NUMERICAL ANALYSIS OF AN INTEGRATED EXHAUST DUCT SYSTEM

Ra. Aravind[1] | S. Kumaravel[2] | M. Kavin[3] | P.Raja[4]

[1](Dept of Mechanical Engg, Asst. Professor,Kongunadu College of Engg &Tech,Trichy, India,aravindra25@gmail.com)
[2](Dept of Mechanical Engg, Asst. Professor,Kongunadu College of Engg &Tech, Trichy, India,kumaravelsrk@gmail.com)
[3](Dept of Mechanical Engg, Asst. Professor,Kongunadu College of Engg&Tech, Trichy, India,kavinmahu@gmail.com)
[4](Dept of Mechanical Engg, Asst. Professor,Kongunadu College of Engg &Tech, Trichy, India,rajapme@gmail.com)

Abstract—*In general, curved 90° elbow bends of air-conditioning, ventilation duct systems and the mill-duct conveying systems shows less energy losses and higher production cost compared to right-angled sharp 90° elbow bends. Formation of a separation zone in sharp elbow bends causes energy loss, which appears just after the corner, questionable and difficult to make numerical prediction of the flow. By using the computational fluid dynamic software's Gambit & Fluent, geometry of the duct and flow pattern inside the duct can be modeled and visualized. Due to high pressure generation in every inlet port of the duct at full load condition, the flow should be highly streamlined but it is not created because of the poor geometry of the duct.*

Keywords—Integrated duct; Flue gas; CFD; Flow pattern; Energy loss

1. INTRODUCTION

Ducts are commonly used in heating, ventilation, and air conditioning (HVAC) to deliver and remove air. Air ducts is one of the most important method of ensuring acceptable indoor air quality as well as thermal comfort [1]. Duct design normally comprises of planning, sizing, optimizing, detailing and finding the pressure losses through a duct system.

Ducts used for carrying air are of round, square or rectangular shape. However, round duct is most efficient based on the volume of air handled per perimeter distance [2] and requires only very less material. Normally, building constructions i.e., ceilings, walls etc. fits square or rectangular duct and is much easier to install between joints and studs [3].

Due to incomplete combustion of carbonaceous fuel, internal combustion engines such as reciprocating internal combustion engines produce air pollution emissions. The main derivatives of the process is particulate matter (PM) like carbon dioxide (CO_2), water and some soot. The inhaling effects of particulate matter have been studied in humans and animals which causes asthma, lung cancer, cardiovascular issues and premature death [4].

Computational Fluid Dynamics (CFD) generates flow simulations with the help of computers and involves the solution of governing laws of fluid dynamics numerically. The complex sets of partial differential equations are solved on in geometrical domain divided into small volumes, commonly known as a mesh [5].

2. SPECIFICATIONS AND BOUNDARY CONDITIONS

A.2D Model of the Duct System

The 2D model of duct system is represented in Fig. 1 as follows.

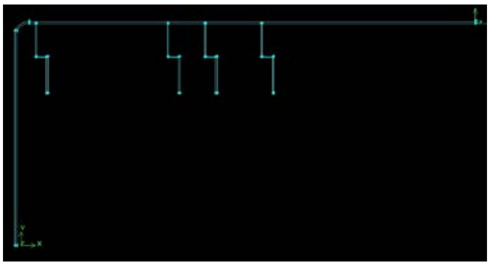

Fig. 1 2D model

B.Duct Specification

- Diameter of the main duct:0.1 m
- Diameter of the intersecting pipes:0.04 m
- Total length of the duct in x-direction:21.64 m
- Total length of the duct in y-direction:10.8 m

The exhaust gas velocity for different engines is listed in Table 1.

TABLE I. EXHAUST GAS VELOCITY FOR DIFFERENT ENGINES

Load (%)		**Velocity (m/s)**	**Mass Flow Rate x 10^{-3} (kg/s)**	**Temp (K)**
25	Egas-1	1.25	1.82	438
	Egas-2	1.25	1.82	438
	Egas-3	1.25	1.82	438
	Egas-4	2.5	3.65	438
100	Egas-1	3.125	4.56	473
	Egas-2	3.75	5.47	473
	Egas-3	3.75	5.47	523
	Egas-4	5	7.29	503

C. Methodology

Study the system → Geometry of the system generated by GAMBIT → Measure the exhaust gas velocity using HOT WIRE ANEMOMETER → Simulate the geometry using FLUENT.

D. Boundary Condition and Governing Equation

The fluid is considered incompressible with constant properties. As an approximation, the properties of air can be used for diesel exhaust gas calculations. The error associated with neglecting the combustion products is usually no more than about 2%. The properties of air is given below

Density (ρ) = 1.616 kg/m3
Thermal conductivity (k) = 0.0263 w/m-k
Viscosity (μ) = 1.85×10-5 pa-s
Specific heat (cp) = 1007 j/kg-k

General boundary condition like mass flow inlet and outflow is considered for every inlet point of the duct and for common exit. Turbulence k-ξ model is used for calculations.

3. RESULT AND DISCUSSION

The numerical analysis conducted by varying the velocity of the flue gas for different loading condition. The analysis conducted for 25% and 100% loading condition on every engine.

A. Case-1 (Load – 25%)

Fig. 2 shows that the flow reversal occurs at the Egas-3 port because of the low pressure region present between the ports of Egas-2 and Egas-3. Turbulence also occurred at the position of Egas-3.

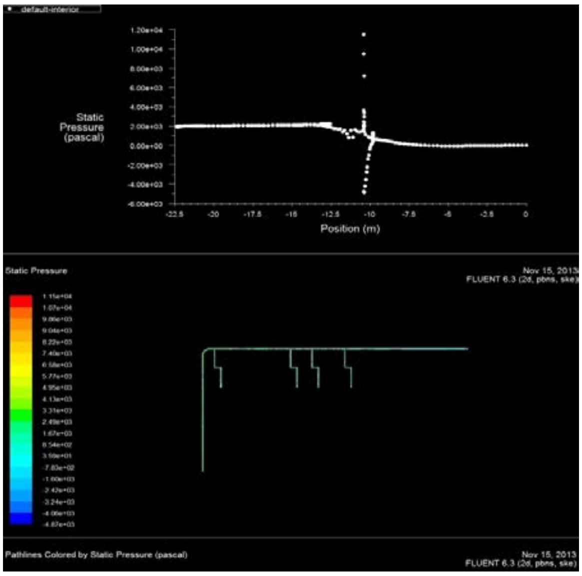

Fig. 2 Path lines and XY plot of static pressure for 25% load

It also shows that the high pressure region occurs at the Egas-4 position. Because of the high velocity region at Egas-4, the velocity of Egas-2 and Egas-3 decreased.

B. Case-2 (Load – 100%)

Fig.3 shows that the reversal flow occurs at the position of Egas-3 because of the turbulence generated at this point.

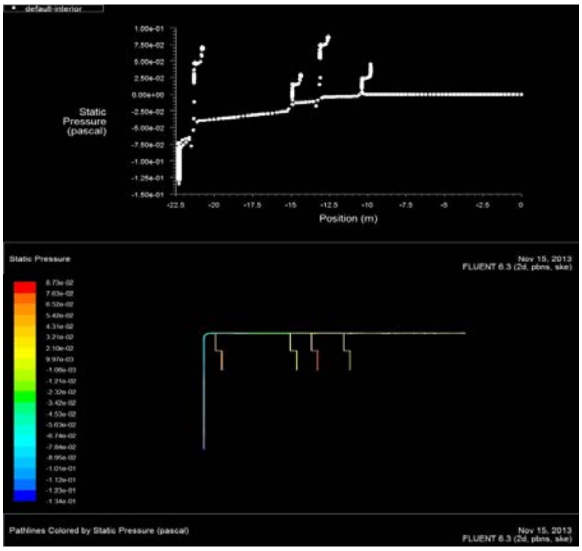

Fig. 3 Path lines and XY plot of static pressure for 100% load

It also shows that the velocity is high at Egas-1 and it decreases for Egas-2 and Egas-3. Because of that, the flow moves steady and no reversal occurred. Then, the velocity is high for Egas-4 because of flow reversal occurred at Egas-3.

4. CONCLUSION

In this paper, two-dimensional numerical simulations are employed to investigate the flow pattern in a long circular duct with multiple T-joints. The duct is constructed without design consideration. Because of this geometry, pressure loss and high turbulence are generated. The 2D model of the duct was generated by GAMBIT and the local loss coefficient can be identified using FLUENT. The numerical analysis is conducted for two different cases. The result shows that the swirling motion and the pressure loss are high for low load condition. It also shows that more streamlined flow is achieved in full load condition and at the same time it also creates some amount of turbulence and pressure variation. Due to high pressure generation in every inlet port of the duct at full load condition the flow should be highly streamlined but it is not created because of the poor geometry of the duct. So, the design has to be changed

REFERENCES

[1] Avvari R and Jayanti S, “Heuristic shape optimization of gas ducting in process and power plants”, Chemical Engineering Research and Design, vol. 91, pp. 999-1008, 2013.

[2] Aydin C and Ozerdem B, “Air leakage measurement and analysis in duct systems”, Energy and Buildings, vol. 38, pp. 207-213, 2006.

[3] Bhadange P J and Thakre S B, “CFD analysis of fluid flow through pipe fitting”, International Journal of Pure and Applied Research in Engineering and Technology, vol. 1, pp. 224-233, 2013.

[4] Chalet D and Chesse P, “Fluid dynamic modelling of junctions in internal combustion engine inlet and exhaust systems”, Journal of Thermal Science, vol. 19, pp. 410-418, 2010.

[5] Cho J, Kim K, Kim J and Jeong E, “Controlling the secondary flows near end wall boundary layer fences in a 90° turning duct using approximate optimization method”, Journal of Mechanical Science and Technology, vol. 25, pp. 2025-2034.