International Journal of Recent Technology and Engineering (IJRTE)
ISSN: 2277-3878, Volume-8 Issue-4, November 2019

Design Optimization of Muffler of an Internal Combustion Engine using Computational Fluid Dynamics

L.Prabhu, R.Mahesh, Venkatesan.L, Buse Sai Teja, Shaik Shahul

Abstract: A Muffler is a gadget for lessening the measure of clamor produced by the fumes of an inner burning motor. Inner burning motors are commonly furnished with a fumes suppressor (Silencer) created by the ignition procedure to smother the acoustic beat. The direct cause of back pressure in the engine will be the stress and temperature of exhaust gases that flow through the silencer. To relieve the back pressure in the exhaust system, the engine power must be used. Execution of Muffler underneath different working state is typically acquired through design analysis. Accessibility of compressible flow execution parameters is limited, and trial testing can be cost restrictive. The mathematical fluid Dynamics analysis provides better results for this case. The ability to use analytical fluid components is a test to determine their suitability to determine the conditions of their display. The venture's goal is to examine t he gases gas stream attributes in the suppressor's current and modified structure. Our effort is to focus on enhancing the structure so that there is less back weight and silencer's expanded existence. The Computational Fluid elements investigation would be completed by utilizing CFD apparatus CFX. With parameters such as weight and temperature dissemination, disturbance power and liquid power at different load conditions, an extensive report would be done.

Keywords: CFD, Muffler

I. INTRODUCTION

A Muffler (Silencer) is a gadget for lessening the measure of commotion transmitted by the fumes of an inward ignition motor [1]. Muffler are introduced inside the fumes arrangement of most interior ignition motors, despite the fact that the Muffler isn't intended to serve any essential fumes work. The Muffler is built as an acoustic sound sealing gadget intended to diminish the tumult of the sound weight made by the motor by method for Acoustic calming [2]. Most of the sound pressure created by the motor is radiated out of the vehicle utilizing a similar funneling utilized by the quiet exhaust gases consumed by a progression of sections and chambers fixed with meandering fiberglass protection [3-5] and additionally resounding chambers agreeably tuned to cause ruinous obstruction wherein inverse sound waves counteract one another. An unavoidable reaction of muffler use is an expansion of back pressure which diminishes motor productivity [6]. This is on the grounds that the motor fumes must have a similar complex leave pathway worked inside the muffler as the sound weight that the muffler is intended to relieve [7-8]. The automobile muffler must be able to allow exhaust gasses to pass while at the same time restricting sound transmission [9]. There is typically no technical gap between a muffler and a silencer as well as the phrases will be used interchangeably regularly [10]. The traditional name for noise attenuation devices has been a silencer, while a muffler is a smaller, mass-produced device designed to reduce exhaust noise from both the engine [11]. The final selection of such an exhaust muffler is focused on a compromise between the projected acoustic, aerodynamic, mechanical and structural reliability in combination with the arising overall cost in certain applications.

Manuscript published on January 30, 2020.
* Correspondence Author
L.Prabhu, Professor, Department of Mechanical Engineering, Aarupadai veedu Institute of Technology, Vinayaka Mission's Research Foundation, Chennai, India, **prabhu@avit.ac.in**
R.Mahesh*, Assistant Professor, Department of Mechanical Engineering, Aarupadai veedu Institute of Technology, Vinayaka Mission's Research Foundation, Chennai, India,
Venkatesan.L, UG Scholar, Department of mechanical Engineering , Aarupadai veedu Institute of Technology, Vinayaka Mission's Research Foundation, Chennai, India.
Buse Sai Teja, UG Scholar, Department of mechanical Engineering , Aarupadai veedu Institute of Technology, Vinayaka Mission's Research Foundation, Chennai, India.
Shaik Shahul, UG Scholar, Department of mechanical Engineering , Aarupadai veedu Institute of Technology, Vinayaka Mission's Research Foundation, Chennai, India.

II. DEFINITION OF PROBLEM

Automotive muffler's sole purpose is to increasing the emission of engine noise. There will be an excessive amount of noise in their environment if the vehicle did not want a muffler. Numerous mufflers have been designed in recent years for the automobiles and has very little work has been carried out in the development of muffler for stationary engines. Published work concentrates on theoretical modeling and experimental results of mufflers of new shapes However, relatively little is published in order to establish methodologies for designing new, higher performance mufflers in relation to the fundamental understanding of the gas flow behavior.

The following problems are addressed with respect to mufflers

• The main cause of the back pressure in the engine would be the stress and temperature of the exhaust gases that stream past the muffler.

- Conventional designs of mufflers are being used in stationary engines which may produce more noise and back pressure.
- A optimized design of muffler is essential for overcoming the above mentioned problems in stationary engines.

The purpose of the proposal will be as follows

- To analyze the flow characteristics of exhaust gases in the existing and modified design of muffler with respect to the following parameters at varying load conditions

a)Pressure b)Temperature distribution

c) Intensity of turbulence d) Fluid force.

To optimize the design to reduce feedback stress and maximize muffler life.

III. METHODS AND MATERIALS

A. Modeling of Mufflers using ProE

This is the underlying advance in examination process. The main role of geometry creation is to produce a strong that characterizes locale for liquid stream. This area depicts making of geometry. Measurements and geometry subtleties of existing model was gathered. Pro E Wild Fire 2.0 was finished and sent out in the IGES group. The platform models with vanes were shown in the accompanying figures 1 to 6.

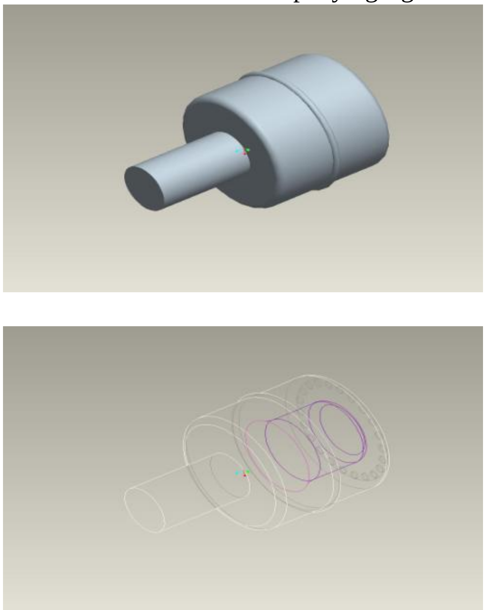
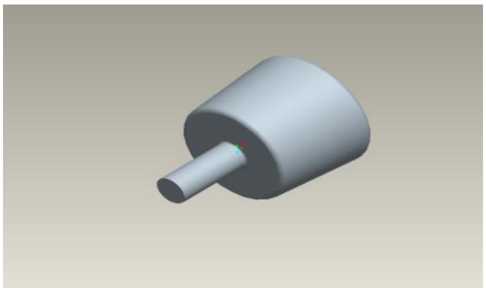


Fig. 1. Existing Model



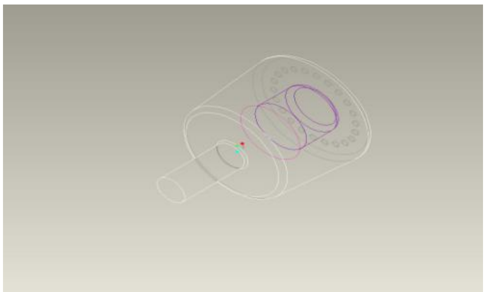


Fig. 2. Conical Model

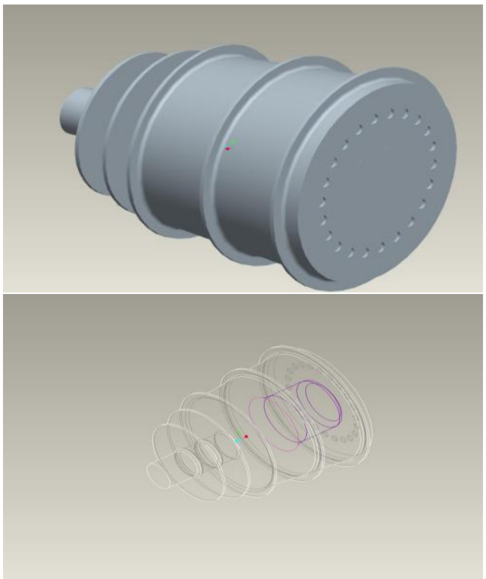


Fig. 3. Conical with fin Model

B. Meshing and Domain Specifications

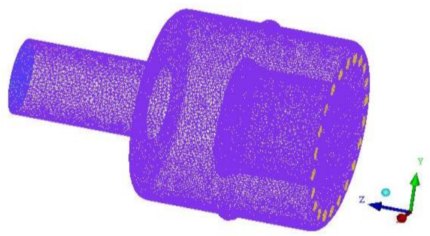


Fig. 4. Meshing Model- Existing

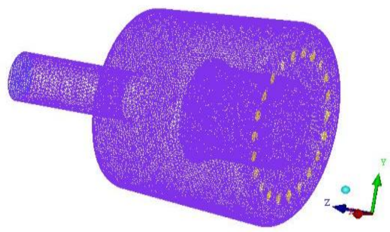


Fig. 5. Meshing Model- Conical

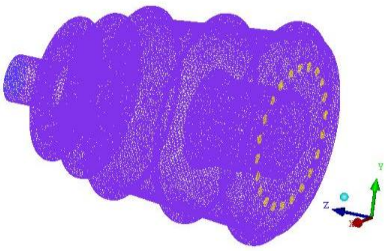


Fig. 6. Meshing Model- Conical with fin

Working age is the procedure for cultivating spatial discretion of the CFD model. Cross section depends on tetrahedron component discretisation. When another work is made, the areas are naturally reassigned.

After model cross section, the matrices on model must be streamlined. This progression is fundamental as it limits The numerical errors in the recalculation work. The aim of this streamlining procedure is to decide an adequate framework size that can arrive at a harmony between the measure of figuring time and the exactness in the arrangement of stream factors. The model is exported in IGES format and used in the CFD tool of ICEM. This tool was used to generate ground and density mesh by identifying the type of mesh component and mesh element size shown in figures 7 to 9.

Element type: Tetrahedron, Global element Factor :2.5, Mesh Type :Volume mesh, Number of Nodes : 172863, No. of elements : 972020

C. Domain Specifications

Type of Domain : Fluid
Domain name : Muffler
Fluid : Exhaust gas
Air Transfer : Thermal Energy
Turbulence : K-Epsilon

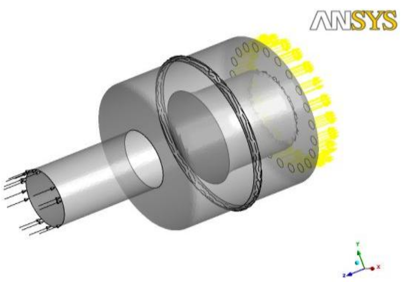


Fig. 7. Existing Model Computational Domain

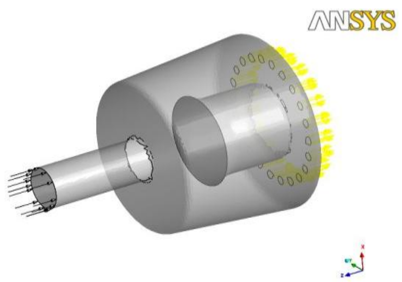


Fig. 8. Conical Model Computational Domain

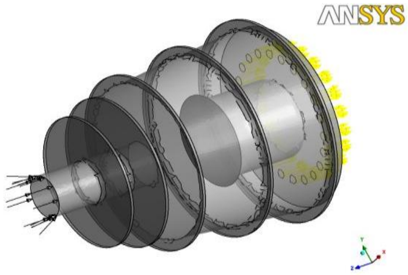


Fig. 9. Conical fin model Computational Domain

D. Boundary Conditions

Inlet

Boundary Type : Inlet
Velocity : 11 m/s (No load)
12 m/s (Full load)
Turbulence : Medium (Intensity=5%)
Gas Temperature : 123 C (No load)
273 C (Full load)
Boundary type of outlet: outlet
Wall Boundary Conditions
Heat Transfer for flow: Adiabatic
Wall on Flow: No Slip condition
Wall Ruggedness: Smooth Wall
Heat transfer coefficient: 20 W/m^{2}K
Outdoor Temperature : 50 C

IV. RESULTS AND DISCUSSION

A. Temperature Distribution

Fig 4 (a, b & c) shows the temperature distribution for existing and modified muffler types. For interpreting the results of temperature distribution in a better way, a user defined temperature range (85°C to 123°C) has been given. While comparing the results of existing and conical type, reduced gas temperature has been observed for the latter case. Consequently this will result drop in pressure along the flow. The result of conical type with fins shows that comparatively reduced gas temperature than others. Moreover significant drop in temperature has been observed in the finned area. The reduced operating temperature also increases the life of the muffler shown in figures 10 to 12.

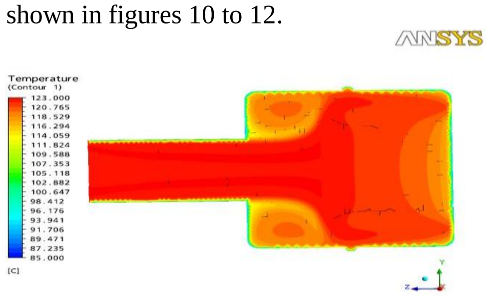


Fig. 10. Existing Model

Retrieval Number: E3107018520/2020©BEIESP
DOI:10.35940/ijrte.E3107.018520
Journal Website: www.ijrte.org

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

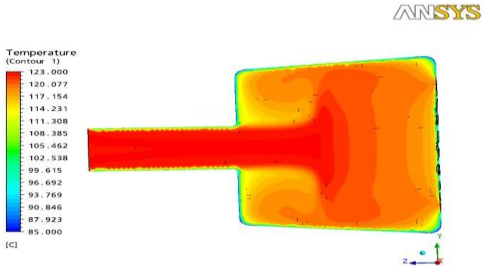


Fig. 11. Conical Model

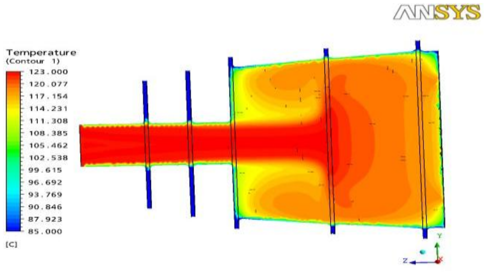


Fig. 12. Conical with fin Model

B. Pressure Distribution

Fig 5 (a, b & c) shows pressure variation along the flow for existing and modified muffler types. The results have been compared with user defined pressure values. In the case of existing muffler sudden pressure drop has been observed near the wall on either sides which result in pressure fluctuation and consequently increased back pressure. For conical type the expansion is relatively gradual and also increased pressure drop. While observing the results conical with fins the expansion and pressure drop is almost similar to the conical type. There has been slightly an increase in pressure has been observed at in inlet of muffler for this case shown in figures 13 to 18.

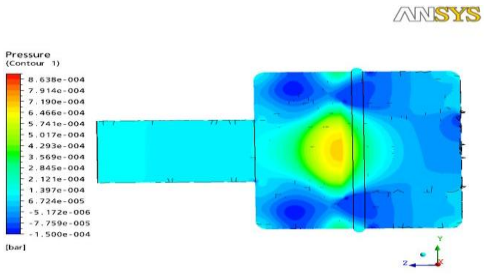


Fig. 13. Existing Model

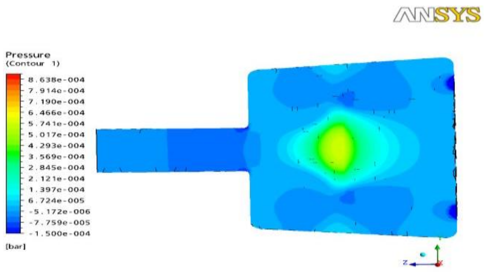


Fig. 14. Conical Model

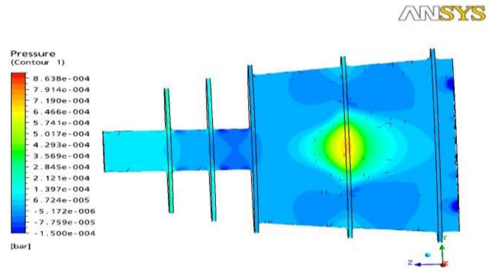


Fig. 15. Conical with fin Model

C. Turbulence Eddy Dissipation variation

Fig 6 (a, b & c) shows the variation of turbulence eddy dissipation for the existing and modified types. Higher the value Turbulence eddy dissipation refers higher the energy loss in the flow. The results have been compared with user defined values. Increased turbulence eddy dissipation has been observed for existing type at the flow entry to porous medium and near wall. This is comparatively less in the case of conical and conical with fin types.

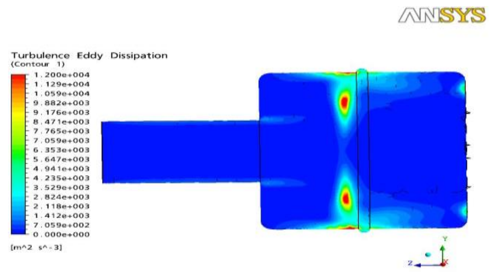


Fig. 16. Existing Model

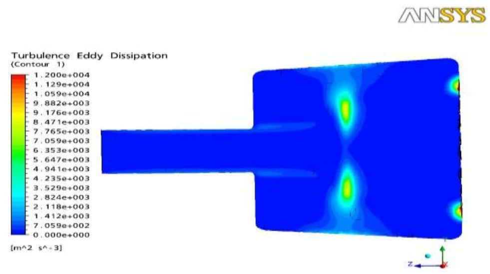


Fig. 17. Conical Model

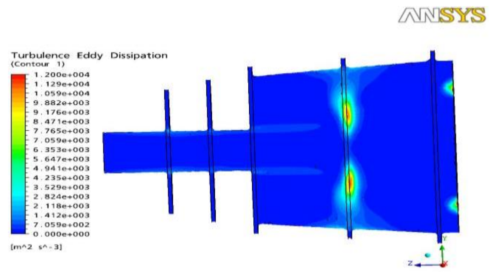


Fig. 18. Conical with fin Model

Retrieval Number: E3107018520/2020©BEIESP
DOI:10.35940/ijrte.E3107.018520
Journal Website: www.ijrte.org

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

D. Pressure Gradient

Fig 7 (a, b & c) shows variation of pressure gradient for the existing and modified muffler types. Higher the pressure gradient in the flow will lead to increased back pressure. It has been observed from the results the pressure gradient exist at flow entry to porous medium. The magnitude of pressure gradient is high for existing type at near wall on either side and it is relatively low in the case of conical type. A sharp rise in pressure gradient has been observed at near outlet for conical and conical with fins shown in figures 19 to 21.

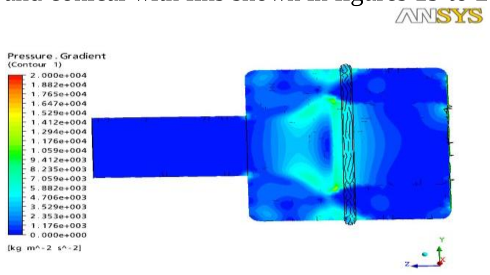


Fig. 19. Existing Model

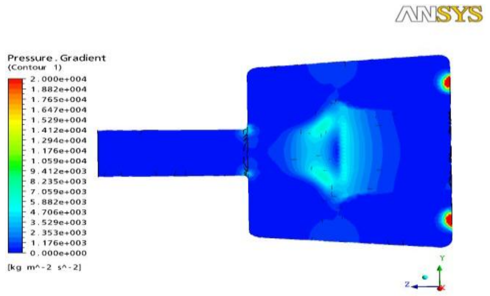


Fig. 20. Conical Model

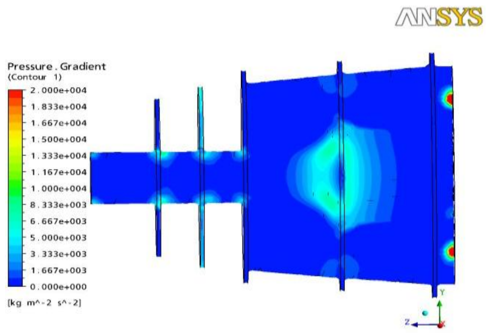


Fig. 21. Conical with fin Model

E. Velocity vectors

Fig 8 (a, b & c) shows the velocity vectors for existing and modified muffler types. The strong recirculation zones have been observed in the flow near porous medium for all the types. Recirculation in the flow causes flow separation and energy dissipation. Increased velocity gradient has been observed for the existing muffler type which results in fluctuation in the flow. The velocity gradient is relatively low in the case of conical and conical with fins in figures 22 to 24.

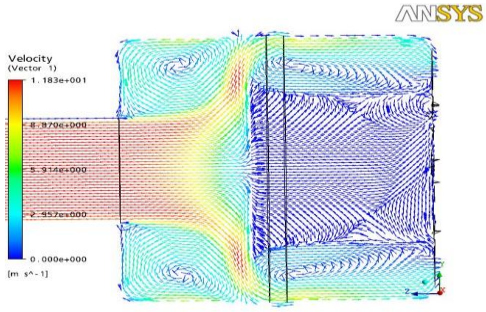


Fig. 22. Existing Model

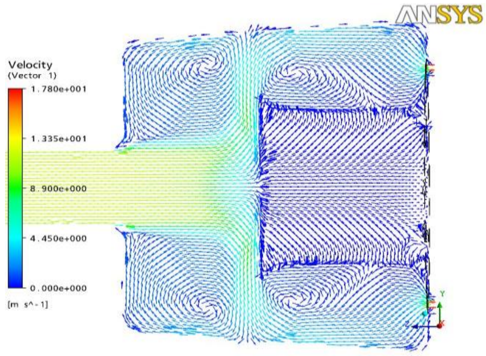


Fig. 23. Conical Model

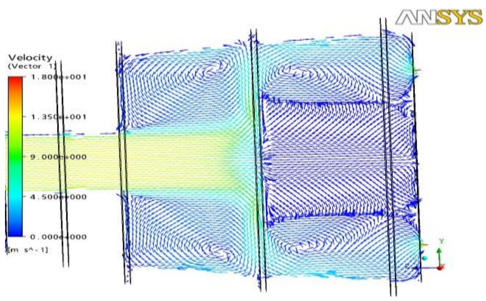


Fig. 24. Conical with fin Model

F. Wall Shear Distribution

Fig (a, b & c) shows the distribution of wall shear for existing and modified muffler types. The wall shear refers the magnitude of shearing force on the wall due to the gas flow. The result of existing type shows that increased wall shear along the entire surface. This will consequently increase the rate of wear and decrease its life. The value of wall shear has been observed high at near inlet and it has reduced significantly for the conical and conical fin types in figures 25 to 27.

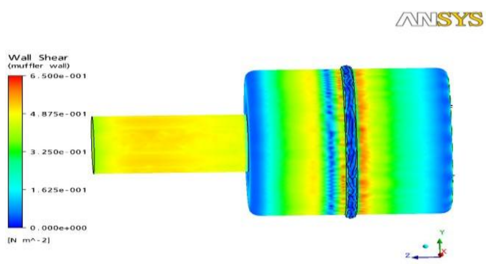


Fig. 25. Existing Model

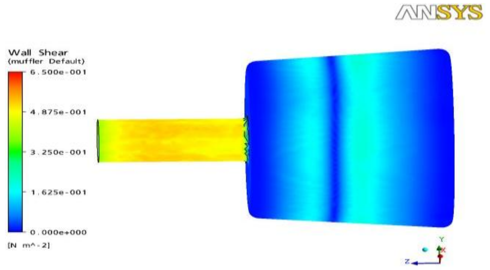


Fig. 26. Conical Model

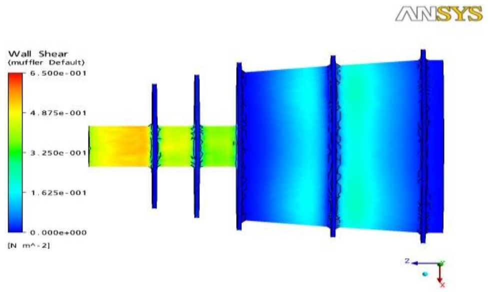


Fig. 27. Conical with fin Model

V. CONCLUSION

Thus Computational fluid dynamics analysis of muffler of an internal combustion engine has been carried out. The parameter variations have been tabulated for no load and full load condition.

The following conclusion has been drawn from the analysis.

- Relatively reduced back pressure and turbulence eddy dissipation have been observed for conical type
- The cooling rate is comparatively higher for conical with fin type muffler
- The velocity gradient is less in the case of modified types which will minimise the fluctuation in the flow.

The shear stress on the muffler walls for modified types has been significantly reduced which will increase the life span.

REFERENCES

1. J.F.Allard. Propagation of sound in porous media . Elsevier Applied science, 1993.
2. A.S. Bonnet, D. Drissi, N. Gmati. Simulation of muffler's transmission loss by a homogenized element method. Journal of computational acoustics. 12(3), pp 447-474, 2004.
3. M.L. Munjal. Acoustics of ducts and mufflers, with application to exhaust and ventilation system design. John Wiley and sons, 1987.
4. M.L. Munjal, S. Krishnan and M.M. Reddy. Flow-acoustic performance of perforated element mufflers with application to design. Applied acoustics, Vol 40, 159-167, 1993.
5. D.R. Ross. A finite element analysis of perforated component acoustic systems, J.Sound and Vibration, **79** (1), 133-143, 1981.
6. E. Sanchez-Palencia. Non homogeneous media and vibration theory. Springer Verlag, Berlin.
7. ANSYS Europe Ltd, ANSYS CFX solver modeling guide, V11, Technical report, ANSYS Europe Ltd, 2007.
8. Ankit Singh, Dr. Nitin Shrivastava, "Study Of Noise Behavior On Mufflers For Ic Engine: A Review" International Journal Of Engineering Sciences & Research Technology, 4.(8): August, 2015.
9. Vishal A. Andhale, Dr. D. S. Deshmukh, "Investigation Of Ball Valve Design For Performance Enhancement" International Journal Of Science, Spirituality, Business And Technology (Ijssbt), Vol. 4, No. 2, May 2016.
10. S.Prabhakar, S.Prakash* , M.Saravana Kumar,K.Annamalai, "Performance Analysis Of Ventilated Brake Disc For Its Effective Cooling" Journal of Chemical and Pharmaceutical Sciences, JCHPS Special Issue 7: 2015
11. L.Prabhu, S.Prakash, S.Sangeetha, Nidhish Thomas "Effect Of Nozzle Angle In Water Jet Machining Using Computational Fluid Dynamics" International Journal of Pure and Applied Mathematics Volume 118 No. 5 2018, 1019-1029.

AUTHORS PROFILE

L.Prabhu*, Professor, Department of Mechanical Engineering, Aarupadai veedu Institute of Technology, Vinayaka Mission's Research Foundation, Chennai, India. prabhu@avit.ac.in

R.Mahesh*,Assistant Professor, Department of Mechanical Engineering, Aarupadai veedu Institute of Technology, Vinayaka Mission's Research Foundation, Chennai, India. mahesh@avit.ac.in

Venkatesan.L – UG Scholar Department of Mechanical Engineering, Aarupadai veedu Institute of Technology, Vinayaka Mission's Research Foundation, Chennai, India

Buse Sai Teja - UG Scholar Department of Mechanical Engineering, Aarupadai veedu Institute of Technology, Vinayaka Mission's Research Foundation, Chennai, India

Shaik Shahul, UG Scholar Department of Mechanical Engineering, Aarupadai veedu Institute of Technology, Vinayaka Mission's Research Foundation, Chennai, India