Open Access Research Journal of
Science and Technology
Journals home page: https://oarjst.com/
ISSN: 2782-9960 (Online)

OARJ OPEN ACCESS RESEARCH JOURNALS

(Research Article)

Check for updates

Optimizing Automotive Silencer Durability: Thermal Analysis to Mitigate High Temperature Failure

Rajesh D. Bhosale [1], Prashant Singh [2] and D. S. Shelar [3,*]

[1] *Research Scholar, Bir Tikendrajit University, Manipur, India & Lecturer, AISSMS Polytechnic, Pune, India.*
[2] *Department of Mechanical Engineering, Bir Tikendrajit University, Manipur, India.*
[3] *Department of Engineering Sciences, AISSMS Institute of Information Technology, Pune, Maharashtra, India.*

Open Access Research Journal of Science and Technology, 2025, 15(02), 102-111

Publication history: Received on 26 October 2025; revised on 02 December 2025; accepted on 05 December 2025

Article DOI: https://doi.org/10.53022/oarjst.2025.15.2.0139

Abstract

This study analyzes and improves the heat dissipation of an exhaust silencer by modifying the inner tube geometry and evaluating the effects using simulation software. The exhaust system directs hot combustion gases away from the engine and occupants while reducing noise. High-speed exhaust flow can cause resonance and potential fatigue failure if not addressed in the design. Recent research has examined heat transfer in automotive exhaust piping to optimize aftertreatment systems. In this study, the silencer is modeled with CAD software to assess current heat dissipation. Modifications are then made to the inner tube length and perforation count. ANSYS simulations are used to analyze temperature distribution and determine the most effective configuration.

Keywords: Silencer; CFD; Temperature analysis; Finite element analysis; 3D Modeling; Contour of temperature

1. Introduction

An Automobile Silencer is a device used to reduce engine noise emissions, with its primary function being to minimize noise produced by exhaust gases. Silencers are also found in many other engines and generators. Their size, shape, and construction vary depending on the engine type and size. A reactive silencer typically features a series of resonating and expansion chambers that lower sound pressure at certain frequencies, using offset, perforated tubes to disperse sound pulses inside a chamber, which leads to destructive interference. An absorptive or dissipative silencer instead relies on absorption to convert sound energy into heat within specific materials. Uniform heat distribution in both types is beneficial. While research on exhaust silencers dates back over ten years, emphasis on heat transfer analysis in these systems has only recently increased due to their significant role in exhaust after-treatment design, especially for optimizing catalyst performance and managing thermal aging and cold-start operation of converters.

Oncoming automotive exhaust emissions standards focus on the minimization of cold start emissions for catalyst-equipped diesel automobiles. In that context, all passive means of achieving Maximum possible catalyst efficiency must be exhaustively examined. The complex geometry of the exhaust silencer and the special flow conditions complicate the problem of accurately estimating several heat transfer parameters. The acquisition of useful data for the estimation of heat transfer rates and their application in the optimized design of various exhaust configurations forms the subject of the present work. Here, we take advantage of computer models to analyze the effect of the variation of various parameters that govern the heat transfer in the silencer. Selected results are illustrated in the form of temperature, heat transfer, and Thermal Analysis for the motorbike exhaust Silencer for ensuring reduction in Hot Spots through Design Enhancement. Some suggestions for design improvements are therefore also presented.

* Corresponding author: D. S. Shelar

2. Literature review

The design and analysis of automotive exhaust systems constitute a sophisticated engineering domain where competing objectives of noise suppression, engine performance, and thermal management must be optimized through an integrated approach. Foundational research firmly established thermal analysis as a critical pillar of exhaust design, with studies [5], [6] developing computational frameworks for steady-state and transient heat transfer, which are essential for ensuring component durability, managing under hood temperatures, and optimizing the light-off behavior of catalytic converters. Parallel to this, the core acoustic challenge maximizing transmission loss (TL) to meet noise regulations while minimizing flow resistance (backpressure) to preserve engine efficiency has been extensively explored. Analytical methods, such as the collocation approach for specific muffler geometries [8] and the advanced transversal sub-structuring mode-matching technique for dissipative silencers [1], provide efficient tools for acoustic prediction. However, the fidelity of such models, particularly for complex elements like perforated ducts, requires rigorous experimental validation, as demonstrated by work [2] that identified critical failure modes in common discrete models.

The advent of computational fluid dynamics (CFD) has revolutionized this space, enabling detailed multi-physics investigations. Researchers employ 3D CFD to dissect the interplay between acoustic attenuation and pressure loss in perforated tube silencers [14], use one-dimensional CFD for efficient tailpipe noise prediction validated against experiments [9], and apply high-fidelity Large Eddy Simulation (LES) to resolve complex flow fields within mufflers [22]. This evolution has progressed toward fully coupled Multiphysics simulation, where acoustics are integrated with structural mechanics to predict vibroacoustic behaviour and shell vibrations that can compromise transmission loss predictions and lead to fatigue failure. This advanced computational capability has directly enabled the field's most significant trend: the shift from passive analysis to active, automated shape and parametric optimization. Sophisticated algorithms, including Simulated Annealing (SA) for multi-chamber crossflow mufflers [3], genetic algorithms coupled with neural networks for rapid assessment [13], and general parametric techniques [4], are now routinely used to navigate high-dimensional design spaces and achieve superior acoustic performance under constraints. The frontier of this optimization is now being pushed by data-driven co-optimization frameworks. Techniques such as parameter fusion convolutional networks (PFCNs) coupled with support vector machines (SVMs) enable the rapid prediction of complex multi-physics fields like spatial temperature distribution and the subsequent co-optimization of multiple structural parameters with negligible computational cost, a method with clear transfer potential to exhaust system design.

Contemporary studies continue to refine these principles, focusing on practical backpressure optimization [16], novel configurations like micro-perforated elements for motorcycles [21] and baffle plate-integrated chambers [19], and comprehensive acoustic-fluid analysis of specific designs [12], [17], [23]. Furthermore, the literature shows emerging interdisciplinary investigations, such as evaluating the coupled thermal-acoustic effects of material choices [15] and designing hybrid mufflers that combine low-pass filter and Helmholtz resonator principles [18]. Review articles systematically consolidate knowledge on reactive muffler performance [20], while applied case studies demonstrate the translation of these principles to specific vehicle platforms, from automotive engines [10] to motorcycles [11]. In synthesis, the body of work reveals a clear epistemological progression from foundational experimental and analytical studies, through a phase dominated by high-fidelity multi-physics simulation, to the current frontier of simulation-driven and data-driven optimization. Despite these advances, a discernible research gap remains in the fully coupled, system-level co-optimization that simultaneously and dynamically accounts for transient thermal effects, nonlinear acoustic behaviour, structural vibrations, and after-treatment chemical efficiency, pointing toward the next evolution in exhaust system design methodology.

3. Methodology

Analytical solutions give the values of the unknowns at all points in the continuum and are appropriate only for simple problems. For complex geometries, boundary conditions and numerical methods are best suited. These give only an approximate solution. Finite element methods are some of the more versatile and numerical methods. In FEM, the whole body is divided into smaller elements. These elements are interconnected at joints called nodes. The unknowns are calculated only at these nodes. Increasing the number of nodes can increase the accuracy of the results. Finite element methods are predominately used to perform computer-based analysis of the static, dynamic, and thermal behaviour of physical systems, structures, and components. Finite Element Analysis (FEA) is a computer-based numerical technique for calculating the strength and behaviour of structures. The computer is required for this analysis because of the astronomical number of calculations needed to analyze a complex structure.

In the finite element method, a structure is broken down into many small simple blocks or elements. The behavior of an individual element can be described with a set of equations. Just as the set of elements would be joined together to build the whole structure, the equations describing the behaviors of the individual elements are joined into an extremely large set of equations that describe the individual elements.

3.1. Solver, Models and Schemes Used For CFD Analysis

The physical model of this test setup would be passing air at a fixed mass flow rate through the silencer and measuring temperature across the silencer. The temperature is the same as the temperature acting on the engine through the exhaust system. The time conditions implemented are steady state. The mass flow input is given as a constant number, 0.01 kg/s, so the flow is subsonic flow.

A flow becomes incompressible for low speed, that is, $M<0.3$ for air, and compressible for Mach number $M\geq0.3$, depending on pressure and density changes relative to the local speed of sound. So, the computational schemes are dictated by various physical conditions: viscosity, incompressibility, and compressibility of the flow.

The solver implemented was temperature-based, as it is used for incompressible flows, which may arise due to difficulties in preserving incompressibility conditions or conservation of mass as the speed of sound becomes much higher than convection velocity components.

In Fluent, solver variables are stored at the center of grid cells. To solve the transport equations, we need to know the values of quantities at the faces of the control volume and the gradient of these quantities in the cell. The quantities are namely momentum, turbulent kinetic energy, turbulent dissipation rate, and energy.

So, for interpolating the cell center data, various interpolating schemes are available in Fluent. This interpolation in ANSYS-FLUENT is called upwind schemes. Upwinding means that the face value is derived from quantities in the cell upstream, or “upwind,” relative to the direction of the normal velocity. Of the available schemes, the first-order upwind method is used for interpolating the values of these quantities so as to reduce computation effort. Gradients are needed not only for constructing values of a scalar at the cell faces but also for computing secondary diffusion terms and velocity derivatives. The gradient of a given variable is used to discretize the convection and diffusion terms in the flow conservation equations. Thus, to determine the gradient of these variables, the Green–Gauss node-based method is implemented. This scheme reconstructs exact values of a linear function at a node from surrounding cell-centered values on arbitrary unstructured meshes by solving a constrained minimization problem, preserving a second-order spatial accuracy. The node-based gradient is more accurate than other gradient schemes, especially on (skewed and distorted) unstructured meshes, at a higher computing power cost. For calculating the cell-face temperature, as a temperature-based solver is implemented, a standard interpolation scheme is used.

Most of the engineering flows are turbulent. Turbulence occurs when velocity gradients are high, resulting in disturbances in the flow domain as a function of space and time. Turbulent flow arises in contact with walls or in between two neighbouring layers of different velocities. With the velocity gradient increasing, the flow becomes rotational, leading to vigorous stretching of vortex lines, which can only be supported in three dimensions. So, turbulent flows are always physically three-dimensional. In turbulent flows large and small scales of continuous energy spectrum, which are proportional to the size of eddy motions, are mixed. Here, eddies are overlapping in spaces, with large ones carrying the small ones. In this process the turbulent kinetic energy transfers from larger eddies to smaller ones, with the smallest eddies eventually dissipating into heat through molecular viscosity.

So, it is necessary to model the turbulence model appropriately. The Standard K-Epsilon model (SKE) is used as it accurately represents engineering turbulence for industrial applications. Also, it is robust and reasonably accurate for a wide range of applications. Also, it accurately represents low-speed incompressible flows in isotropic turbulence. The standard K-Epsilon model is a two-equation turbulence model and allows the determination of both a turbulent length and time scale by solving two separate transport equations. It is based on model transport equations for the turbulence kinetic energy (K) and its dissipation rate (ε). The model transport equation for K is derived from the exact equation, while the model transport equation for ε is obtained using physical reasoning. In the derivation of the K-ε model, the assumption is that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard K-ε model is therefore valid only for fully turbulent flows.

4. Analysis

4.1. Creation of model of silencer

For our analysis we need 3 different models. Sketching, Extrude, and Revolve are examples of basic modeling tools used for geometry creation and detailing. Fig. 1 displays 3D model development with Revolve.

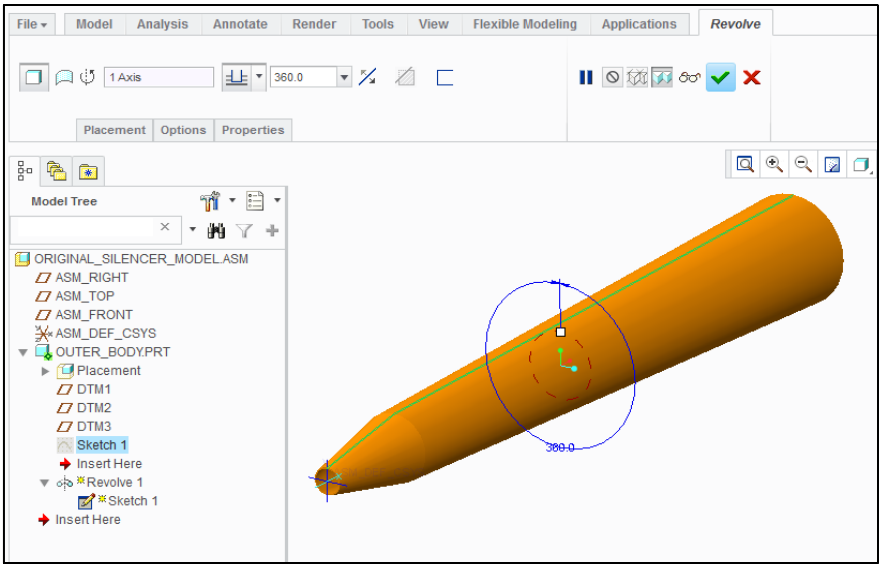

Figure 1 3-D model of silencer

4.2. Meshing

Meshing is a crucial aspect for any finite element analysis. Here, meshing is done by the smart sizing option. One problem encountered is that if the same element size is given to all the models, then the problem of uneven distribution of meshes comes up, which would affect the result. Therefore, the meshes are selected really fine for bigger models, and with a decrease in the thickness, the coarseness was increased at a rate of change in mesh for every 2 thickness changes.

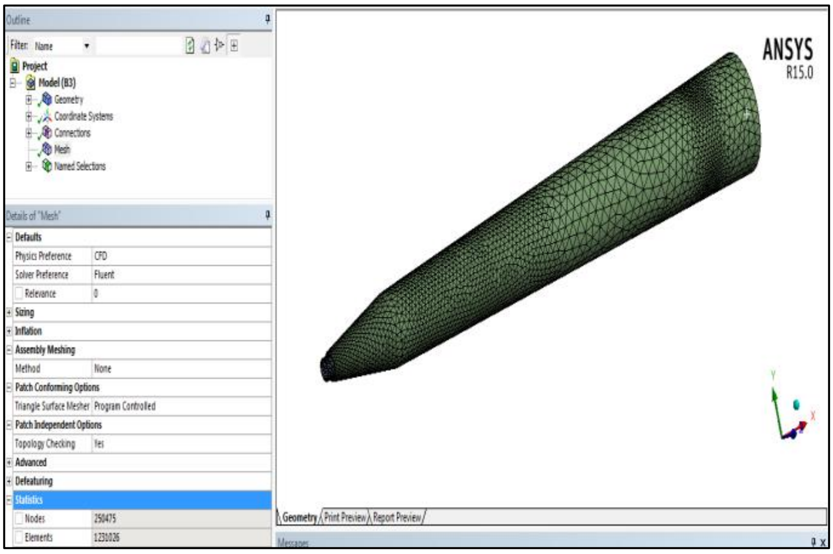

Figure 2 Meshed model of silencer

4.3. Boundary Conditions

The mass flow rate is assigned to air at the rated speed of the engine as 0.01 kg/s. The tests were taken at a constant maximum inlet temperature of 400°C and an outlet temperature of 27°C. The outlet boundary condition is set to pressure outlet with gauge pressure of 0 pa. The density of the air is set as 1.125 kg/m^3, thermal conductivity is 0.0242 W/m-k, and the viscosity of the air is 1.7894×10^{-5} kg/(ms). Turbulent intensity is assumed to be 5%, and the turbulent viscosity ratio is 10.

5. Result And Discussion

5.1. Effect of change in length and no. of perforations of inner tube of silencer

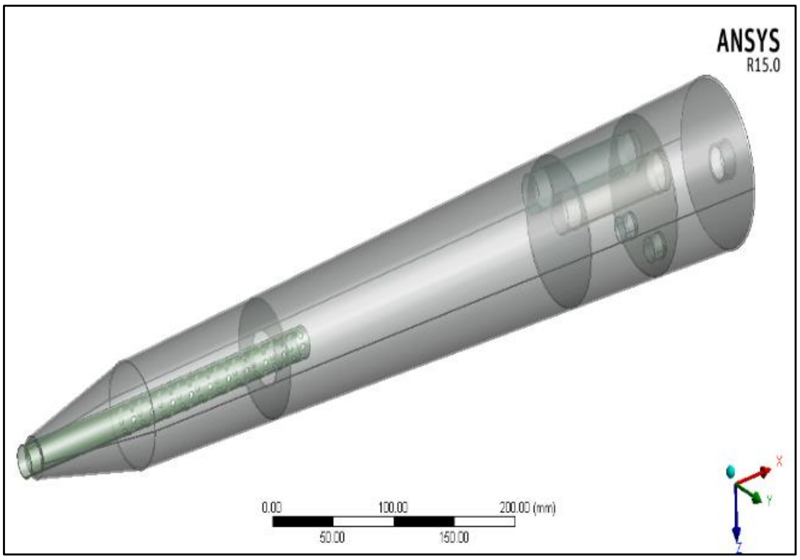

Figure 3 CAD Model of silencer

5.1.1. Model 1: Inlet tube Diameter-24mm, Tube Length: 340 mm, No. of Perforations- 90 (9 x 10)

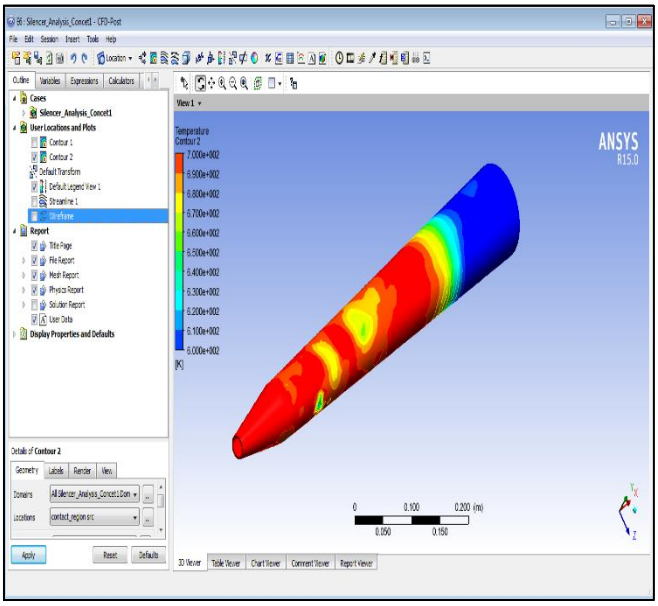

Figure 4 Temperature contour of model 1

Temperature is reduced in this model but high temp is observed which is not recommended. As diameter of tube reduced and number of perforations increased, heat in uniformly distributed over the length. But high temperature is there.

5.1.2. Model 2: Inlet tube Diameter- 26mm, Tube Length- 380 mm, No. of Perforations- 88 (8 x 11)

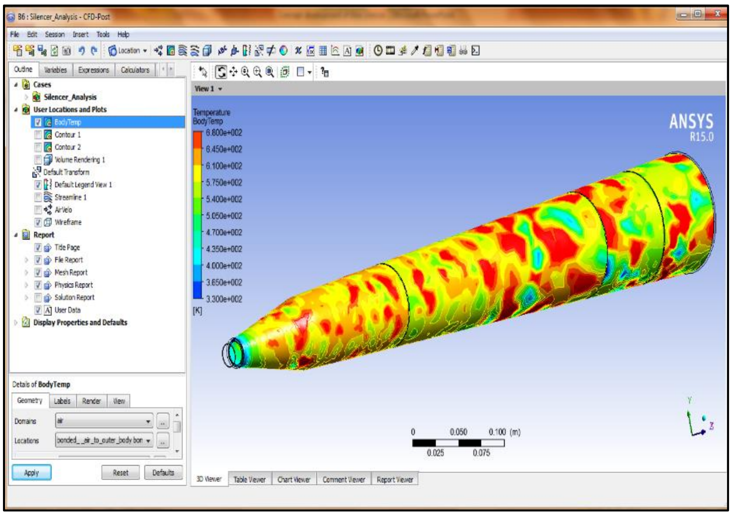

Figure 5 Temperature contour of model **2**

Temperature is reduced in this model but many high temperature spots are observed. Also temperature distribution is not uniform. Here tube length increased in this case temperature is reduced and number of perforations is reduced.

5.1.3. Model 3: Inlet tube Diameter- 26mm, Tube Length- 320 mm ,No. of Perforations-96 (8 x 12)

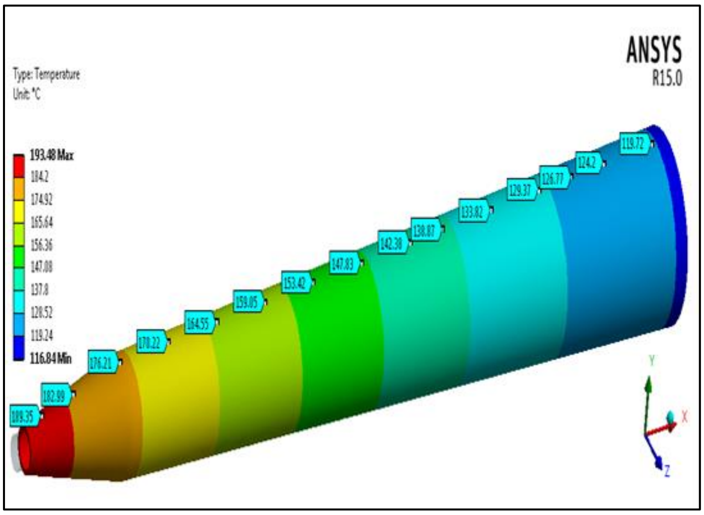

Figure 6 Temperature contour of model 3

Three different concepts were developed to achieve the target for the lowest possible hot spots. The third concept is such that it has an optimum number of holes distributing hot air evenly in the silencer's volume. This concept can be further experimented with, as the flow of gases is also uniform. Pressure distribution is found to be adequately distributed throughout the volume.

6. Experimentation

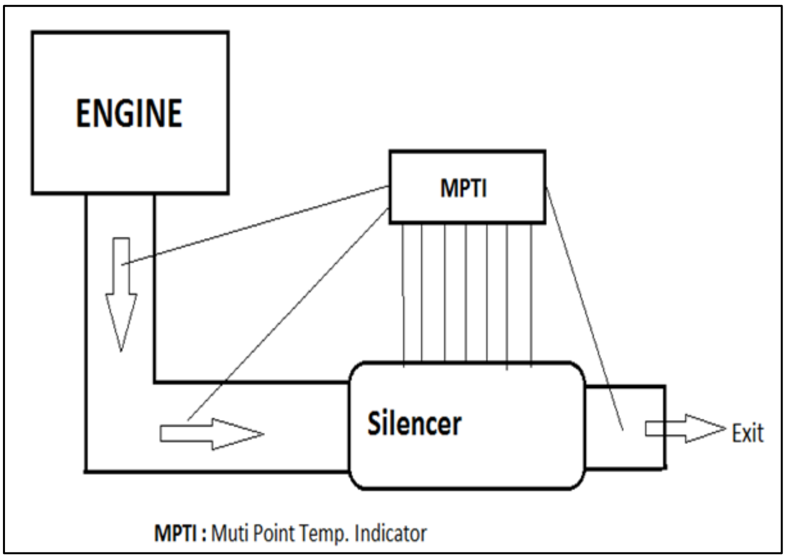

Figure 7 Schematic of Expt. Setup

Figure 7 shows the schematic diagram of the experimental setup for temperature measurement with a thermocouple. The thermocouple used for temperature measurement is K-type with accuracy ±10°C. The silencer used for the project is a reactive-type silencer with specified dimensions. The automobile silencer under study belongs to a popular 2-wheeler manufacturer in India with the rated HP of the engine up to 13.5 HP. The exhaust gases coming out from the engine are at very high speed and temperature. The spots are located at a distance of 10 cm downstream of the silencer. The location of temperature should be at such a location where the length between two spots is constant. Silencer tests in the experiment have been performed for a fixed mass flow rate of 0.01 kg/s. All the solver conditions, turbulence modeling, and boundary conditions have been kept the same as in the CFD analysis of the perforated inner pipe of the silencer.

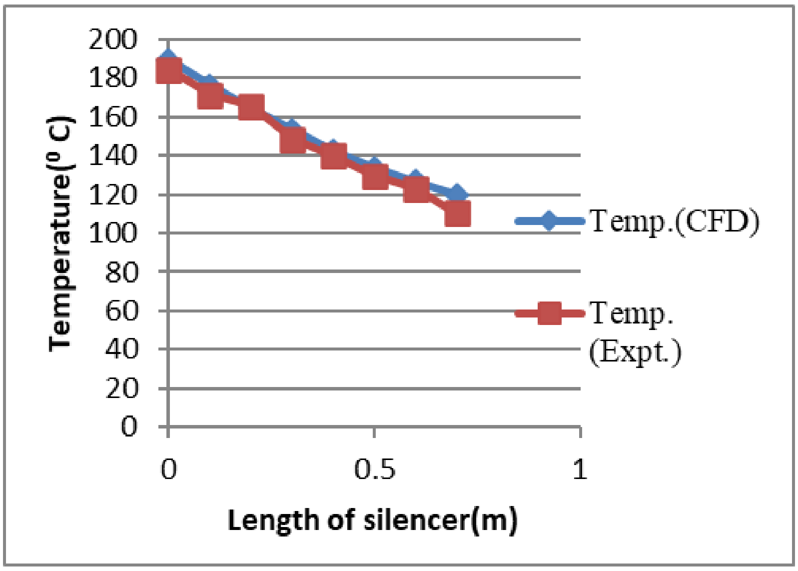

Figure 8 Graph of Temp. Vs Length

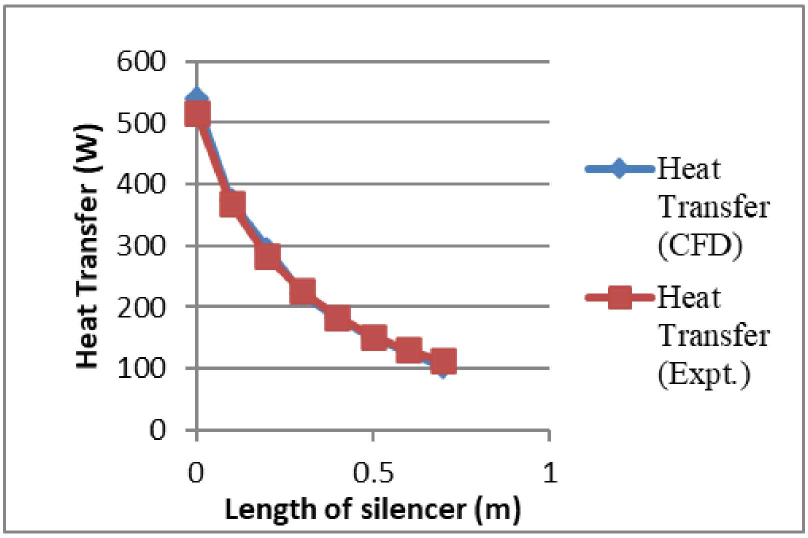

Figure 9 Graph of Heat transfer Vs Length

7. Conclusion

The dimensions of the silencer were varied while keeping certain parameters constant, and the corresponding effects on temperature were observed. It was found that temperature variation is nonlinear and cannot be accurately predicted using any simple equation. Modifying the porosity of the inner tube of the silencer had a significant impact on temperature, with higher porosity resulting in a substantial reduction. Similarly, an increase in the tube diameter led to a marked decrease in temperature. Changes in the number of holes also had a pronounced effect, with even small variations in pipe diameter causing sharp changes in temperature values.

Three silencer models were tested with different porosity levels. A sudden decrease in temperature was observed as porosity increased. The computational fluid dynamics (CFD) results showed good agreement with the experimental measurements, validating the numerical model.

Compliance with ethical standards

Acknowledgement

I would like to acknowledge the use of Grammarly and QuillBot for providing grammar, spelling, and punctuation checks to improve the readability and language of the manuscript.

Disclosure of conflict of interest

No conflict of interest to be disclosed.

Compliance with ethical standards

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References

[1] I. P. Kandylas and A. M. Stamatelos, "Engine exhaust system design based on heat transfer computation," Energy Convers. Manag., vol. 40, no. 10, pp. 1057–1072, 1999. [Online]. Available: https://www.sciencedirect.com/science/article/abs/pii/S0196890499000084sciencedirect

[2] P. A. Konstantinidis, G. C. Koltsakis, and A. M. Stamatelos, "Transient heat transfer modelling in automotive exhaust systems," Proc. Inst. Mech. Eng., Part C: J. Mech. Eng. Sci., vol. 211, no. 1, pp. 1–15, 1997.

[3] M. R. Kulemin, A. V. Sazhin, and A. V. Yushkin, "Durability prediction of EN 1.4512 exhaust mufflers under thermomechanical fatigue," Int. J. Mech. Sci., vol. 84, pp. 140–150, Jul. 2014. [Online]. Available: https://www.sciencedirect.com/science/article/abs/pii/S0020740314001271. pp. 140–150.

[4] J. Albelda, F. D. Denia, M. I. Torres, and F. J. Fuenmayor, "A transversal sub-structuring mode-matching method applied to the acoustic analysis of dissipative mufflers," Appl. Acoust., vol. 67, no. 7, pp. 652–675, Jul. 2006.

[5] Y. Aurélien and M. Leroux, "Failures in discrete models for flow ducts with perforations: An experimental investigation," J. Sound Vib., vol. 312, no. 4–5, pp. 589–611, May 2008.

[6] F. D. Denia, A. Selamet, F. J. Fuenmayor, and R. Kirby, "Acoustic attenuation performance of perforated dissipative mufflers with empty inlet/outlet extensions," J. Sound Vib., vol. 305, no. 4–5, pp. 757–776, Sep. 2007.

[7] C. J. Wu, X. J. Wang, and H. B. Tang, "Transmission loss prediction on SIDO and DISO expansion chamber mufflers with rectangular section using the collocation approach," World J. Eng., vol. 7, suppl. 3, pp. 425–426, 2010.

[8] L. Fagerberg, "Multiphysics simulation provides accurate muffler designs," COMSOL Blog, Feb. 24, 2016. [Online]. Available: https://www.comsol.com/blogs/multiphysics-simulation-provides-accurate-muffler-designs

[9] K. Fonseca de Lima, A. Lenzi, and R. Barbieri, "The study of reactive silencers by shape and parametric optimization techniques," Appl. Acoust., vol. 66, no. 11, pp. 1258–1271, Nov. 2005.

[10] M. C. Chiu and Y. C. Chang, "Shape optimization of multi-chamber cross-flow mufflers by SA optimization," Appl. Acoust., vol. 67, no. 10, pp. 1005–1020, Oct. 2006.

[11] Y.-C. Chang, M.-C. Chiu, and M.-R. Wu, "Numerical assessment of automotive mufflers using FEM, neural networks, and a genetic algorithm," Arch. Acoust., vol. 33, no. 4, pp. 535–550, 2008.

[12] Suprayitno and M. Y. Pratama, "Multiobjective optimization of three-pass perforated muffler design for improved acoustic performance and reduced fluid pressure drop using genetic algorithms," Int. J. Automot. Mech. Eng., vol. 12, no. 1, pp. 4525–4540, 2025.

[13] J. Zhang et al., "Data-driven co-optimization of multiple structural parameters for the combustion chamber in a coke oven with a multi-stage air supply system," Processes, vol. 13, no. 12, p. 3818, 2025.

[14] V. D. Prajapati and A. J. Desai, "Design and analysis of automotive muffler," Int. J. Eng. Res. Technol., vol. 5, no. 5, pp. 216–222, May 2016.

[15] N. V. Kalyankar, Y. M. Khan, S. S. Tarawade, and S. R. Rathod, "A case study on exhaust muffler backpressure optimization," ESP J. Eng. Technol. Adv., vol. 3, no. 4, pp. 34–37, 2023.

[16] Xin and Y. Zhang, "Study on acoustic and fluid characteristics of exhaust muffler," WSEAS Trans. Acoust., vol. 13, pp. 1–8, 2017.

[17] X. Dong et al., "A CFD results based approach to investigating acoustic attenuation performance and pressure loss of car perforated tube silencers," Appl. Sci., vol. 8, no. 5, p. 751, May 2018.

[18] T. Yasuda, C. Wu, N. Nakagawa, and K. Nagamura, "Studies on an automobile muffler with the acoustic characteristic of low pass filter and Helmholtz resonator," Appl. Acoust., vol. 74, no. 1, pp. 49–57, Jan. 2013.

[19] H. Kepekçi and M. E. Ağca, "Numerical investigation of the thermal and acoustic effect of material variations on the exhaust muffler," Int. J. Automot. Eng. Technol., vol. 13, no. 1, pp. 33–44, 2024.

[20] A. D. Al-Rubaie and A. B. Al-Rubaie, "A novel design for muffler chambers by incorporating baffle plate," Appl. Acoust., vol. 197, Art. no. 108888, Feb. 2022. [Online]. Available: https://www.sciencedirect.com/science/article/pii/S0003682X21006232

[21] Z. Parlar, S. Ari, R. Yilmaz, E. Ozdemir, and A. Kahraman, "Development of muffler design and its validation," Appl. Acoust., vol. 180, Art. no. 108132, Sep. 2021. [Online]. Available: https://www.sciencedirect.com/science/article/pii/S0003682X21002632journals.sagepub

[22] N. L. Panwar and M. K. Shrimali, "A novel design for cruiser type motorcycle silencer based on micro perforated elements," arXiv:1908.06451, Aug. 2019. [Online]. Available: https://arxiv.org/abs/1908.06451

[23] I. R. Pani and P. C. Pati, "A review on acoustic performance analysis of reactive muffler," Mater. Today: Proc., vol. 63, pp. 613–622, 2022.

[24] A. Pal, R. R. Kulkarni, P. A. Makasare, and L. V. Kamble, "Design, analysis & optimization of muffler for four stroke petrol engine motorcycle," IJRASET, vol. 11, no. 5, pp. 1234–1245, May 2023.

[25] H. Hu, X. Zhang, and X. Li, "Large eddy simulation of a muffler with the high order spectral difference method," arXiv:1310.6860, Oct. 2013. [Online]. Available: https://arxiv.org/abs/1310.6860

[26] Z. Parlar, S. Ari, R. Yilmaz, E. Ozdemir, and A. Kahraman, "Acoustic and flow field analysis of a perforated muffler design," in Proc. IEEE Int. Conf. Acoust., Speech, Signal Process. (ICASSP), Vancouver, BC, Canada, 2013, pp. 4195–4199.