York: Springer Science +

[6]. Anup Kumar and R. saharish. Structural and Thermal Analysis of Brake Deum. Middle East Journal of Scientific Research Vol20. (8). IDOSI Publication. Pp 1012-1016. . 2014.

[7]. Frank Kreith. Mechanical Engineerimg Handle Book. Boca Raton: CRC Press LLC. Pp 4-7. 1999.

[8]. Daniel Dannelley, John Baker, Clark K. Midkeff, Robert P. Taylor, Keith A. Woodburry. Enhancement of Extended Surface Heat Transfer
Using Fractal-like Geometry. Department of mechanical Engineering, Graduate School of the University of Alabama. Pp 14-15. 2013.

[9]. Fred Puhn. Brake Handbook. 2rd edition.HP Books Pp4, 15-21. 1985.

[10]. Giri N.K. Automobile Technology. Sixth edition (reprint). Khanna Publisher. Pp 1269-1272.2012.