International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Analysis of Automobile Exhaust System

***Y. Vivek*[1], *D. Sai Siva Surya Prasad*[1], *R. Guna Sekhar*[1], *D. Aravind*[1], *Dr. V. Chitti Babu*[2]**

[1] UG Student, Department of ME, Aditya Institute of Technology and Management, Tekkali, Srikakulam, Andhra Pradesh – 532201
[2] Professor, Department of ME, Aditya Institute of Technology and Management, Tekkali, Srikakulam, Andhra Pradesh – 532201
Email: darlapudiprasad13@gmail.com

ABSTRACT:

The automobile exhaust system is a crucial system that expels combustion byproducts from the engine. It consists of a catalytic converter which reduces harmful emissions, a resonator that minimizes noise and vibration, and a muffler which further dampens exhaust noise before releasing it into the atmosphere. Back pressure, turbulence, and mass flow are essential parameters for the smooth and better performance of an exhaust system. The problems associated with these parameters are reduced engine efficiency, engine performance degradation, poor fuel economy, increased noise and vibration, and exhaust system overheating. A comprehensive analysis of automobile exhaust system has been the focus of the present study which include reduction of back pressure, and turbulence, while concurrently increasing mass flow rate. Analysis has been carried out by leveraging advanced design software CATIA V5 for modelling the exhaust system and computational fluid dynamics (CFD) software ANSYS 2020 R1 with the Fluid flow Fluent tool for fluid dynamic analysis. An exhaust system of a TATA INDICA 2018 car, has been taken as a reference and the effect of the change in dimensions of perforation diameter and the effect of the change in cross-section of the perforated hole of the internal tube and internal honeycomb structure of the catalytic convertor was investigated using ANSYS Fluent and the simulated data was then compared with the existing

Keywords: Exhaust System Optimization Tata Indica Resonator Muffler Pressure Drop Turbulence Mass Flow Rate CATIA V5 ANSYS FLUENT CFD Analysis

Introduction:

The automobile, a machine that has reshaped the landscape of human mobility, continues to captivate us. Once a novelty for the privileged few, it has transformed into the lifeblood of our modern world, weaving its way into the social, economic, and environmental fabric (Durbin & Taylor, 2007). From the bustling city streets to the open expanse of highways, automobiles have redefined how we travel, work, and connect.

This project delves into the intricate world of automobiles, with a particular focus on automobile exhaust system. By building upon existing research and exploring the latest advancements in the field, this project aims to CFX analysis on the TATA INDICA exhaust system to increase the mass flow rate and also reduce the back pressure, resonating frequency, and turbulence.

Examples: Car, jeep, bus, truck, scooter, etc.

Automobile exhaust system and main parts:

The internal combustion engine, a cornerstone of modern transportation, relies heavily on an efficient exhaust system to function optimally. This project focuses on the exhaust system of the Tata Indica, a popular diesel car in India, specifically targeting the catalytic converter, resonator, and muffler for performance improvement.

Essential Exhaust System:

Resonator (Kates & et al., 2007): Situated downstream of the catalytic converter, the resonator is designed to manipulate the exhaust gas pressure waves. Inadequately designed resonators can exacerbate unwanted resonant frequencies, leading to unpleasant noise and reduced engine power (Eriksson, 2006).

Muffler (Munjal & S. K. Ghosh, 2000): The muffler serves as the final stage of the exhaust system, attenuating noise generated by the engine and exhaust flow. However, conventional mufflers can restrict exhaust flow, reducing mass flow rate and impacting engine performance (Gatowski et al., 2014).

Problem Statement:

The existing Tata Indica diesel exhaust system might be experiencing issues with:

Backpressure: Excessive backpressure due to the catalytic converter can restrict exhaust flow, reducing engine power and fuel efficiency (Heywood, 1988).

Resonating Frequency: An improperly designed resonator can cause unwanted noise and power loss due to resonance phenomena (Eriksson, 2006).

Turbulence: Inefficient muffler design can lead to turbulence within the exhaust stream, hindering smooth flow and reducing mass flow rate (Gatowski et al., 2014).

Proposed Solution:

This project aims to address the aforementioned issues by optimizing the design of the catalytic converter, resonator, and muffler for the Tata Indica's exhaust system. By utilizing Computer-Aided Design (CAD) software like CATIA V5 and Computational Fluid Dynamics (CFD) analysis tools like ANSYS 2020 R1 with FLUENT, the project will explore design modifications to:

Optimize the resonator to mitigate unwanted resonant noises and ensure smooth exhaust flow.

Improve muffler design to reduce turbulence and enhance mass flow rate through the exhaust system.

The project will create and analyse multiple design iterations using CFD simulations, comparing the results with the existing exhaust system's rmance. This data-driven approach aims to achieve a balance between emission control, noise reduction, and improved engine performance for the Tata Indica.

Literature review:

Durr et al. (2005) conducted a comprehensive review of three-way catalysts used in passenger car applications. Their research highlighted the crucial role these converters play in reducing harmful pollutants like hydrocarbons, carbon monoxide, and nitrogen oxides from exhaust gases through a series of chemical reactions. However, the study also acknowledged the potential for traditional catalytic converters to create backpressure within the exhaust system, which can hinder engine performance by restricting exhaust flow (Durr et al., 2005). This finding emphasizes the need for optimizing catalytic converter design to achieve a balance between efficient pollutant conversion and minimal back pressure.

Literature review:

Eriksson's (2006) research, titled "Performance analysis of the intake and exhaust system of a high-performance four-stroke engine," offers valuable insights applicable to optimizing the Tata Indica's exhaust system. The study employed performance analysis techniques to investigate the interplay between the intake and exhaust systems in a high-performance engine. This analysis is particularly relevant to this project because it highlights the influence of exhaust system design on engine performance. Eriksson's key findings emphasize the importance of considering the exhaust system as an integral part of the entire engine operation. The study demonstrates that an improperly designed exhaust system can lead to resonance issues and inefficient gas flow.

Gatowski et al. (2014), titled "The influence of the exhaust muffler design on the performance of a spark- ignition engine," explores the impact of muffler design on engine performance. While their study focused on spark- ignition engines, the findings hold relevance for the Tata Indica's diesel engine as well. Gatowski et al.'s investigation identified potential drawbacks associated with conventional muffler designs. Their research suggests that mufflers can restrict exhaust flow and hinder mass flow rate.

Kates et al.'s (2007) research, titled "Experimental investigations of a Helmholtz resonator muffler for motorcycle exhaust noise reduction," explores the potential of Helmholtz resonators for mitigating noise in exhaust systems. While their study focused on motorcycles, the principles apply to the Tata Indica as well. This research highlights the concept of using a Helmholtz resonator, a strategically designed chamber within the muffler, to manipulate sound waves. Kates et al.'s experiment demonstrated that a properly designed resonator can improve noise reduction frequency and tuned unwanted frequencies.

Nilesh V. Kalyankar et al, muffler is essential for reducing internal combustion engine noise. By attaching silencers to the exhaust, they reduce noise pollution in the air before it escapes. There is a fine line to be drawn, though: engine lifespan is not a substitute for muffling. Manufacturers of engines establish a maximum backpressure limit because going over it can result in overheating, decreased performance, and higher fuel consumption. As such, accurate computations are essential. By carefully meshing the chromium-rich mild steel catia v5 model of the muffler in ansys fluent software and analyzing it, engineers can optimize the muffler design for maximum noise reduction while maintaining engine respiration.

Objectives:

To measure the physical configuration of every part of the existing exhaust system of the Tata Indica.

To model the observed physical configuration of the exhaust system using CATIA V5 to facilitate further analysis.

To perform computational fluid dynamic analysis using ANSYS 2020 R1 with its Fluent tool to simulate how air flows through the exhaust system, which will help us identify the causes of the problems like resistance, loud noises, and air flow issues.

To identify the parts of the exhaust system that are creating too much resistance, unwanted noise, and problems of air flow.

To investigate various designs of the resonator, potentially making it a straight pipe or incorporating improved baffles and perforated tubes to improve air flow efficiency within the resonator.

To explore design changes for the muffler, focusing on making the air flow path smoother by reducing bends and obstructions using straight-through designs or improved baffles and perforated tubes to manage noise levels.

To create three different digital models in CATIA V5, each one with specific design changes based on what we learned in the previous steps.

To analyse each modified exhaust system model using the ANSYS Fluent software to see how the design changes affect resistance, noise frequencies, and air flow.

To compare the results of the original exhaust system analysis with the simulations of the three modified models to identify the most effective design that could achieve the goal of improving the performance of the Tata Indica's exhaust system by increasing pressure drop reducing transmission losses, and increasing the mass flow rate.

Design analysis and simulation

This project is based on improving the efficiency of Diesel Engine Muffler. A muffler is a device for decreasing the amount of noise emitted by the exhaust of an internal combustion engine. This whole process is carried out by designing and using flow simulation through the muffler. Two muffler designs have been modelled and CFD gas flow simulation has been carried in both of them at various boundary conditions. The geometry of the model is prepared in CATIA V5 and analysis is carried out in ANSYS FLUENT using Computational Fluid Dynamics (CFD).

For Existing model Boundary Conditions:

1. Velocity at inlet is taken as 9.84 m/s.
2. Pressure at outlet is considered as Zero Pa.

Turbulence Intensity taken as 0.48%.

One of the major parameters for determination of muffler performance is transmission loss. It is the difference between the power incident at the inlet of a muffler and that transmitted downstream at the outlet and expressed in the unit of decibel. For better noise attenuation a 48 higher value of transmission loss is desired. Mathematically, transmission loss is represented as follows:

$$T.L = 10\log_{10}\left|\frac{S_i p_i{}^2}{S_o p_o{}^2}\right|$$

where, Si and So are the cross-sectional areas of the inlet and outlet of the muffler. pi and po are the acoustic pressure of the incident wave at the inlet of the muffler and transmitted wave at the outlet of the muffler respectively. In the present case where the inlet and outlet of the muffler are of equal cross-sectional area the above formula can be represented in modified form as follows:

$$T.L = 20\log_{10}\left|\frac{p_i}{p_o}\right|$$

For reference values following data has been considered for air in both the models which has been computed from inlet.

Area (m2)	0.0073
Density (kg/m^3)	0.696
Length(m)	2.25
Pressure (Pa)	1276.377
Temperature(k)	500
Velocity(m/s)	9.84
Viscosity(kg/ms)	2.7e-05
Ratio of specific Heats	1.4

Catia models:

Figure 1 Existing model

Figure 2 Instance model with Diameter 57mm

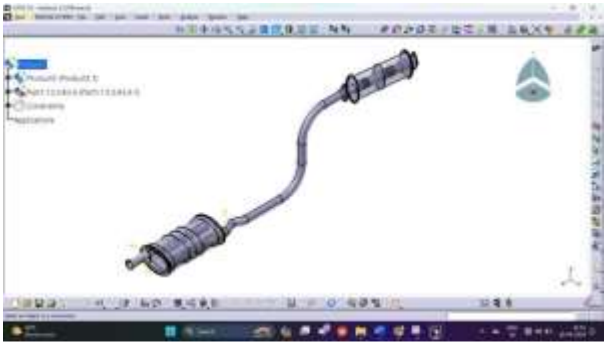

Figure 3 Instance model with Diameter 67mm

Figure 4 Instance model with Diameter 37mm

Import models into Ansys:

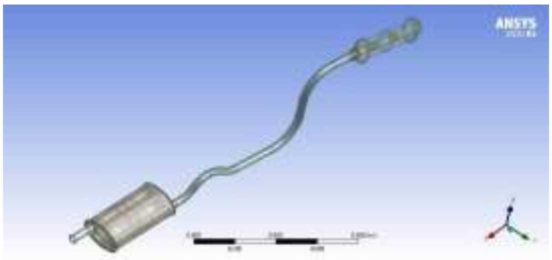

Figure 5 Imported Existing model from Catia in Ansys

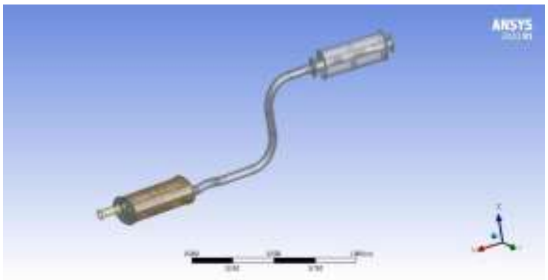

Figure 6 Imported Instance model 1 from Catia in Ansys

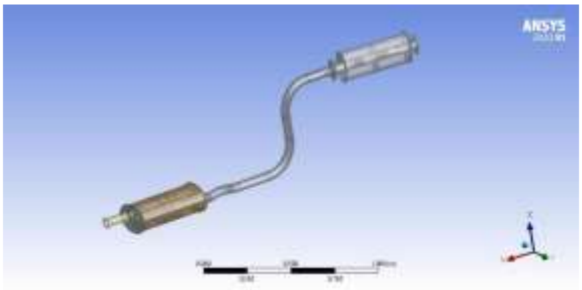

Figure 7 Imported Instance model 2 from Catia in Ansys

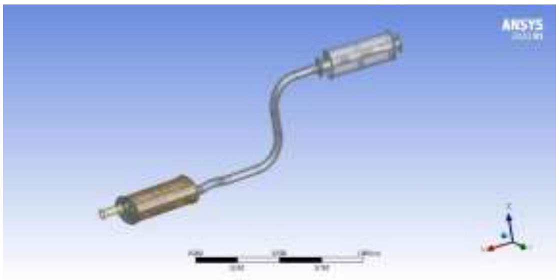

Figure 8 Imported Instance model 3 from Catia in Ansys

Meshing:

Ansys provides general purpose, high performance, automated, intelligent meshing software that produces the most appropriate mesh for accurate, efficient multi physics solutions from easy, automatic meshing to highly crafted mesh Smart defaults are built into the software to make meshing a painless and intuitive task, delivering the required resolution to capture solution gradients properly for dependable results. Ansys meshing solutions range from easy, automated meshing to highly crafted meshing Methods available cover the meshing spectrum of high order to linear elements and fast tetrahedral and polyhedral to high quality hexahedral and mosaic. Ansys meshing capabilities help reduce the amount of time and effort spent to get to accurate results. Since meshing typically consumes a significant portion of the time it takes to get simulation results, Ansys helps by making better and more automated meshing tools. Whether performing a structural, fluid, or electromagnetic simulation, Ansys can provide us with the most appropriate mesh for accurate and efficient solutions. The below image gives us a glimpse of Ansys meshing.

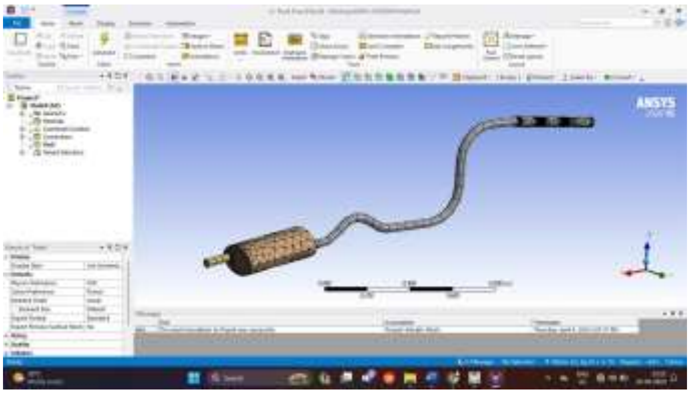

Figure 9 Meshing of Existing model

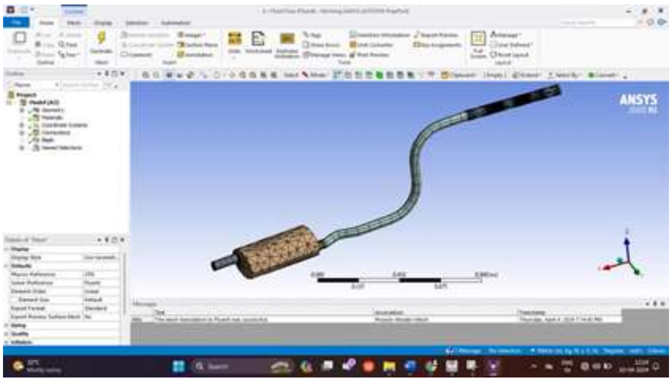

Figure 10 Meshing of Instance model 1

Figure 11 Meshing of Instance model 2

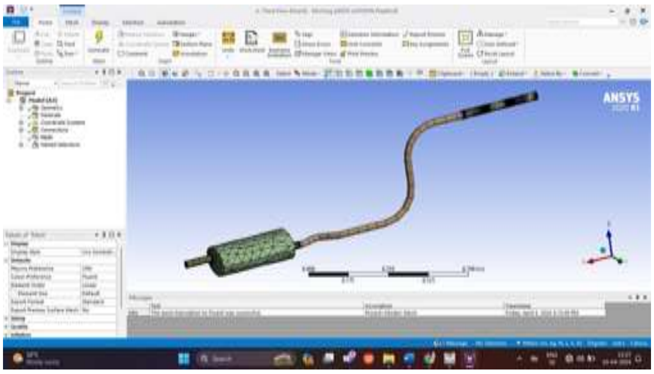

Figure 12 Meshing of Instance model 3

Results and discussions:

EXISTING MODEL:

Pressure Contour:

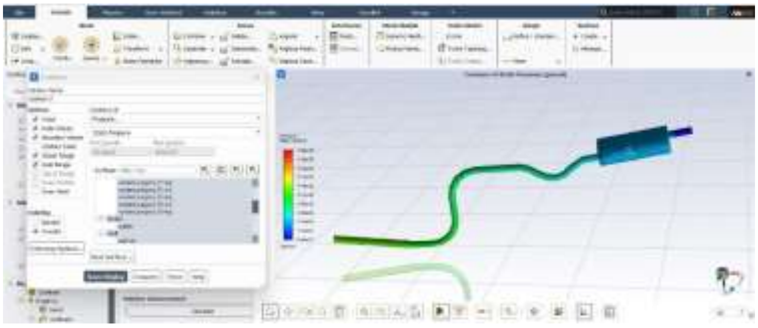

Figure 13 Total Pressure Contour

From Figure 13, it can be observed that the pressure is decreasing from inlet to outlet. Inlet pressure is 1276 pa and Outlet pressure is 34 pa.

Velocity Contour:

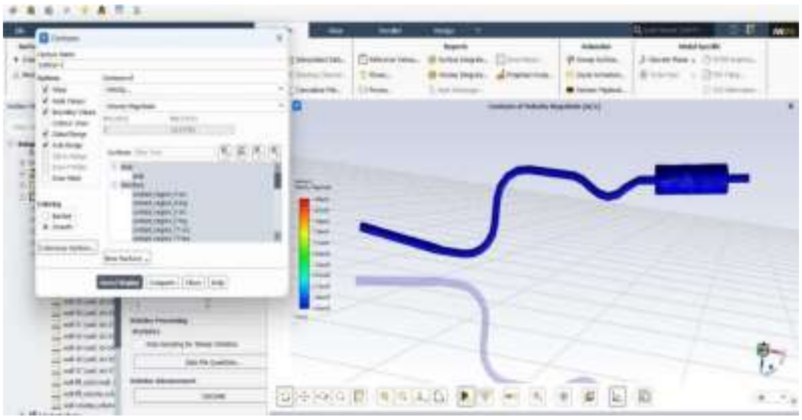

Figure 14 Velocity Contour

From Figure 14, it can be observed that the velocity is increasing from inlet to outlet. Inlet velocity is 9.84 m/s and Outlet velocity is 18.57 m/s.

Scaled Residuals:

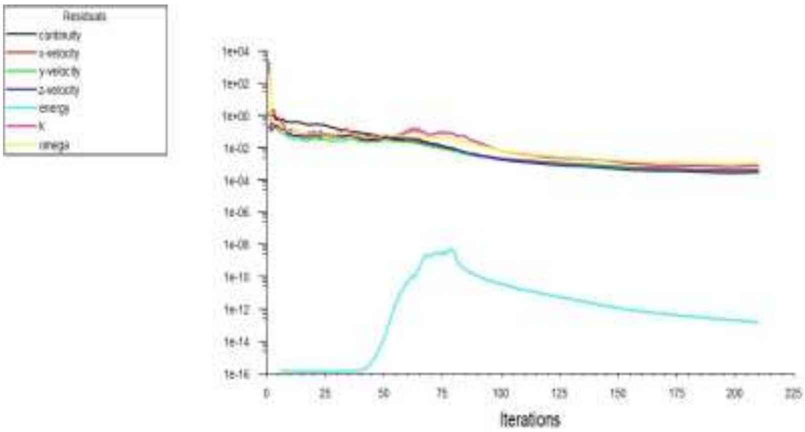

Figure 15 Scaled Residuals

Mass Flow Rate:

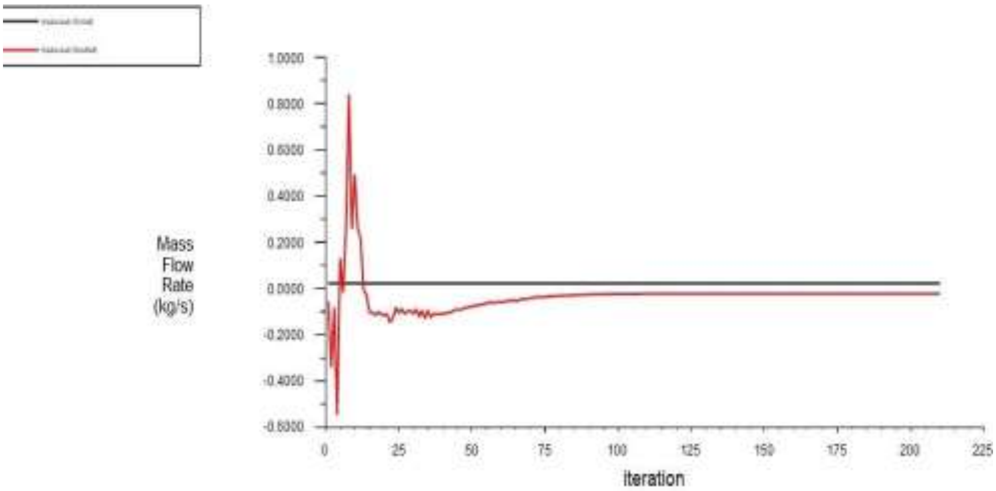

Figure 16 Mass Flow Rate

The obtained Mass Flow Rate at Inlet is 0.022658808 kg/s and at Outlet is -0.022634336 kg/s.

Stream Line:

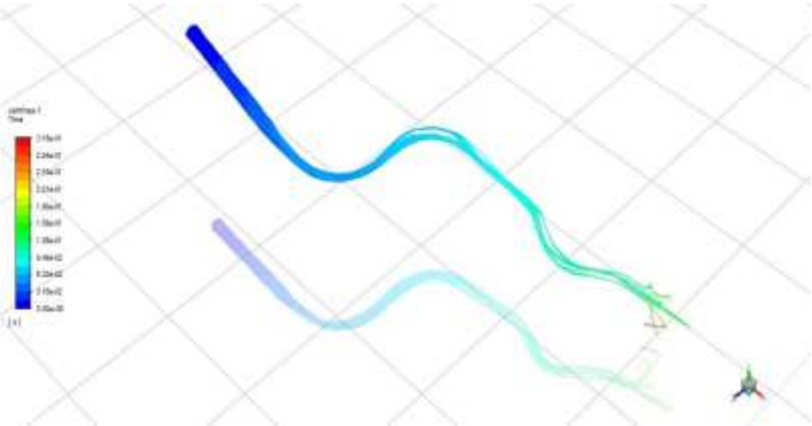

Figure 17 Stream Line

Turbulence Kinetic Energy:

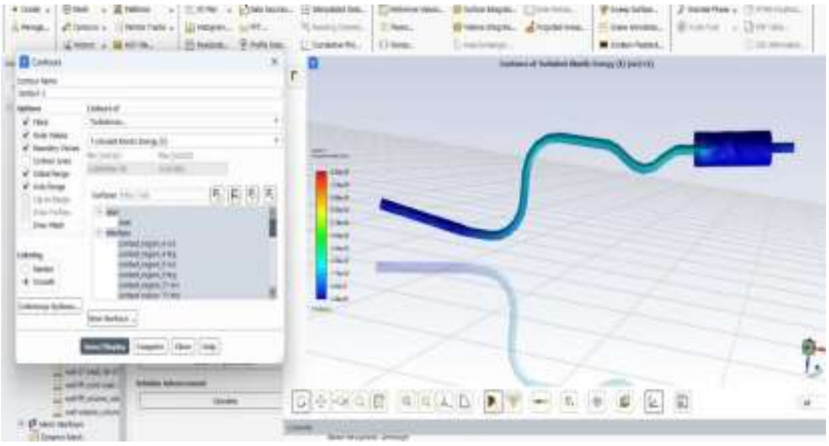

Figure 18 Turbulence Kinetic Energy

INSTANCE-1

Pressure Contour:

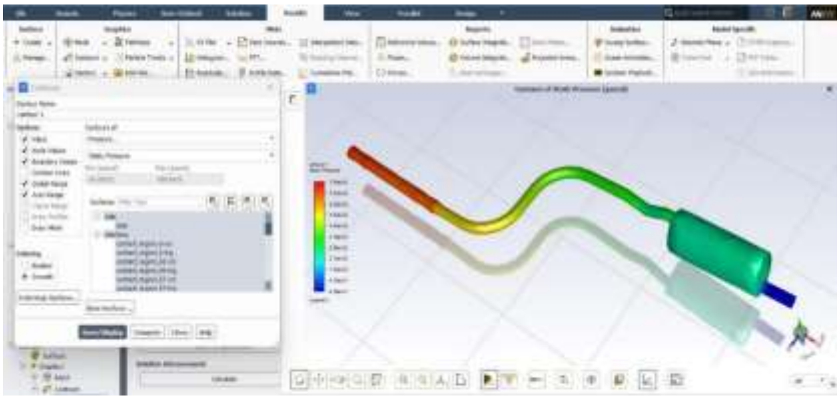

Figure 19 Pressure Contour

From Figure 19, it can be observed that the pressure is decreasing from inlet to outlet. Inlet pressure is 789.94 pa and Outlet pressure is 44.51 pa.

Scaled Residuals:

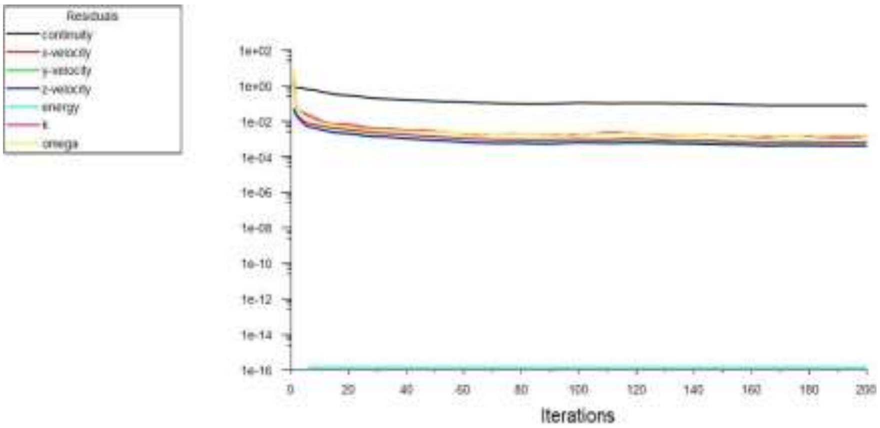

Figure 20 Scaled Residuals

Mass Flow Rate:

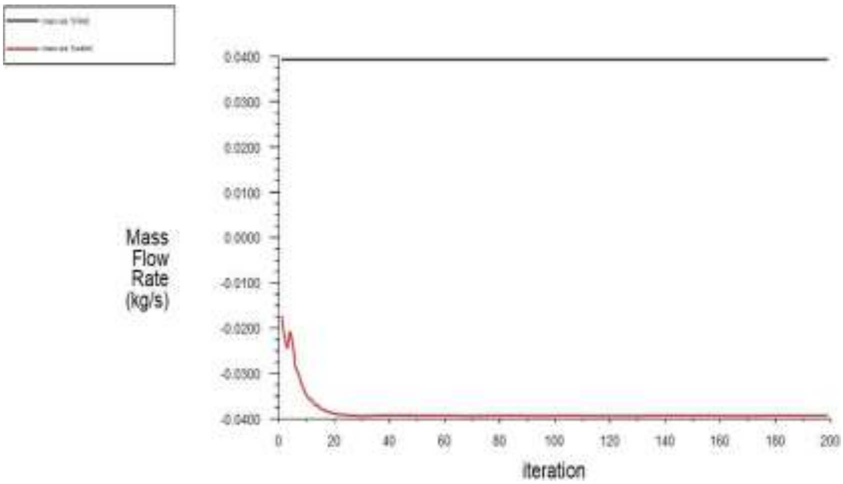

Figure 21 Mass Flow Rate.

The obtained Mass Flow Rate at Inlet is 0.039300939 kg/s and at Outlet is -0.039289993kg/s.

Stream Line:

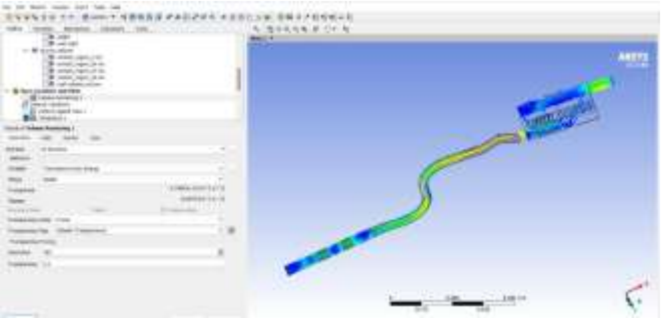

Figure 22 Stream Line

Turbulence Kinetic Energy

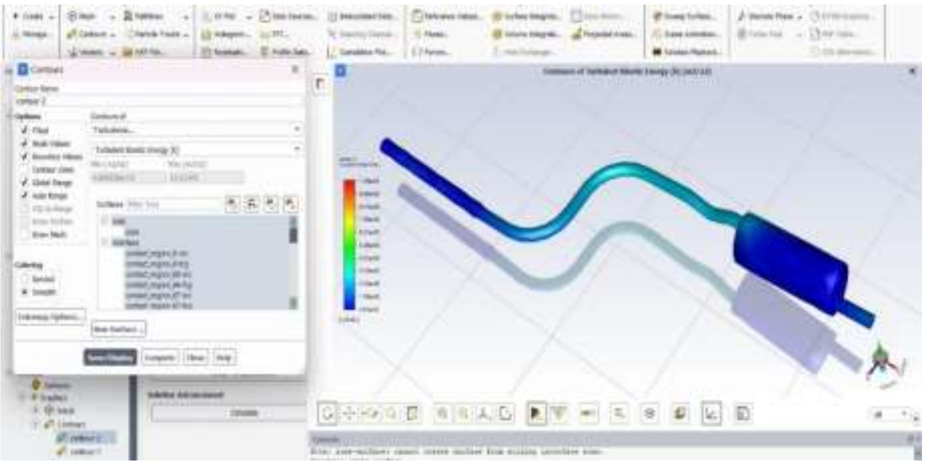

Figure 23 Turbulence Kinetic Energy

INSTANCE-2

Pressure Contour:

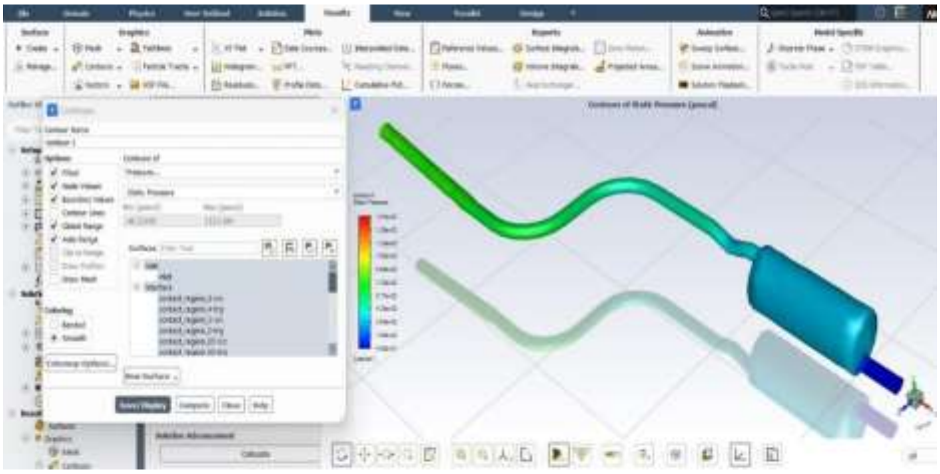

Figure 24 Pressure Contour

From Figure 24, it can be observed that the pressure is decreasing from inlet to outlet. Inlet pressure is 1513.58 pa and Outlet pressure is 68.50 pa.

Velocity Contour:

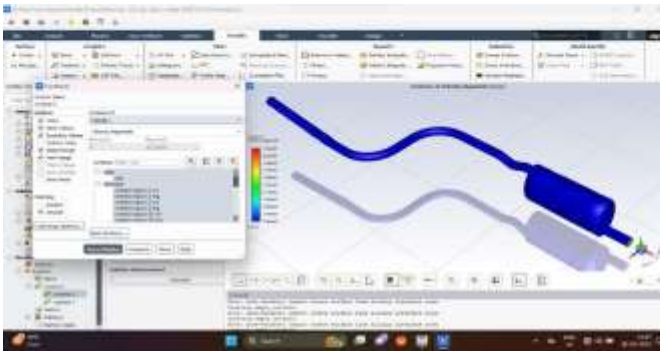

Figure 25 Velocity Contour

From the figure 25, it can be observed that the velocity is increasing from inlet to the Outlet. Inlet velocity is 14.03 m/s and Outlet velocity is 24.266 m/s.

Scaled Residuals:

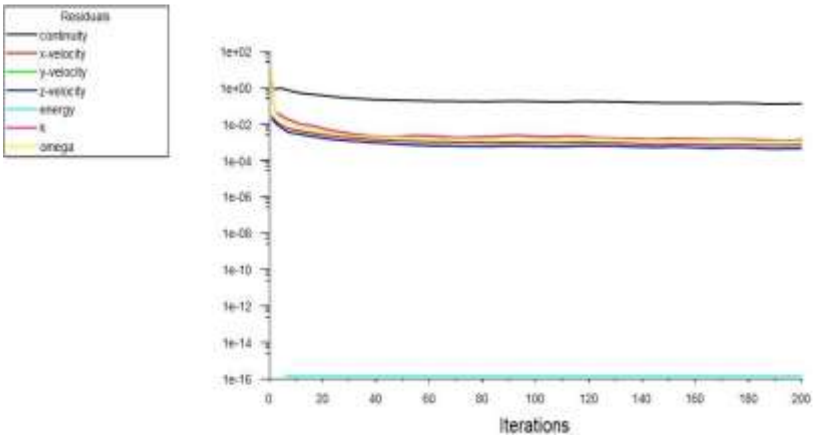

Figure 26 Scaled Residuals

Stream Line:

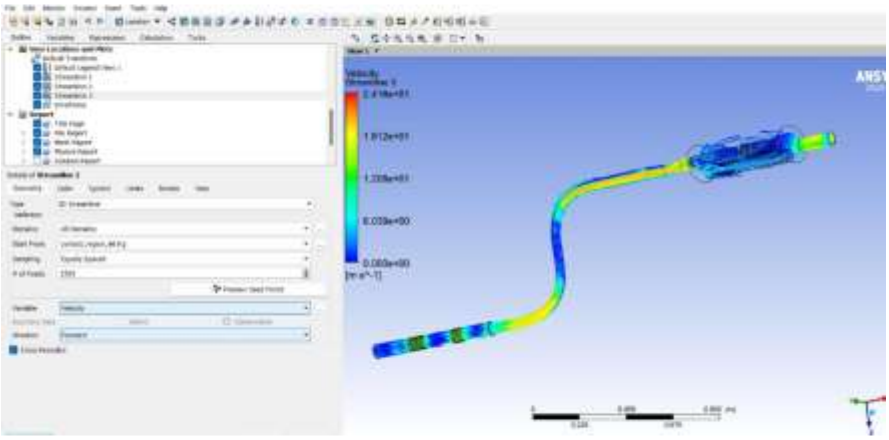

Figure 27 Stream Line

Turbulence Kinetic Energy:

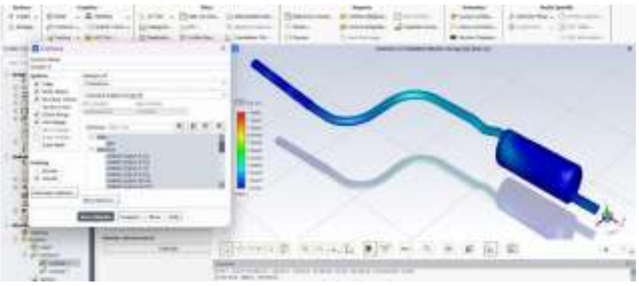

Figure 28 Turbulence Kinetic Energy

INSTANCE-3

Pressure Contour:

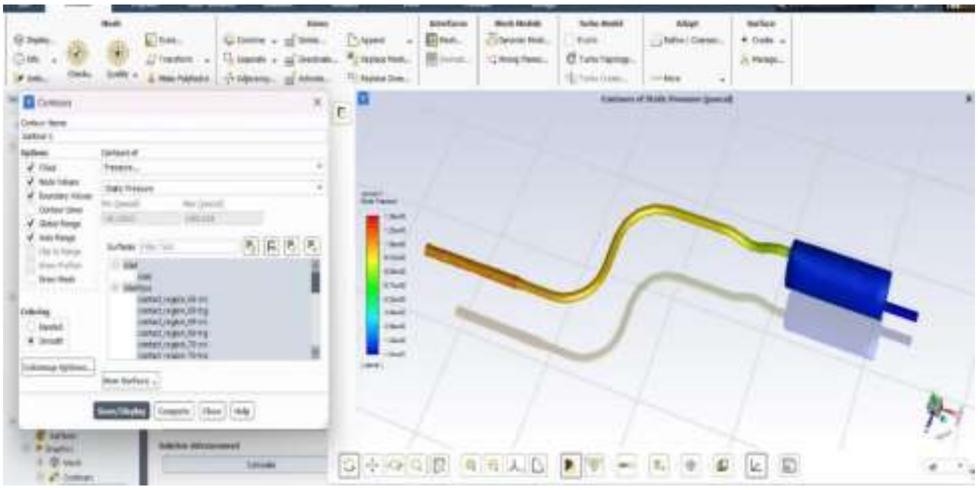

Figure 29 Pressure Contour

From Figure 29, it can be observed that the pressure is decreasing from inlet to outlet. Inlet pressure is 1360.218 pa and Outlet pressure is 21 pa.

Velocity Contour:

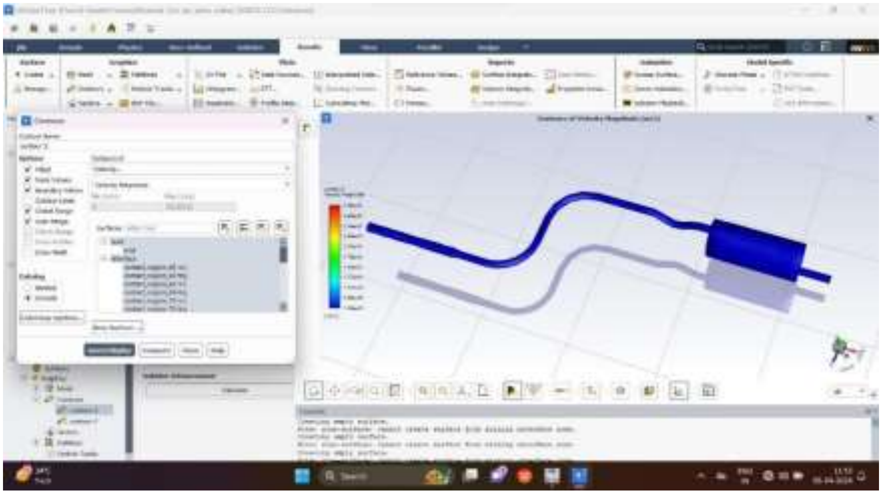

Figure 30 Velocity Contour

From the figure 30, it can be observed that the velocity is increasing from inlet to the Outlet. Inlet velocity is 7.749 m/s and Outlet velocity is 35.53 m/s.

Scaled Residuals:

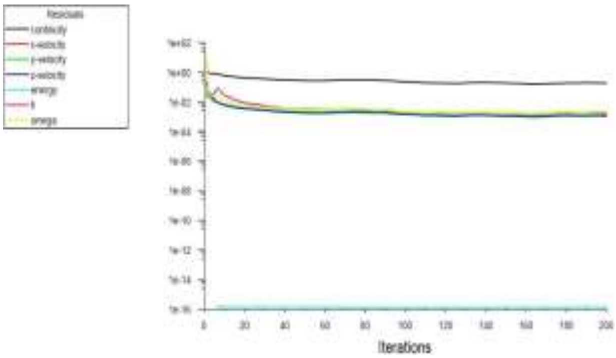

Figure 31 Scaled Residuals

Mass Flow Rate:

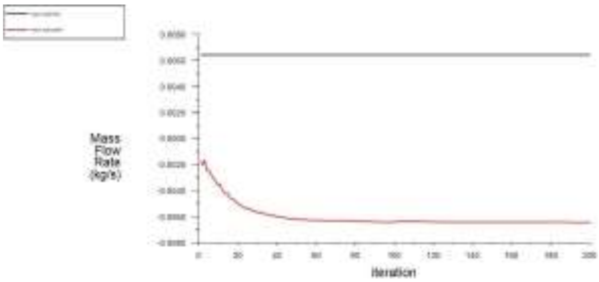

Figure 32 Mass Flow Rate

The obtained Mass Flow Rate at Inlet is 0.006422407 kg/s and at Outlet is -0.0064258338 kg/s.

Stream Line

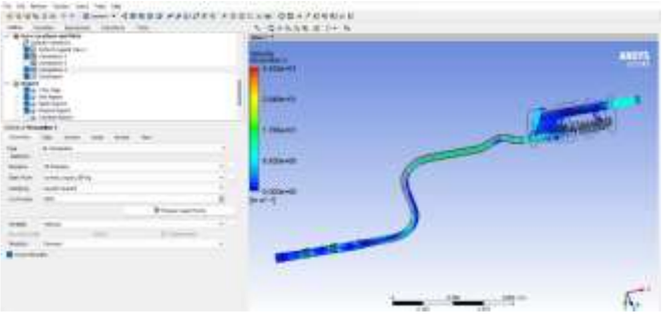

Figure 33 Stream Line

Turbulence Kinetic Energy

Figure 34 Turbulence Kinetic Energy

Table 5.1 CFD Analysis results on each models

Model no:	Vi(m/s)	Vo(m/s)	Pi(pa)	Po(pa)	Mass flow Rate (kg/s)	Pressure Drop (%)	Transmission Loss (TL)
Existing Model	9.84	18.58	1276.37	33.69	0.022658808	96	31.56
Instance 1	11.93	23.87	789.94	44.15	0.039300939	94	25.05
Instance 2	14.03	24.26	1513.58	68.50	0.064099896	97	26.88
Instance 3	7.75	35.53	1360.21	21.00	0.006422407	98	36.22

Conclusions and future scope:

This Project emphasizes the importance of the design methodology a practical approach from the concept design to proto manufacturing and validation of exhaust muffler. This design methodology will help designers in understanding the importance of each step of designing in detail from concept level to validation level. Although the practical approach has become an important tool in making muffler design more of art than science, the need for design verification will always be necessary at end of each step.

A Numerical analysis is carried out on acoustical and flow behaviour analysis of various muffler design and the following conclusions are made.

1. From the various instances considered for best possible muffler designs,1st instance is found to be a preferable choice at it is having least transmission losses.
2. The reduction in pressure of exhaust in existing model is 96%, whereas in instance 1 is 94%, in instance 2 is 97%, in instance 3 is 98%. Hence, we conclude that instance 3 is more efficient in reducing the exhaust pressure when compared to other models.
3. The Mass flow rate of exhaust in instance 2 is higher than the other models, which is 0.064099896 kg/s and found to be a preferable choice.
4. At last instance 2 given the better results than the other models among the various parameters like Mass flow Rate (kg/s), Pressure Drop (%), Transmission Loss (TL).

References:

- Durr, F., et al. (2005). Three-way catalysts for passenger car applications: A review. Applied Catalysis B: Environmental, 56(1-2), 263-

284. https://journals.sagepub.com/doi/10.1177/09 54407019859822

- Eriksson, L. (2006). Performance analysis of the intake and exhaust system of a high-performance four-stroke engine. Applied Mechanics Reviews, 59(2), 117-

132. https://link.springer.com/article/10.1007/s12 239-021-0067-6

- Gatowski, L., et al. (2014). The influence of the exhaust muffler design on the performance of a spark-ignition engine. Applied Mechanics and Materials, 505, 531-

536. https://search.proquest.com/openview/9ae94 9d82f11346bfbacf8ea212e2c6e/1?pq- origsite=gscholar&cbl=4998670

- Heywood, J. (1988). Internal combustion engine fundamentals. McGraw-Hill Book Company.
- Kates, M., et al. (2007). Experimental investigations of a Helmholtz resonator muffler for motorcycle exhaust noise reduction. Applied Acoustics, 68(10), 1222-

1234. https://journals.sagepub.com/doi/full/10.11 77/14613484231151273

Munjal, M. L., & Ghosh, S. K. (2000). Automotive engines. Tata McGraw-Hill Education