RESEARCH PAPER | Engineering | Volume : 3 | Issue : 6 | June 2013 | ISSN - 2249-555X

Case Study on use of Computational Fluid Dynamics for prediction of hydrodynamic characteristics of Exhaust Muffler

KEYWORDS: Exhaust Muffler, Reynolds number, Computational Fluid Dynamics

Prof. Dr. M.T. Bhoite
Dept Mechanical Engineering, Maharashtra Institute of Technology Academy Of Engineering , Alandi (D) , Pune, Maharashtra, India -412105

Akshay I. Babar
Dept Mechanical Engineering, Maharashtra Institute of Technology Academy Of Engineering , Alandi (D) , Pune, Maharashtra, India -412105

Prasad S. Yadav
Senior Project Engineer, Noise Vibration and Harshness Laboratory, Automotive Research Association of India, Pune

N.V.Karnath
Deputy Director, Noise Vibration and Harshness Laboratory, Automotive Research Association of India, Pune

ABSTRACT *In the present paper, an attempt is made to illustrate how hydrodynamic characteristics can be determined for a typical exhaust muffler using Computational Fluid Dynamics. Pressure drop in the muffler has major two components- a contribution from static pressure drop and contribution from dynamic pressure drop. Dependence of both the contributions on Reynolds number is investigatd in this paper and expressed in the form of correlating equations. Pressure-velocity coupling is handled using segregated solver.*

1.0INTRODUCTION

Exhaust mufflers are are part of the exhaust system and are expected to produce the large amount of pressure drop to minimize the noise production. They are also responsible for chemical reduction of emissions of NO_x, CO etc.

Current analysis is focused on pressure drop characteristics of an exhaust muffler. For simplicity a straight flow baffled muffler configuration is chosen. Pressure drop and Reynolds number characterstics of a flow are reported in terms of Reynolds number and friction factor as revealed by standard reference like White[1] or Bejan[2].

2.0MATHEMATICAL FORMULATION AND SOLUTION METHODOLOGY

Fig.1 represents the geometrical model of the exhaust muffler under test. The configuration of muffler consists of tail pipe from engine of 25 mm radius which is expanded to double radius by a conical section. In the way of exhaust pipe to outlet, three symmetric baffles are assumed to be placed in order to produce heavy pressure drop across the muffler.

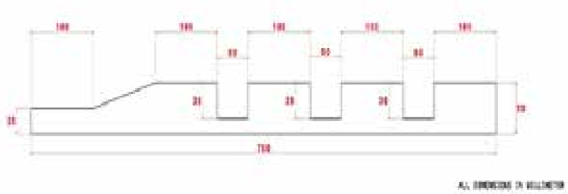

Fig.1 Geometrical Configuration of Exhaust Muffler

To obtain the pressure drop, a numerical solver needs to solve following set of governing equations.

Equation of state: $\rho = P/RT$ (1)

Continuity Equation: $\partial \rho / \partial t + \mathrm{div}(\rho V) = 0$ (2)

Momentum Equations in vector form :
$\partial(\rho V)/\partial t + \mathrm{div}(\rho VV) = -\mathrm{grad}(p) + \mathrm{div}(\tau) + \rho g$ (3)

Energy Transport Equation:
$\partial(\rho E)/\partial t + \mathrm{div}(V(\rho E + p)) = -\mathrm{div}.(K\, \mathrm{grad}(T) + \tau . V)$ (4)

where V is a velocity field, E is the internal energy of the gas and is expressed as

$E = h - p/\rho + |V|^2/2$ (5)

and all the other symbols carry their usual meaning. The enthalpy h is related with temperature with relation

$dh = C_p\, dT$. (6)

The set of above interlinked partial differential equations needs to be solved to know variations of density, velocity, pressure and temperature inside the domain.

For simplicity, the geometrical configuration is assumed to be symmetric around central axis and a simplified analysis is carried out. The governing equations (which are in vector notations) are formulated in cylindrical coordinates and gradients in θ direction and velocity component in θ direction are assumed to be zero.

Axi-symmetric analysis can be carried out in commercial CFD code-ANSYS-FLUENT .Computational Procedure involved preprocessing and meshing of the geometry, deployment of suitable boundary conditions. A mesh of 2.5 mm size is deployed across the control volume after grid independence tests. The mesh is depicted in the Fig. 2

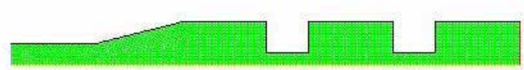

Fig.2 A typical mesh for the silencer geometries.

Boundary conditions are summarized as : At inlet, a known mass flow rate of air is prescribed and at outlet, pressure is anchored. Walls are assumed to be adiabatic boundaries and at axis boundary conditions are specified such that quantities remain finite and still satisfy the governing equations.

Numerical algorithm involved discretization of governing equations over finite control volumes shown in Fig.2. Control volume integration procedure uses Gauss Divergence theorem and converts differential equations into the set of

Volume : 3 | Issue : 6 | June 2013 | ISSN - 2249-555X

algebraic equations which are solved using pointwise iterative solver. Convection terms are handled using second order upwind schemes and diffusion terms are handled with central differencing schemes, large pseudo-time step is deployed to obtain fast convergence as transient results are practically of no interest as compared to steady state results. The detailed procedure is available in [3]

Results of the present study are reported in terms of Pressure drop, friction factor where friction factor is pressure drop scaled with dynamic pressure and friction factor is a function of Reynolds number and geometrical configuration. However, in a given configuration, friction factor is correlated with Reynolds number. Thus, in the present study, friction factor is found out for various Reynolds number.

3.0 RESULTS

The results are presented in the form of scaled pressure drop across the silencer and which is calculated based on area weighted average of static pressure at inlet and outlet. The scaled/dimensionless pressure drop is also referred to as friction factor is defined as

$f = \Delta p / (\rho U_{in}^2)$ (7)

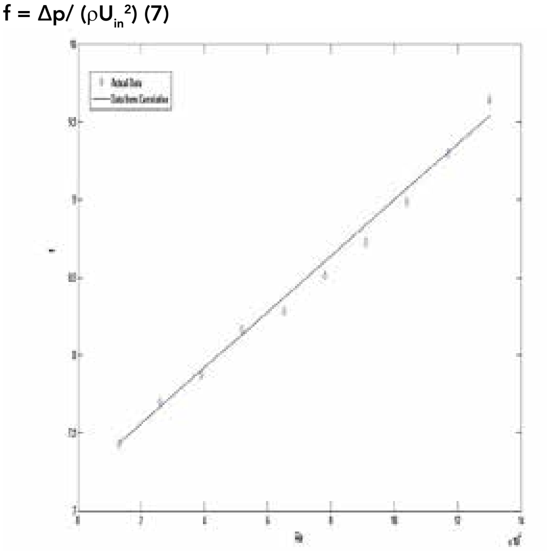

Fig. 3 Variation of friction factor with Reynolds number

It can be seen from Fig.3 that friction factor monotonously increase with the increase in Reynolds number and follows almost a linear increase. The values of friction factor are substantially large because the flow is of compressible nature and Reynolds number is quite large.

The data of friction factor is correlated with Reynolds number using least square fitting technique and this resulted into following correlation

$f = \Delta p / (\rho U_{in}^2) = a + b\,Re$ (8)
($a = 7.2$ and $b = 1.8 \times 10^{-5}$)
(Correlation Coefficient = 0.9954 and Mean Square Error = 0.0052)

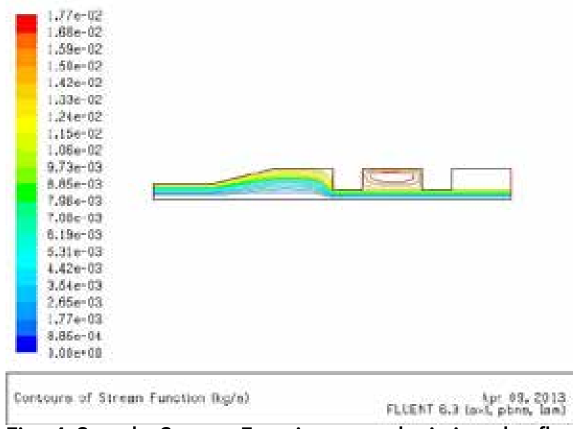

Fig. 4 Sample Stream Function map depicting the flow profile at Re = 1.169×10^5.

Fig 4 depicts sample stream function map for exhaust muffler. As can be seen from streamfunction maps flow stream passes through main central passage and a recirculation zone is created in the annular spaces between baffles, a small amount of reverse flow is seen at the end section.

Additionally, dynamic pressure drop can be found out using ideal flow theory and can be given by,

$$\Delta P_{dynamic} = \tfrac{1}{2}(\rho V^2_{out} - \rho V^2_{in}) \quad (9)$$

In this expression, inlet and outlet velocities are interrelated based on mass balance of inlet and out.

4. CONCLUSIONS

Application of computational fluid dynamics for silencer design is exemplified in this paper, similar procedure can be applied for various geometrical configurations of silencer.

It is found that friction factor and total pressure drop increase with the increase in Reynolds number. Reynolds number is the parameter of importance for a given geometry.

REFERENCE [1] F.M. White (2008) Fluid Mechanics, . McGraw-Hill, New York. | [2] A. Bejan (1984) Convection Heat Transfer, Wiley, New York,. | [3] S.V. Patankar, (1980) Numerical Heat Transfer and Fluid Flow. McGraw-Hill, New York. |