CFD Analysis of Diesel Autorickshaw Exhaust System

Monil Yadav\textsuperscript{1} and Sanjiv M Sansgiri\textsuperscript{2}
\textsuperscript{1}Mechanical Department, Army Institute of Technology, Dighi Hills, Pune, Maharashtra, India
\textsuperscript{2}Head of Mechanical Department, Mechanical Department, Army Institute of Technology, Dighi Hills, Pune, Maharashtra, India

E-mail: monilyadav2407@gmail.com

Abstract. This work aims to study exhaust flow pattern of one of the Three Wheeler Exhaust System (Bajaj RE Diesel Auto Rickshaw) in order to understand the exhaust characteristics, implications to surroundings, flow tendencies and exhaust dispersion at outlet. This work utilizes Finite Volume Computational Fluid Dynamics (CFD) Analysis which is performed using Solid Works Flow Simulation tool to analyse Autorickshaw Exhaust System Model developed using Proe Creo. The Model is created by measuring actual dimensions of the Exhaust System Components, neglecting all hangar positions as their contribution to thermal behaviour is negligible.

1. Introduction

The branch of fluid mechanics which uses numerical analysis to solve and analyse problems or system that involve fluid flows which might be forced or natural and also to perform calculations required to simulate interaction of fluid with surfaces defined by boundary conditions of given system is called as Computational Fluid Dynamics. Fundamentals of all CFD problems is the Navier-Stokes equations, which define many single-phase (gases or liquids, but not both) fluid flows. The equations are simplified by removing terms describing viscous actions to obtain the Euler equations [1]. Further simplification is achieved by removing terms describing vorticity to obtain the full potential equations. For small perturbations in subsonic and supersonic flows (not transonic or hypersonic) these equations are linearized to obtain the linearized potential equations [2].

In history, these methods were first developed to solve the linearized potential equations. Two-dimensional methods, utilizing the conformal transformations of flow about a cylinder to flow about an airfoil were developed in the 1930s [1]. Current research yields software capable of improving the accuracy and speed of complex simulation scenarios like transonic or turbulent flows. Utilizing high-speed supercomputers, enhanced solutions can be achieved [3]. Initial Experimental validation of such software is performed using a wind tunnel and the final validation is performed using full-scale testing, e.g. flight tests [2].

The three wheelers are basic part of commercial and public transportation in underdeveloped and developing countries, Intermediate Public Transports (IPTs) generally referred as Autorickshaws in India form a major chunk of these three wheelers. These vehicles have small Horizontal Exhaust System which is suspended at rear below the Frame of IPT.

The exhaust system is composed of piping used to guide exhaust gases liberated by combustion away from combustion chamber inside an engine or stove. The system transfers flue gases from the
combustion site to location where exhaust is dispersed into surroundings after processing to reduce various pollutions and includes one or more exhaust pipes. Depending on system design, the flow path of exhaust gas may include:

- Cylinder Head
- Exhaust Manifold
- Catalytic Converter
- Turbocharger
- Exhaust Pipe
- Muffler or Silencer (UK/India), to reduce noise
- Tail Pipe

The following work is novel study of Autorickshaw Exhaust System aimed to analyze exhaust flow pattern within the system and dispersion pattern at outlet along with various thermal parameters.

1.1. Objective

- To determine and understand temperature, pressure & velocity profile within system.
- To study flue gas flow
- To understand dispersion pattern at outlet

The primal objective of this work is to develop a CFD Model of the Three Wheeler Exhaust System so as to perform a CFD Analysis using appropriate tool to obtain the flow patterns and value of various parameters, the next objective is study and understand flow pattern within exhaust system and identify flow concentration regions and the last objective is to study these flow pattern and thus predict the flow dispersion pattern at outlet.

2. Model

The CFD Model is generated by importing the CAD Model and defining all, fluid properties, material properties, heat transfer parameters, flow parameters and the boundary conditions.

2.1. CAD Model

The CAD model is developed by measuring actual dimensions of Bajaj Diesel 4-stroke RE Model Auto Rickshaw. The design is modular and contains three basic modules.

Figure 1. Exhaust Head & Muffler Flange Module.
The Exhaust Head Flange, Exhaust Pipe and One of the Flanges of Muffler are one module. The Second Module is Muffler main body itself while third module contains other end of Flange and Tail pipe.

![Figure 2. Muffler Flange and Tail Pipe Module.](image)

All three modules are secured together using bolted joints using Gaskets, Bolts and Flanges present on muffler side of each of the module.

![Figure 3. Muffler Module.](image)

2.2. Governing Equation

The governing equation for finite volume CFD method is as follows [4],

\[
\frac{\partial}{\partial t} \int \int Q \, dV + \int \int F \, dA = 0
\]  

(1)

Where, “Q” is Conservative Variable Vector, “F” is Flux Vector, “V” is Volume and “A” is Surface Area.

2.3. Thermophysical Properties of Fluid

| Property                  | Value       | Unit         |
|---------------------------|-------------|--------------|
| Boiling Point             | 194.686     | K            |
| Critical Density          | 10.63       | Mole\text{dm}^{-3} |
| Critical Pressure         | 7.38        | MPa          |
| Critical Temperature      | 304.13      | K            |
| Critical Volume           | 94.12       | cm\text{^3}\text{mol}^{-1} |
| Density                   | 40.8        | Mole\text{m}^{-3} |
| Heat capacity             | 37.35       | J\text{mol}^{-1}K^{-1} |
| Heat capacity             | 28.96       | J\text{mol}^{-1}K^{-1} |
| Thermal Conductivity      | 0.01663     | Wm\text{^{-1}}C^{-1} |
| Dynamic Viscosity         | 1.495       | cP           |
2.4. CFD Model
The CFD Model neglects all bolts and gaskets because though they are necessary for securing the modules together and generate an air tight assembly, they do not actually contribute to flow path.

Figure 4. Three Wheeler Exhaust System CFD Model.

The model at either of its opening is enclosed by lids with lids of “2mm” in order to create an air tight volume thus define a closed fluid domain.

Figure 5. Sectional View of CFD Model.

The Boundary Conditions are calculated from specification manual of above mentioned model and are as follows,

Figure 6. Boundary Condition of CFD Model.
3. Assumptions and Calculations

3.1. Assumptions

- Constant Discharge at Engine Exhaust Head
- The Speed of Vehicle remains constant at 40 Km/hr
- Exhaust is Discharged at Environmental Conditions
- Power Cycle is Ideal Diesel Cycle (Adiabatic Processes)
- The Exhaust is Entirely Composed of Carbon Dioxide
- Vehicle is Operating at Rated Parameter Values

3.2. Calculations

i) Given [6],
   i. Engine Type = 4-Stroke CI
   ii. Number of Cylinder = 1
   iii. Bore (D) = 86mm
   iv. Stroke (L) = 77mm
   v. Displaced Volume (Vd) = 447.3cc
   vi. Compression Ratio (r) = 24
   vii. Max Power (W) = 5880W
   viii. Rated Speed (N) = 3000RPM
   ix. Mechanical Efficiency (η_m) = 88%
   x. Thermal Efficiency (η_t) = 45%
   xi. Air Temperature (T_0) = 27ºC
   xii. Exhaust Temperature at Outlet(T_2) = 53ºC

ii) Time for Opening of Exhaust Valve (t),
    For 4-Stroke Engine, Revolutions per Cycle “n” is “2”,
    \[ t = \frac{n}{N} = \frac{2}{50} = 0.04\text{sec} \] (2)

    Inflow Rate (Q),
    \[ Q = \frac{V}{t} = \frac{0.0004473}{0.04} = 0.0111825\text{m}^3 \] (3)

iii) Volumetric Efficiency (\(\eta_v\)),
    Swept Volume (V_s) is given by,
    \[ V_s = \frac{n}{4} D^2L = \frac{n}{4} \times 0.086^2 \times 0.077 = 0.0004472\text{m}^3 = 447.2\text{cc} \] (4)

    Clearance Volume (V_c) is given by,
    \[ \frac{V}{V_c} = r = 24 = \frac{V_s + V_c}{V_c} = 1 + \frac{V_s}{V_c} = 1 + \frac{0.0004472}{V_c} \] (5)
    \[ V_c = 0.0000179\text{m}^3 = 17.9\text{cc} \]
\[ V = V_s + V_c = 0.0004651 \text{m}^3 = 465.1 \text{cc} \] (6)

\[ \eta_v = \frac{V_d}{V} = \frac{0.0004473}{0.0004651} = 0.9617 = 96.17\% \] (7)

iv) Overall Efficiency (\(\eta\)),

\[ \eta = \eta_t \eta_m \eta_v = 0.45 \times 0.88 \times 0.9617 = 0.3808 = 38.08\% \] (8)

v) Heat Supplied (\(Q_{\text{in}}\)),

\[ Q_{\text{in}} = \frac{W}{\eta} = \frac{5880}{0.3808} = 15439.3702 \text{W} \] (9)

vi) Heat Supplied (\(Q_{\text{out}}\)),

\[ Q_{\text{out}} = Q_{\text{in}} - W = 15439.3702 - 5880 = 9559.3702 \text{W} \] (10)

vii) Temperature at End of Compression (\(T_c\)),

\[ \frac{T_c}{T_1} = \left( \frac{P_c}{P_1} \right)^{\frac{(\gamma - 1)}{\gamma}} = (24)^{\frac{(1.4 - 1)}{1.4}} = \frac{T_c}{300} \] (11)

\[ T_c = 743.819 \text{K} = 470.819\degree \text{C} \]

viii) Temperature at End of Isobaric Heat Addition (\(T_m\)),

\[ T_m = T_c + \frac{Q_{\text{out}}}{\rho Q C_p} = 743.819 + \frac{9559.3702}{1.225 \times 0.0111825 \times 1005} \] (12)

\[ T_m = 1438.184 \text{K} = 1165.184\degree \text{C} \]

ix) Temperature at End of Expansion (\(T_2\)),

\[ \frac{T_m}{T_2} = \left( \frac{P_m}{P_2} \right)^{\frac{(\gamma - 1)}{\gamma}} = (24)^{\frac{(1.4 - 1)}{1.4}} = \frac{1438.184}{T_2} \] (13)

\[ T_2 = 580.054 \text{K} = 307.054\degree \text{C} \]
3.3. Data Table

Table 2. Input Data Table.

| No | Parameter                  | Values          | Type          |
|----|----------------------------|-----------------|---------------|
| 1  | Engine Type                | 4 Stroke - CI   | Reference⁶    |
| 2  | Cylinders                  | 1               | Reference⁶    |
| 3  | Bore                       | 86mm            | Reference⁶    |
| 4  | Stroke                     | 77mm            | Reference⁶    |
| 5  | Swept Volume               | 447.2cc         | Calculated    |
| 6  | Clearance Volume           | 17.9cc          | Calculated    |
| 7  | Cylinder Volume            | 465.1cc         | Calculated    |
| 8  | Volume Displaced           | 447.3cc         | Reference⁶    |
| 9  | Compression Ratio          | 24              | Reference⁶    |
| 10 | Max Power                  | 5880W           | Reference⁶    |
| 11 | Rated Speed                | 3000RPM         | Reference⁶    |
| 12 | Volumetric Efficiency      | 96.17%          | Calculated    |
| 13 | Mechanical Efficiency      | 88%             | Reference⁶    |
| 14 | Thermal Efficiency         | 45%             | Reference⁶    |
| 15 | Overall Efficiency         | 38.08%          | Calculated    |
| 16 | Heat Supplied              | 15439.3702W     | Calculated    |
| 17 | Heat Rejected              | 9559.3702W      | Calculated    |
| 18 | Air Temperature            | 27°C            | Reference⁶    |
| 19 | Exhaust Temperature at Engine Outlet | 307.054°C | Calculated |
| 20 | Exhaust Flow Rate at Engine Outlet | 0.0111825m³/s | Calculated |
| 21 | Exhaust Pressure at Engine Outlet | 101325Pa | Calculated |
| 22 | Exhaust Temperature at System Outlet | 40 - 60°C | Reference⁶ |
| 23 | Mass Inflow Rate           | 0.013698Kgm⁻³  | Calculated    |

4. Results Discussion

4.1. Flue Gas Temperature within Exhaust System

![Figure 7. Temperature Plot on Flow Trajectory.](image)

![Figure 8. Temperature Section Plot (Front).](image)

![Figure 9. Temperature Section Plot (Top).](image)
The initial temperature of exhaust as it enters the Exhaust System is 580.054K, the temperature of exhaust gradually decreases due to convection mode of heat transfer as it travels through muffler and becomes almost constant about 330K which is 57°C & as per the technical data it should be between 40 to 60°C and as the analysis is performed for rated speed the temperature value is close to upper limit, tough as visible in above diagram, localized heating may occur in region of stagnation.

4.2. Flue Gas Pressure within Exhaust System

![Pressure Plot on Flow Trajectory](image1)

![Pressure Section Plot (Front)](image2)

![Pressure Section Plot (Top)](image3)

The pressure within Exhaust System initially increases due to resistance in flow path as it enters exhaust pipe & flows through it, followed by which it slowly decreases as it passes through muffler tough there may be local increase in certain regions due to attenuation requirements & then it rapidly decreases as it passes through tailpipe.

4.3. Flue Gas Velocity within Exhaust System

![Velocity Plot on Flow Trajectory](image4)

![Velocity Section Plot (Front)](image5)

![Velocity Section Plot (Top)](image6)
The velocity of flue gases is initially high as they are pushed into exhaust system by the piston but it gradually decreases due to system resistance of exhaust system in which major contributor is muffler, though the flow velocity increases again as it enters final chamber of muffler where there is a pressure rise due to stagnation and as pressure head drops to ground state the velocity head increases by complementary amount as datum head is constant, hence at high speed flow velocity as high as 4m/s are possible opening of tail pipe to environment.

4.4. Validation
The Exhaust temperature as per technical specification is 60°C and as per analysis is 57° thus validated.

5. Conclusion
- The exhaust gases enter exhaust pipe at high temperature & velocity.
- The rise in pressure is mostly due resistance offered by the flow path within muffler resulting in back pressure.
- The temperature drop of about 270°C takes place within the muffler which makes it liable to heat hazard hence must be appropriately shielded.
- The muffler generates several flow vortex within its resonant chambers in order attenuate the fluid velocity by utilizing appropriate flow passage.
- The perforation within pipes allow gas to escape into chamber along with venture effect generating turbulence thus increasing attenuation hence reducing noise.
- The exhaust pressure locally increases in certain region of flow especially at regions of sudden change of flow.
- The muffler pressure is relatively on higher side due 4-chamber design though it helps in reducing noise.
- The exhaust flow velocity rapidly increases as exhaust enters tail pipe due to rapid drop in pressure head as it drop to ground state. Which is in accordance with fact that these exhaust system model more susceptible to breaking of tail pipe with time due to erosion & vibration.
- The temperature gradient of 13°C to 33°C ensures that exhaust gases rise and disperse once liberated from tail pipe into environment.
- The high velocity at exhaust system outlet causes significant whirling of dust as observed in most of such vehicles.

References
[1] Virginia R Stovin and Adrian J Saul Wal, A Computational Fluid Dynamics (CFD) Particle Tracking Approach To Efficiency Prediction Water Science & Technology, 1998, 37, pp.288-293
[2] Richard J Blint and Daniel C Haworth, Modeling Oxidation Of Exhaust Gases Diluted By Air Under Exhaust Manifold Conditions Proc. of The Combustion Institute, Warren, MI 48090-9055, USA, 2000, 28, pp.2451-2457
[3] Xu et al., CFD Analysis of a Gasoline Engine Exhaust Pipe, J Appl Mech Eng 2016, pp.2168-2176
[4] Cedric Louis, Yao Liu, Simon Martinet, Dilution effects on ultrafine particle emissions from Euro 5 and Euro 6 diesel and gasoline vehicles, Atmospheric Environment, 2017, 169, pp.80-88
[5] C. P. Ariara Guhana, G. Arthanarees Warana, K. N. Varadarajan, S. Krishnan, Exhaust System Muffler Volume Optimization of Light Commercial Vehicle Using CFD Simulation, Proc. of Materials Today, IMME17, National Institute of Technology, Tiruchirappalli-620015, India, 2018, 5, pp.8471-8479
[6] Bajaj Diesel IPT, Compact Diesel Brochure, Bajaj Auto Ltd, India, 2013, pp.4-5