International Journal in IT & Engineering (IJITE)
Volume 6 Issue 4, April 2018 ISSN: 2321-1776 Impact Factor: 6.341
Journal Homepage: http://ijmr.net.in, Email: irjmss@gmail.com
Double-Blind Peer Reviewed Refereed Open Access International Journal

The Methodology For The CFD Investigations In The Exhaust Gas Manifold Pipe Of A Single Cylinder Spark Ignition Engine To Improve Its Performance

M. Marouf Wani
Mechanical Engineering Department, National Institute of Technology, J&K, Srinagar, India.

Abstract

This paper presents the methodology for CFD investigations in the exhaust gas manifold pipe of a single cylinder spark ignition engine using the Numerical Finite Volume Method. The investigations are done with an aim to improve the thermodynamic design of the engine under consideration. The results were computed in the thermodynamic simulation software AVL BOOST. The results were computed by solving the conservation equations of mass, momentum and energy with the help of the equation of state for an ideal gas. The values of the pressure, velocity, density and temperature were computed at various length steps along the pipe length under consideration for each time step for one complete cycle. The results were also computed for variation of the power, torque and BSFC of the engine with respect to the length steps along the pipe length under consideration.

Keywords: CFD, Methodology, Manifold Pipe, Exhaust Gas, *Engine, Spark Ignition, Numerical Method, and Performance*

Introduction

The computational fluid dynamic investigations can be used to predict the thermodynamic and gas dynamic behavior of each of the components of the intake and exhaust gas manifolds of an internal combustion engine. The design of the intake manifold of an internal combustion engine effects the engine performance parameters like volumetric efficiency, power, torque, brake specific fuel consumption and its emissions characteristics. The design of the exhaust gas manifold involves the catalyst type components related to emission control from I C engines, mufflers related to noise reduction, EGR system based pipe connections for reducing the NOx emissions from engines, The addition of turbochargers in the gas exchange system of an internal combustion engine, for boosting the power of the engine or alternatively downsizing the engine, can also be optimized for best possible matching with the engine using the CFD simulations with finite volume method. Further the lengths of exhaust gas manifold components have to be optimized in such a way so that the superposition of the forward moving pressure waves and the reflected pressure waves result in minimum negative effect on engine performance parameters like power and torque.

Literature Survey

John B Heywood in his book writes that two basic types of models have been developed for the processes that govern the performance and emissions of I C engines. These are categorized as thermodynamic or fluid dynamic in nature depending on whether the equations which give the model its predominant structure are based on energy conservation or on a full analysis of the fluid motion. Thermodynamic energy conservation based models are: Zero – dimensional , since in absence of any flow modeling, geometric features of the fluid motion cannot be predicted; Phenomenological, additional detail beyond the energy conservation equations is added for each phenomenon in turn; Quasi-dimensional, specific geometric features, e.g., the spark-ignition engine flame or the diesel fuel spray shapes, are added to the basic thermodynamic approach; Fluid-dynamic based models or multidimensional models, these have inherent ability to provide detailed geometric information on the flow field based on solution of the governing flow equations. The governing equations for open thermodynamic system are: conservation of mass; conservation of energy and conservation of momentum. [1]

Morel, T., et al., conducted one-dimensional fluid dynamic simulations of the flow in the engine manifolds, exhaust and intake elements with an objective to design it both for engine performance as well as for engine acoustics. The experimental investigations followed verified the prediction of the good results based on simulation.[2]

Fortunate, F., et al conducted transient CFD investigations of the exhaust gas manifold fitted with a catalyst. The new European driving cycle, NEDC , was adopted for these transient simulations to evaluate the vehicle fuel economy and its emissions characteristics. Further the effect of the exhaust gas temperature on the catalyst light-off time were also simulated by designing manifolds with different configurations and materials. The computed results were reasonably validated by the experimental investigations. The results showed that a thin exhaust gas manifold made of steel improves the thermal light-off of the catalytic convertor as compared to a similar manifold made of thicker cast iron. [3]

Maftouni, N., et al conducted 3-D simulations on the intake manifold of a XU7 Engine under steady state and unsteady state operation using 3-D CFD code. The simulations were carried out with three different runner lengths of 110, 120 and 130% of the original to see its effects on the volumetric efficiency of the engine under variable speed operation. The steady state results were compared with the flow bench rig based data for validation. 1-D WAVE code was used for generating the boundary condition for unsteady state simulation. The results showed that the volumetric efficiency of the engine increases with 120% runner length.[4]

Nanni, D. et al., conducted experimental investigations on the silencer of a motorcycle engine to validate the CFD model simulated to predict the performance of the same. The experiments were conducted on a test bench where the mass flow rate of the exhaust gas through the silencer was measured for several inlet-outlet pressure gradients.The finite volume method based numerical data for mass flow rate was computed for the same with several mesh sizes and computational settings.

The results showed that the finite-volume model is a promising method for analyzing the performance of silencers with different designs. Further the FVM model does not give 100% accurate results to predict the absolute performance of the silencers.[5]

Pal, D., et al., conducted simulation studies on the design of intake manifolds in order to achieve a required level of engine performance. First of all the simulations were done using 1-D CFD code to optimize various parameters of the intake manifold. A few shortlisted designs were tested experimentally which showed strong correlation with simulated results. The second series of investigations was done by using a 3D CFD code for the intake manifolds to analyze the 3D effects of the manifold geometry. The third set of simulations was carried out to tune the manifold geometry further for possible optimum dimensions by using a coupled 1D and 3D CFD code. The results showed that the simulation studies lead to a better prediction of the engine performance. [6]

The 3D CFD simulations can give the detailed information about fluid and flow properties in complex 3D domains while 1D CFD simulation can provide important information at a system level, i.e about the performance of the entire engine. The drawbacks of the two simulation methods are that the former requires high computational cost while the latter is not able to capture complex local 3D features.

Testa, F., et al conducted computational analysis of the unsteady flow in a single cylinder two stroke gasoline engine using advanced numerical tools. The results were validated by the experimental measurements. A STAR-CD code based 3D model was used for the entire engine. Also a GT-POWER 1D and Converge-LITE 3D codes based 1D-3D integrated fluid dynamics model was used for predicting the performance and gas-dynamics in the whole intake and exhaust systems. The results showed that the methodology accurately predicted the phenomena in whole engine. Further it was observed that the wave motion based analysis strongly affects 3D design of muffler in small two stroke engines. [7]

Methodology Used In Present Investigations

The computational methodology was started by writing conservation equations for a control volume in the partial differential form because of the involvement of more than one variable. Starting from the fundamental control volume approach, only one length step along flow was considered. The integral calculus based integration was used to integrate the computations for the entire pipe length for each time step.

In the modeled engine manifolds with complex design having multiple types of components, the integration was done component to component along the flow in the entire pipe length. The CFD results presented in this paper are for the entire pipe length and the complete engine as a system. The integration was carried out with by using the actual limits of integration for each pipe element under consideration.

Since the basic need is to know the values of the thermodynamic properties at each node along the pipe length for proper manifold design with all components needed as per demand in market, the finite volume method was used for solving the equations written for control volume.

Again for analysis and design of the engine manifold pipe design the boundary conditions were known at the exhaust port of the engine at the time of exhaust opening computed by the thermodynamic full cycle simulations inside the engine cylinder. The atmospheric boundary conditions at the end of the exhaust gas manifold pipe where the gases flow into atmospheric surroundings are also known.

The central difference method of the finite difference scheme was employed for computing the values of thermodynamic properties at each node of the pipe along the flow.

The number of length steps, for the corresponding time steps for full cycle simulation, were governed by the principle of minimization of error as per the CFL stability criterion.

Further as per the principles of wave propagation for a particular element with certain geometry under consideration, the waves are reflected at point of geometrical discontinuity along the flow in the pipe which represents the change in medium.

In order account for the effect of the reflected waves of the time step under consideration and the forward moving wave generated at the next time step the equations for the waves were solved as per D-Alembert's principle.

The D-Alembert's solution was used to write the wave equation for each thermodynamic variable as a vector sum of the forward moving wave and the reflected backward moving wave.

Theoretical Basis.[8,9]

Conservation Equations for mass, momentum and energy in the following form are used for solving the research problems related to 1-Dimensional compressible flow of gas in a pipe.

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\rho u}{F}\frac{dF}{dx} \text{ ---(1)}$$

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2 + p)}{\partial x} + \frac{\rho u^2}{F}\frac{dF}{dx} + \rho G = 0 \text{ ---(2)}$$

$$\frac{\partial \rho e_0}{\partial t} + \frac{\partial(\rho u h_0)}{\partial x} + \frac{\rho u h_0}{F} - \rho q = 0 \text{ --(3)}$$

where

$$G = \frac{1}{2}\, u\,|u|\, f\, \frac{4}{D} \text{ --(4)}$$

The term $u\ |u|$ is used to ensure that the pipe wall friction always opposes the fluid motion The above three equations written in conservation law form can be written in symbolic vector form as

$$\frac{\partial \boldsymbol{W}}{\partial t} + \frac{\partial \boldsymbol{F}(\boldsymbol{W})}{\partial x} + \mathbf{C}\,(\mathbf{W}) = 0 \text{ ---(5)}$$

Computation of thermodynamic and gas dynamic properties of the gas throughout the pipe length is done by solving the conservation equations for getting the numerical values of various properties by integration as follows: The equations are integrated with respect to time step and length step limits. The integral form of the governing equations can be written as

$$\int_o^x \int_0^t \left[\frac{\partial W}{\partial t} + \frac{\partial F(W)}{\partial x} + C(W)\, dx\, dt\right] = 0 \text{ --------------------------------------- (6)}$$

where

$$\mathbf{W} = \begin{bmatrix} \rho \\ \rho u \\ \rho e_o \end{bmatrix} \tag{7}$$

$$\mathbf{F}(\mathbf{W}) = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho u h_o \end{bmatrix} \tag{8}$$

and

$$\mathbf{C}(\mathbf{W}) = \begin{bmatrix} \rho u \\ \rho u^2 \\ \rho u h_o \end{bmatrix} \frac{d(LnF)}{dx} + \begin{bmatrix} 0 \\ \rho G \\ -\rho q \end{bmatrix} \tag{9}$$

Equation (6) on integration, gives

$$(W_i^{n+1} - W_i^n)\Delta x + (F^n_{i+\frac{1}{2}} - F^n_{i-\frac{1}{2}})\Delta t + C_i^n \Delta x \Delta t = 0 \tag{10}$$

In this equation, W represents the average of dependent variables for the cell given by

$$W_i = \frac{1}{\Delta x}\int_{x_{i-1/2}}^{x_{i+1/2}} W dx \tag{11}$$

And F is the average flux across cell boundaries over an interval of time Δt, given by

$$F_{i\pm 1/2} = \frac{1}{\Delta t}\int_{t^n}^{t^{n+1}} F dt \tag{12}$$

Equation (10) is the fully discrete, integral form of the one-dimensional system of conservation laws defined already.

When the vector of source terms, C , is omitted, equation (10) reduces to the form

$$W_i^{n+1} = W_i^n - \frac{\Delta t}{\Delta x}(F^n_{i+\frac{1}{2}} - F^n_{i-\frac{1}{2}}) \tag{13}$$

The left-hand side of the equation represents the solution at the new time level n+1, and the first term on the right-hand side represents the solution at time level n. Summing this equation with respect to the spatial index, i, and omitting source terms gives

$$W_i^{n+1} = \Delta x \sum_{i_{min}}^{i_{max}} W_i^n + \Delta t(F^n_{i_{min-1/2}} - F^n_{i_{max+1/2}}) \tag{14}$$

The equations were applied to fluid flow through a pipe of constant cross-sectional area, where $F_{i_{min-1/2}}$ and $F_{i_{max+1/2}}$ are the fluxes through the extreme ends of the pipe, all internal fluxes cancel out. This guarantees the integral form of the conservation equations, thereby ensuring, for example, the conservation of mass through a pipe.
The finite difference methods used above when interpreted in the form of equations (10) to (12) are called conservative, finite volume schemes.

CFL Stability Criterion

During the course of numerical integration of the conservation laws defined in the Eq.1, Eq.2 and Eq.3, special attention should be focused on the control of the time step. In order to achieve a stable solution, the CFL criterion (stability criterion defined by Courant, Friedrichs and Lewy) must be met:

$$\Delta t \le \frac{\Delta x}{u+a} \text{--}(15)$$

This means that a certain relation between the time step and the lengths of the cells must be met. The time step to cell size relation is determined at the beginning of the calculation on the basis of the specified initial conditions in the pipes. However, the CFL criterion is checked every time step during the calculation. If the criterion is not met because of significantly changed flow conditions in the pipes, the time step is reduced automatically.

An ENO scheme is used for the solution of the set of non-linear differential equations discussed above. The ENO scheme is based on a finite volume approach. This means that the solution at the end of the time step is obtained from the value at the beginning of the time step and from the fluxes over the cell borders

Results And Discussions

The Effect Of Length Step On The Instantaneous Pressure Of Exhaust Gas Along The Flow In Pipe

The Fig.1 below shows the computed pressure at the nodal points along the length of the pipe in the exhaust manifold. The pressure in the pipe increases with length steps towards the catalyst because of the effect of backward moving reflected pressure waves.

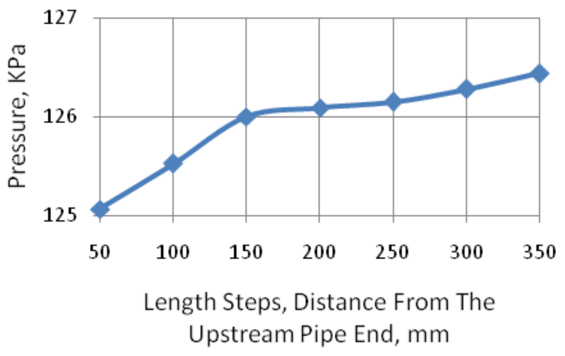

Fig.1 Effect of Length Step on the Instantaneous Pressure of Exhaust Gas in The Pipe of The Exhaust Manifold

The Effect Of Time Step On The Pressure Of Exhaust Gas At The Same Node Along The Flow In Pipe

The Fig.2 below shows the computed pressure as a function of time steps at the same node in the pipe of the exhaust gas manifold.

The variation in pressure as well as its peak value with time steps at the node under consideration is due to the changing thermodynamic boundary conditions at the exhaust port of the cylinder of the engine corresponding to same time steps.

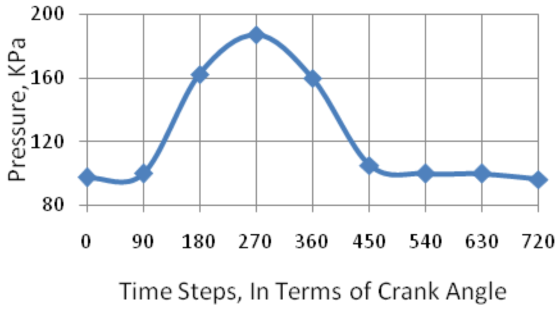

Fig.2 Effect of Time Step on the Pressure Of Exhaust Gas in The Pipe Of Exhaust Manifold At The Same Node

The Effect Of Length Steps On The Instantaneous Temperature Of Exhaust Gas Along The Flow In Pipe

The Fig.3 below shows the computed temperature at the nodal points along the length of the pipe in the exhaust gas manifold.

It is seen from the figure that the temperature first increases with length steps and then decreases towards the pipe end. This is because the pressure rise of the gas remains a dominant factor first and then the increase in density of gas remains a dominant factor numerically when substituted in the equation of state.

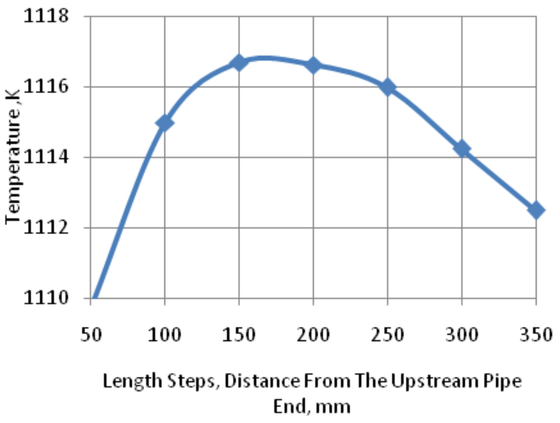

Fig.3 Effect of Length Step on the Instantaneous Temperature Of Exhaust Gas in the Pipe Of Exhaust Gas Manifold

The Effect Of Length Step On The Instantaneous Density Of The Exhaust Gas Along The Flow In Pipe

The Fig.4 below shows the computed density at the nodal points along the length of the pipe in the exhaust gas manifold.

It is seen that the density of the exhaust gas in the pipe under consideration decreases and then again increases as the length steps advance towards the catalyst in the exhaust gas manifold. This is because the rise in temperature of the exhaust gas remains a dominant factor first and then the rise in pressure becomes a dominant factor when substituted numerically in the equation of state.

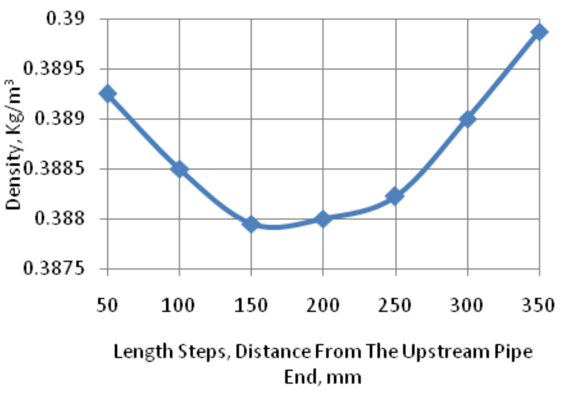

Fig.4 The Effect of Length Step on the Instantaneous Density Of Exhaust Gas in the Pipe Of the Exhaust Gas Manifold

The Effect Of Length Step On The Instantaneous Velocity Of Exhaust Gas Along The Flow In Pipe

The Fig.5 below shows the computed velocity of the exhaust gas at the nodal points along the length of the pipe in the exhaust gas manifold.

It is seen that the velocity of the exhaust gas in the pipe under consideration decreases with the length steps towards the catalyst in the exhaust gas manifold.

This is because of the rise in back pressure due to reflected pressure waves at the catalyst near the pipe. The resistance due to backward moving reflected pressure waves gets increased towards the pipe end.

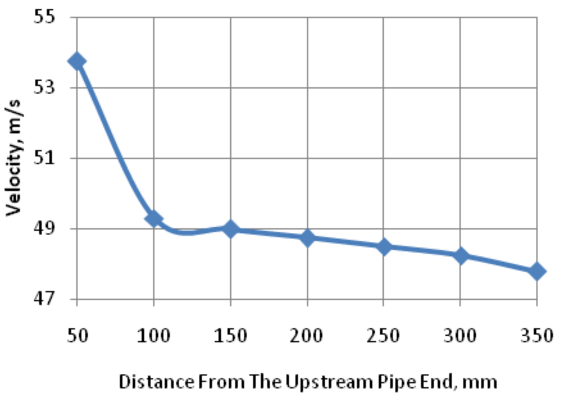

Fig.5 The effect of Length Step on the Instantaneous Velocity Of Exhaust Gas in the Pipe Of Exhaust Gas Manifold

The Effect Of Length Step On The Instantaneous Mass Flow Rate Of Exhaust Gas Along The Flow In Pipe

The Fig.6 below shows the computed mass flow rate of the exhaust gas at the nodal points along the length of the pipe in the exhaust gas manifold.

It is seen that the mass flow rate of the exhaust gas in the pipe under consideration decreases and then again increases as the length steps advance towards the catalyst in the exhaust gas manifold. This is because of the combined effects of the thermodynamic properties of pressure, density and the temperature of the exhaust gas along the pipe towards the catalyst.

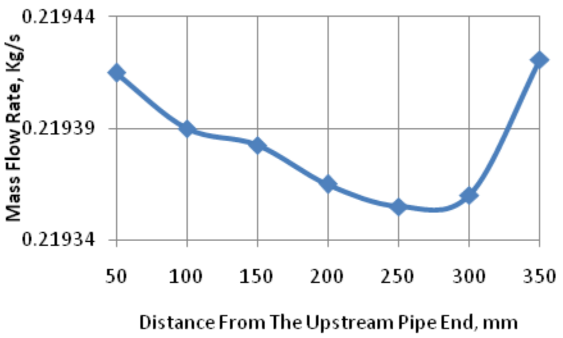

Fig.6 The Effect of Length Step on the Instantaneous Mass Flow Rate Of Exhaust Gas in the Pipe Of Exhaust Gas Manifold

The Effect Of Length Step On The Instantaneous Enthalpy Flow Along The Flow In Pipe

The Fig.7 below shows the computed enthalpy of the exhaust gas at the nodal points along the length of the pipe in the exhaust gas manifold for the same time step.

It is seen that the enthalpy flow associated with the mass flow of the exhaust gas decreases with the length steps towards the catalyst in the exhaust gas manifold due to the combined effect of mass flow rate and the temperature of the exhaust gas at each node.

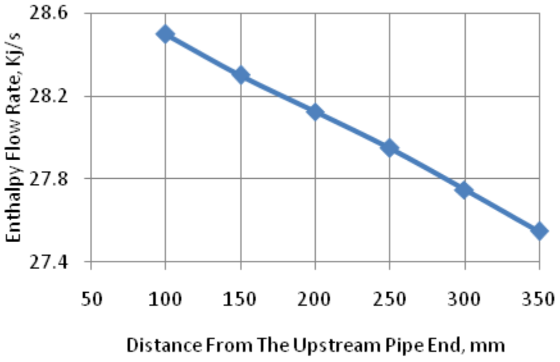

Fig.7 Effect of Length Step on the Instantaneous Enthalpy Flow Rate in the Pipe of the Exhaust Gas Manifold

The Effect Of Length Step On The Instantaneous Courant, Friedrichs And Lewy Stability Results

The Fig.8 below shows the effect of length steps on the percentage convergence as per the CFL stability criterion being computed at each node along the pipe at the instantaneous value of time under consideration.

It is seen from the figure that the results have converged to a required accuracy.

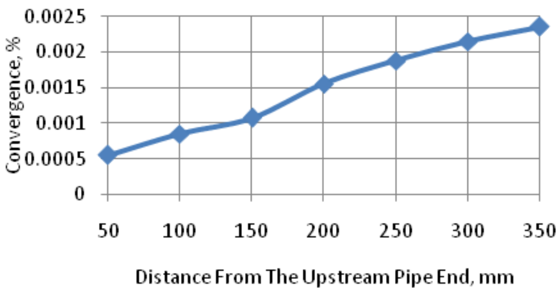

Fig.8 Convergence Of Properties Of Exhaust Gas At Nodal Points in Pipe Of Exhaust Manifold

Effect Of Length Of The Exhaust Gas Manifold Pipe On Volumetric Efficiency Of The Engine

The Fig.9 below shows the effect of the length of the pipe on the volumetric efficiency of the engine. The volumetric efficiency of the engine increases by increasing the pipe length for the engine manifold design.

This is because of the better removal of the exhaust gases from the engine cylinders under the considered design and operating conditions. The tuning of the engine manifold increases with increase in the pipe length because of favorable thermodynamic wave propagations along the flow in the pipe of the exhaust gas manifold.

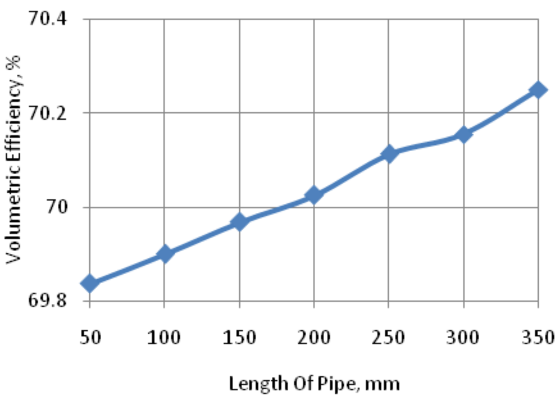

Fig. 13 Effect Of the Length Of The Exhaust Gas Manifold Pipe On The Volumetric Efficiency Of The Engine

Effect Of Length Of The Exhaust Gas Manifold Pipe On Power Developed By The Engine

The Fig. 10 below shows the effect of the length of the pipe on the power developed by the engine.

It is seen that as the length of the pipe is increased the power developed by the engine increases. This is because of the better tuning of the exhaust gas manifold and the corresponding rise in the volumetric efficiency of the engine by increasing the length of this pipe.

The fluctuations in the power curve along the length of the pipe is due to the combined effect of the forward moving waves and the backward moving reflected waves for the pipe length under consideration.

Once a particular pipe length is chosen the power output of the engine will be optimized accordingly.

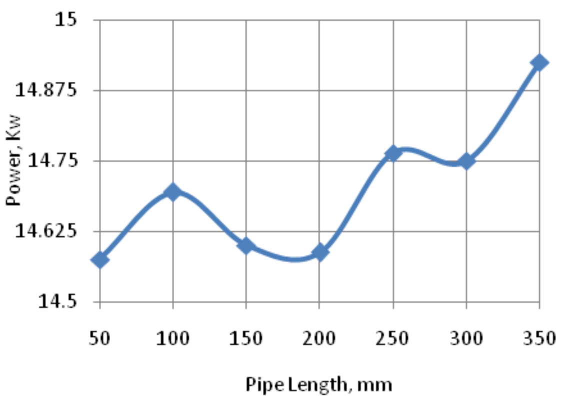

Fig. 10 Effect of the Length of the Exhaust Gas Manifold Pipe on the Power Developed By The Engine

Effect Of Length Of The Exhaust Gas Manifold Pipe On Torque Developed By The Engine

The Fig. 11 below shows the effect of the length of the pipe on the torque developed by the engine.

It is seen that as the length of the pipe is increased the torque developed by the engine increases. This is because of the better combustion characteristics in the engine cylinders due to better tuning of the exhaust gas manifold.

The fluctuations in the torque curve along the length of the pipe is the effect of the superposition of the forward moving thermodynamic waves and the reflected thermodynamic waves.

The selection of a particular pipe length will make the torque developed by the engine independent of the pipe length.

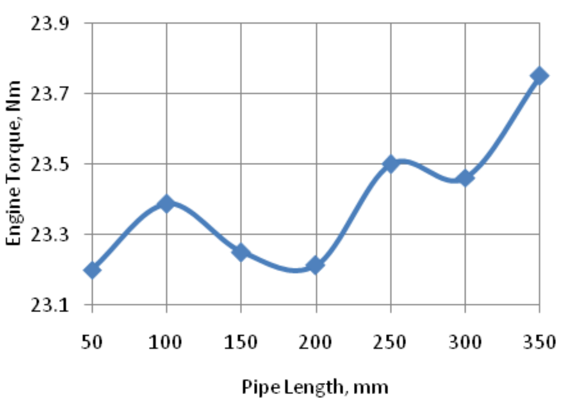

Fig. 11 Effect of the Length of the Exhaust Gas Manifold Pipe On the Torque Developed by the Engine

Effect Of Length Of The Exhaust Gas Manifold Pipe On The Brake Specific Fuel Consumption Of The Engine

The Fig. 12 below shows the effect of the length of the pipe on the brake specific fuel consumption of the engine.
It is seen that as the length of the pipe is increased the brake specific fuel consumption of the engine decreases.
This is because the manifold tuning improves with the pipe length which improves the volumetric efficiency of the engine as well as the power developed by it.
The fluctuations in the BSFC curve along the length of the pipe is due to the combined effect of the forward moving waves and the backward moving reflected waves for the pipe length under consideration.
After deciding about the suitable pipe length the BSFC of the engine will be independent of the pipe length.

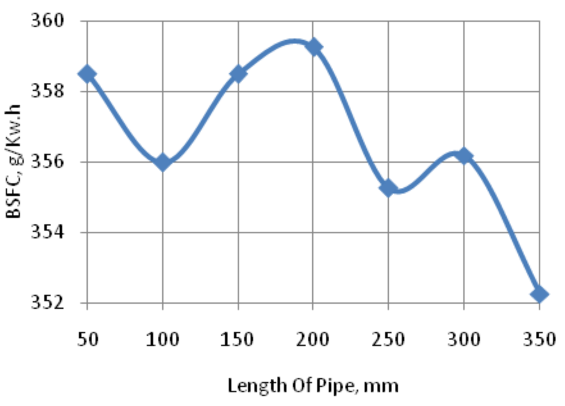

Fig. 12 Effect of the Length Of the Exhaust Gas Manifold Pipe On The Brake Specific Fuel Consumption Of The Engine

Conclusions

1. The Numerical Finite Volume Method based CFD analysis can be used to predict the numerical values of the thermodynamic properties at each node along the length of the exhaust gas manifold pipe..
2. The Numerical Finite Volume Method can be used in the design of the exhaust gas manifold pipe for improving the volumetric efficiency of the engine.
3. The CFD analysis using FVM can be used to optimize the length of the exhaust gas manifold for maximum engine performance in terms of power, torque and BSFC etc.
4. The numerical FVM can be used on variable speed automotive engines for designing the manifolds which can filter the noise over a wide range of the operating frequencies of the engine. This can be accomplished by designing the exhaust gas manifolds with multiple expansion chambers or Helmholtz resonators in suitable series and parallel combinations.
5. In general the Numerical Finite Volume Method can be applied in any flow based engine component for predicting the values of thermodynamic properties both on length step basis as well as on time step basis. This will further help to optimize the design of the engine component under consideration.

Acknowledgements

Author is thankful to AVL Austria and its unit AVL India Ltd Gurgaon for providing BOOST thermodynamic engine simulation software with free license for academic research purposes.

Appendix-A

Nomenclature

a = speed of sound
C = vector of source terms
c_V = specific heat at constant volume
c_p = specific heat at constant pressure
D = pipe diameter
f = wall friction coefficient
F = vector in x direction
F =element in vector **F**
k = ratio of specific heats
p = static pressure
P_0 = stagnation pressure
q = wall heat flow
t = time
T = temperature
T_0 = stagnation temperature
e = specific internal energy
u = flow velocity
V = cell volume (A.dx)
W = vector of convective fluxes
ρ = density
Δt = time step
Δx= cell length

Table-1[10]

Engine Specifications	
Engine Type	Single Cylinder
Method of Ignition	Spark Ignition
Displacement, CC	500
Compression Ratio	9
Number of Cylinders	1
Rated Speed	6000 rpm

Table-2[10]

Physical And Chemical Properties Of Petrol

Fuel Property	Petrol
Formula	C4 TO C12
Density, Kg/m3	750
Lower heating value, MJ/Kg	42.5
Stoichiometric air-fuel ratio, weight	14.6
Octane No.	80-98
Auto-ignition Temperature, C	280

References

[1] Heywood John B., "Fundamentals of internal combustion engines" McGraw Hill International, 1989.

[2] Morel, T., Silvestri, J., Goerg, K., and Jebasinski, R., "Modeling of Engine Exhaust Acoustics", SAE Technical Paper 1999-01-1665, 1999

[3] Fortunato, F., Quadrini, F., and Bova, S., "Catalyst Light-off Evaluation Using CFD Simulation of the Exhaust Manifold", SAE Technical Paper 2005-01-3895, 2005

[4] Maftouni, N., Ebrahimi, R., and Hossein pour, S., " The effect of Intake Manifold Runners Length on the Volumetric Efficiency by 3-D CFD Model" SAE Technical Paper 2006-32-0118, 2006.

[5] Nanni, D., Rossi, R., Tarabusi, S., and Di Piazza, S., "Experimental Validation of a CFD Model to Predict Performance of a Motorcycle Silencer", SAE Technical Paper 2008-01-0897, 2008

[6] Pai, D., Singh, H., and Muhammad, P., "Simulation Based Approach for Optimization of Intake Manifold", SAE Technical Paper 2011-26-0074, 2011

[7] Testa, F., Gagliardi, V., Ferrai, M., Fontanesi, S., et al ., "Guidelines for the Optimization of a Muffler in a Small Two Stroke Engine", SAE Technical Paper 2016-32-0050, 2016

[8] Winterbone DE and RJ Pearson," Design Techniques for Engine Manifolds",SAE International USA, 1999.

[9] Winterbone DE and RJ Pearson," Theory of Engine Manifold Design", Professional Engineering Publishing Ltd, UK, 2000.

[10] AVL LIST Gmbh , AVL BOOST Theory, Version 2009.1