IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
Volume 2, Issue 2, February 2022
Impact Factor: 6.252

Comparative Study and CFD Simulation of Existing Automotive Muffler with Different Perforation

Chetana V. Ahire[1] and P. P. Palande[2]
Department of Mechanical Engineering
Matoshri College of Engineering and Research Centre, Eklahre, Nashik, Maharashtra, India
chetanapurkar95@gmail.com[1] and dpalande@gmail.com[2]

Abstract: *These paintings ambitions to examine the exhaust glide sample of one of the 3 Wheeler Exhaust systems to apprehend the exhaust traits, float tendencies, and exhaust dispersion at the outlet. This paintings makes use of Finite volume Computational Fluid Dynamics (CFD) evaluation that's executed using stable Works glide Simulation tool to examine three-wheeler Exhaust system version developed the usage of CAD. The version is created by measuring the actual dimensions of the Exhaust gadget components.*

Keywords: CFD, Muffler, CAD

I. INTRODUCTION

In inner combustion engines as a necessary factor for the exhaust segment mufflers are installed. it's far a tool to cut back the loudness of on the spot sound stress created by burning-warm exhaust gas exiting the engine at high velocity and also by a sequence of passages and chambers covered and resonating chambers harmonically tuned to reason adverse interference, wherein opposite sound waves cancel each different out to provide a noisy noise. Due to this phenomenon again, pressure is created within the muffler that is an unavoidable aspect impact of noise reduction that immediately declines the engine efficiency. Today, finite element and different numerical models are used for predicting the streamline glide, stress contour, the kinetic strength of exhaust particles in CFD simulation of components associated with fluid particularly air and gas. Handy however unrealistic values are often chosen because many homes of the bodily structure are very tough to measure. Because of using CFD software, it will become smooth to visualize the effects at the side of the indoors design with changed design systematically.

Damping factors or loss factors are required in FE modeling to predict deflections and stresses as a result of dynamic hundreds. Finite element modelers regularly have to select damping elements at random because damping is impossible to achieve correctly from material properties and geometry on my own. To reduce the vibration and temperature of warm gases popping out of engine muffler are used as a completely crucial part of an exhaust device. So, it will become a necessary parameter to discharge these exhaust gases with minimum restrict to the environment. Even as discharging again strain is created so because of which it consumes extra fuel and decline within the efficiency of an engine is determined.

II. PROBLEM STATEMENT

Comparison and CFD simulation of the present 3-wheeler muffler component with different perforations like 2mm, 4mm, and 6mm, to increase the mass flow in addition as decrease the pressure drop across the cross-sectional area.

2.1 Objective

1. Determining the mass flow rate, velocity, and temperature of the existing 3-wheeler vehicle muffler.
2. To extend the mass flow rate for the low pressure drop across its cross-sectional area.

2.2 Scope

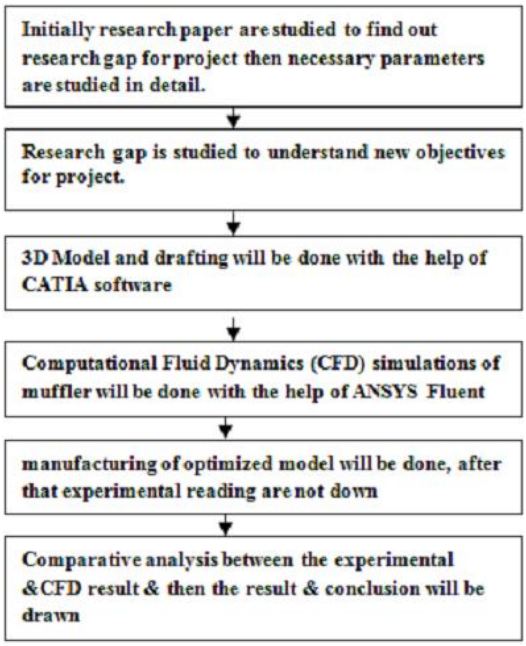
1. CFD simulation for velocity, mass flow rate, and temperature profile.
2. The similar study can be performed for various mufflers to validate experimental results.

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
Volume 2, Issue 2, February 2022
Impact Factor: 6.252

III. Result and Discussion

Comparison of through a CFD simulation of the existing 3-wheeler muffler part to extend the mass flow rate as well as decrease the pressure drop across the cross-sectional area.

3.1 Methodology



3.2 CFD Simulation Steps

1. CAD Design
2. Meshing (2D/ 3D Mesh)
3. Solver
4. Mesh Import
5. Boundary conditions
6. Material Properties
7. Calculations
8. Post Processing

3.3 Design of the System

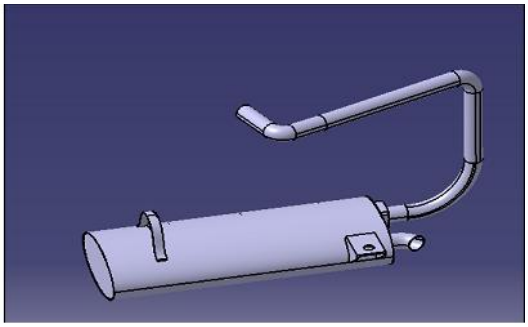


Figure 1: CATIA Model of the Muffler

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
Volume 2, Issue 2, February 2022
Impact Factor: 6.252

3.4 Analysis of System

An Analysis of existing system.

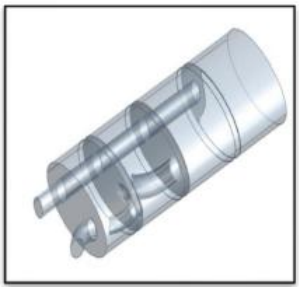


Figure.2: Geometry with extracted Volume in ANSYS

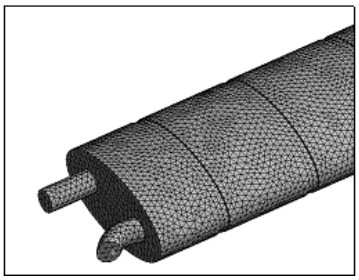


Figure 3: Details of the meshing

Details of "Body Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	6.0 mm

Statistics	
Nodes	29149
Elements	147199

Figure 4: Details of the meshing

3.5 Steps in CFD Simulation

1. In the general box, gravity is defined in the y direction as - 9.81 m/s2 (negative sign for downward direction)
2. In models, energy is kept on with viscous model as epsilon, realizable, scalable wall function
3. Material for gas is defined with properties
4. Inlet velocity of the gas is defined as 220 m/s as the mass flow rate is defined as 0.05 kg/s to convert mass flow rate into velocity following calculation are performed
5. Mass flow rate = discharge x density of gas
6. Discharge = area x velocity
7. Area of discharge is known as 451 mm2 (from geometry uploaded in ANSYS)
8. So unknown velocity parameter is calculated in the above equation

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
Volume 2, Issue 2, February 2022
Impact Factor: 6.252

9. Velocity = 46 m/s with gas temperature as 400 degree Celsius (standard references from the paper.
10. In the solution method second-order upwind are selected instead of the first order Hybrid initialization is selected in solution initialization Initial 1000 number of iterations are selected

MUFFLER WITH 2mm PERFORATION

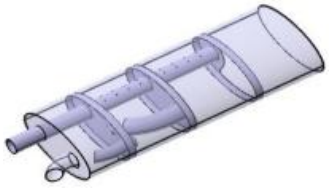


Figure 5: Perforation with 2mm hole

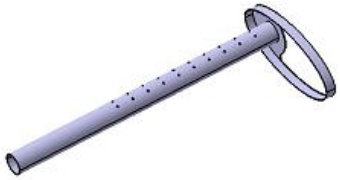


Figure 6: Inlet body

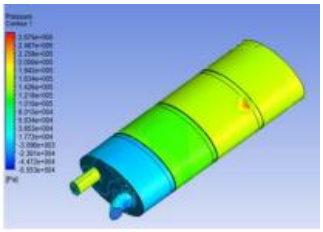


Figure 7: Pressure contour

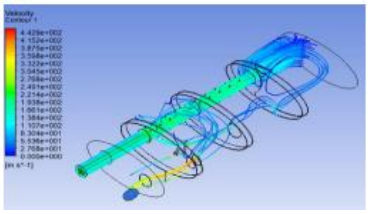


Figure 8: Velocity streamline

- It is observed from contour that modifying the existing muffler with perforation of diameter 2 mm pressure drop is observed around 277 mpa compared to existing 350 mpa

MUFFLER WITH 4mm PERFORATION

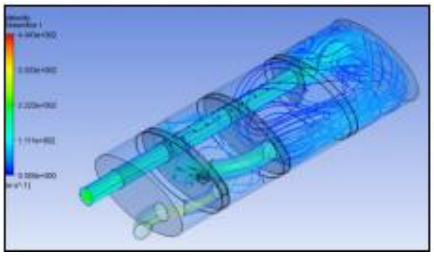


Figure 9: Velocity with streamline contour

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
Volume 2, Issue 2, February 2022
Impact Factor: 6.252

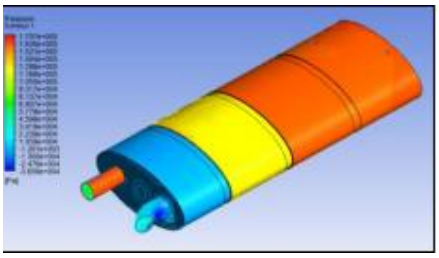


Figure 10: Pressure contour.

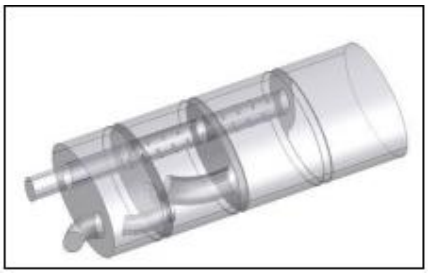


Figure 10: Modifying the existing Muffler

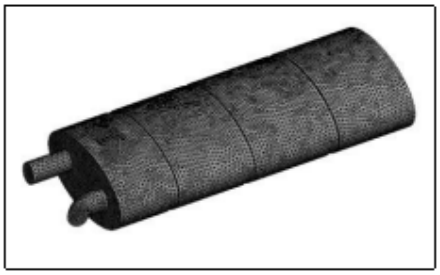


Figure 11: Meshing details

- Modifying the existing muffler to drop the pressure by introducing perforation of diameter 4 mm along inlet pipe
- It is observed from contour that modifying the existing muffler with perforation of diameter 4 mm pressure drop is observed around 177 mpa compared to existing 350 mpa

MUFFLER WITH 6mm PERFORATION

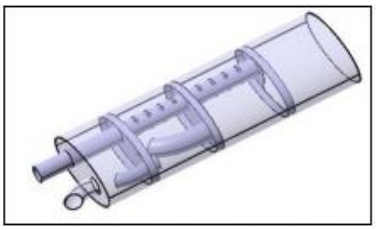


Figure 12: With 6mm hole

Figure 13: inlet body

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
Volume 2, Issue 2, February 2022
Impact Factor: 6.252

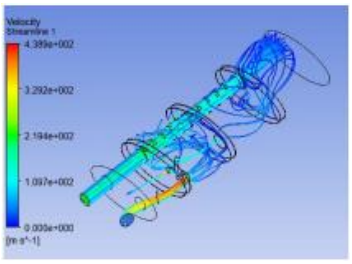


Figure 14: Pressure plot

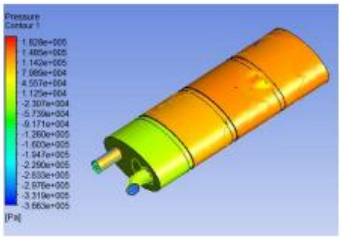


Figure 15: Velocity streamline

- It is observed from contour that modifying the existing muffler with perforation of diameter 4 mm pressure drop is observed around 182 mpa compared to existing 350 mpa.

IV. RESULT

Table 1: Comparison of Pressure for different perforation.

Sr. No.	DIAMETER OF PERFORATION	PRESSURE CONTOUR
1	Without Hole	350
2	2	277
3	4	177
4	6	182

V. CONCLUSION

- It is observed from contour that modifying the existing muffler with perforation of diameter 4 mm pressure drop is observed around the range of 174 to 177.70 Mpa compared to existing 350 Mpa.
- It is observed from experimental and CFD simulation results that for specified existing boundary condition velocity is observed around 13.1 and 16.58 m/s along with exit temperature around 46.6 and 45 degrees Celsius respectively.

REFERENCES

[1]. S.H. LEE, E.S. Kim, J.Y. Park, J. Choi, Numerical analysis of thermal deformation and residual stress in automotive muffler by MIG welding, Journal of Computational Design and Engineering(2018), doi:https://doi.org/10.1016/j.jcde.2018.05.001

[2]. M.H. Nazir a, Z.A. Khana, A. Saeeda, K. Stokes A predictive model for life assessment of automotive exhaust mufflers subject to internal corrosion failure due to exhaust gas condensation Engineering Failure Analysis 63 (2016) 43–60

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
Volume 2, Issue 2, February 2022
Impact Factor: 6.252

[3]. Takashi Yasuda Chaoqun Wua, Noritoshi Nakagawa Predictions and experimental studies of the tailpipe noise of an automotive muffler using a one dimensional CFD model applied Acoustics 71 (2010) 701–707

[4]. M.C. Li H. Zhang a, R.F. Huang " Effect of SO2 on oxidation of type 409 stainless steel and its implication on condensate corrosion in automotive muffler corrosion Science 80 (2014) 96–103

[5]. Jun Fu a,b, Minghui Xu a, Wei ZhengEffects of structural parameters on transmission loss of diesel engine muffler and analysis of prominent structural parameters applied Acoustics

[6]. 173 (2021) 107686

[7]. Paweł Bałona, Jacek Cieślika, Łukasz Halamac A method of manufacturing car mufflers by wrapping sheets, using the innovative forming device with the use of servo drive Procedia Manufacturing 50 (2020) 17–21

[8]. Ying-Chun Chang a, Min-Chie Chiu numerical analysis of circular straight mufflers equipped with three chambers at high order-modes Applied Acoustics 155 (2019) 167–179

[9]. Anand Gupta Optimization of emission characteristics of spark ignition engine with chambered straight muffler running in methanol.