

Modeling of Silencers for I.C. Engine Intake and Exhaust Systems by Means of an Integrated 1D-multiD Approach

G. Montenegro and A. Onorati

Dipartimento di Energetica - Politecnico di Milano

Copyright © 2008 SAE International

ABSTRACT

This paper describes the development of a fully 1D and of a 1D-multiD integrated approach for the simulation of complex muffler configurations. The fully 1D approach aims to model the muffler recurring to an equivalent net of 1D pipes. An expansion chamber with offset inlet and outlet pipes was modeled with this procedure and the results compared to CFD simulations, pointing out some critical aspects in the TL prediction. The HLLC Riemann solver and its extension to the second order were implemented both in the 1D and multiD models and exploited to handle the interface between the calculation domains. The integrated 1D-multiD approach was used afterwards to predict the transmission loss of more complex geometries such as series chambers with extended inlet and outlet pipes and with flow reversals. A new procedure has been adopted to calculate the transmission loss, imposing a pressure impulse at the inlet and evaluating the response of the muffler. This method, along with the adoption of more accurate numerical schemes improved considerably the accuracy of the result and shortened the computational time. The results pointed out a better prediction of the coupled approach if compared to the results of the 1D simulations. Moreover, a real engine configuration has been adopted to predict the acoustic performance of the reverse chamber under realistic operating conditions, in which pressure perturbations have finite amplitude and mean flow is present.

INTRODUCTION

Nowadays a great attention is paid to the level and quality of noise radiated from the tailpipe of intake and exhaust systems, to control the gas dynamic noise emitted by the engine as well as the characteristics of the cabin interior sound. The muffler geometry can be optimized consequently, to attenuate or remark certain spectral components of the engine noise, according to the result expected. Evidently the design of complex silencing systems is a time-consuming operation, which must be carried out by means of concurrent experimental measurements and numerical simulations. In particular, 1D and multiD linear/non-linear simulation codes can be applied to predict the silencer behavior in the time and frequency domain. Firstly, it is important to predict the attenuation features of the muffler (for example the Transmission Loss - TL) as an independent element, separated from the engine, to evaluate the attenuation peaks and troughs versus frequency related the resonances in the ducts. Secondly, the silencing system must be coupled to the engine source and the simulation model can be useful to predict the response of the muffler submitted to the finite amplitude pressure pulsations, with significant mean flow, generated by the engine gas exchange process. The interaction between the acoustic source (i.e. the I.C. engine) and the silencing duct system is fundamental to correctly capture its real attenuation characteristics, the consequent pressure losses (which influence the engine performances) and the radiated tailpipe noise level and quality. The prediction of the silencer attenuation (TL) in the frequency domain can be fulfilled by means



Documents

Document Sharing Hub

of both linear (acoustic) and non-linear (fluid dynamic) models, 1D or multiD. 1D linear acoustic codes have generally been used for this purpose, due to their simplicity, accuracy and very short computational times. The acoustical behavior of complex shape devices (air boxes, resonators, asymmetric expansion chambers, etc.), characterized by a significant non-planar wave motion, can be correctly captured resorting to multiD linear acoustic models, based on the FEM or BEM method [1, 2]. On the other hand, an accurate prediction of the TL can be achieved also by means of non-linear models, 1D or CFD, with the advantage of a more general approach, suitable to account for finite amplitude wave effects; the drawback is the computational effort, typically two orders of magnitude greater in this case. Furthermore, the adoption of non-linear models is recommended for the calculation of the silencing system connected to the engine source, to carry out a realistic prediction of the muffler interaction with the engine, when submitted to high amplitude pressure waves. In this case an integrated 1D-multiD fluid dynamic (non-linear) simulation of the whole system, represented by the engine and the complete intake and exhaust ducting, can be very useful to test and optimize the silencer configuration on a virtual engine test bench and calculate the tailpipe noise radiated by the inlet and outlet open ends. For example, a flexible use of integrated 1D-multiD codes can allow the hybrid modeling of the whole engine by means of 1D and 3D computational domains connected together. The CFD approach could be eventually applied for the simulation of the cylinder, the intake plenum, the exhaust manifold, the silencers, resorting to the 1D approach only to model the simple geometry pipes which connect the various devices of the air management system. In this way the prediction of the attenuation features of the silencing elements can be generally more accurate, thank to the use of 3D grids to represent the complex shapes and to the correct capturing of the non-linear effects dictated by the finite amplitude wave motion with significant mean flow.

As an alternative to the non-linear integrated 1D-CFD modeling of complex geometry duct systems, some researchers have proposed to use the 1D approach only and build up a suitable net of one-dimensional pipes, to create a schematic of the silencer or air box which should be acoustically equivalent to the real 3D geometry. This method seems promising, but can lead to an incorrect interpretation of the fluid dynamic and acoustic behavior of the system in some cases. In this paper both multiD and 1D non-linear models have been developed and applied to the simulation of I.C. engine silencers. The CFD code OpenFOAM and the 1D code GASDYN have been coupled to calculate the TL of reactive duct-systems, including expansion chambers with extended inlet/outlet and flow

reversals. The experimental data of TL for the validation of the modeling approach have been found in [3, 4]. In particular, a strict coupling between the 1D and CFD codes has been realized on the basis of Riemann solvers at the computational domain interface. In this way the upstream and downstream boundary conditions required to evaluate the TL of the silencers by the CFD code, namely a white noise anechoic inlet and an anechoic outlet respectively, have been imposed by the 1D approach, which has been used to model the first and last duct of the muffler, whereas the rest has been simulated by the CDF model. As a comparative method, a net of 1D pipes ha also been tested to point out the advantages and disadvantages of the fully 1D approach. Finally, a S.I. engine with the complete intake and exhaust systems has been modeled by means of the hybrid 1D-multiD code, to point out the potential of the coupled simulation.

FUNDAMENTAL EQUATIONS

The study of intake and exhaust systems can be reduced to the analysis of the propagation of pressure waves in more or less complex systems of pipes where the viscosity of the gas has a minor effect on the wave dynamics. Since fresh air and exhaust gas viscosity are low, this assumption can contribute to a substantial simplification of the problem preserving, at the same time, a good level of accuracy in the calculations. The governing equations of the unsteady flow inside the intake and exhaust systems are the conservation of mass, momentum and energy:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{U}) = 0 \quad (1)$$

$$\frac{\partial \rho \vec{U}}{\partial t} + \nabla \cdot (\rho \vec{U} \otimes \vec{U}) = -\nabla p \quad (2)$$

$$\frac{\partial \rho e_0}{\partial t} + \nabla \cdot (\rho e_0 \vec{U}) = -\nabla \cdot (p \vec{U}) \quad (3)$$

Depending on the complexity of the geometry, further simplifications can be introduced. For instance, parts in which the paramount dimension is the axial one can be assumed as planar wave regions. This last assumption, in addition to the inviscid flow hypothesis, leads to the 1D Euler system of equations. The adoption of numerical methods with first order accuracy, along with explicitly time marching techniques is an acceptable choice to achieve results in short time and to avoid the CFL constraints typical of second order schemes and of flux limiters [5]. As a matter of fact the prediction of engine performances such as volumetric efficiency, hence torque and power, are slightly affected by the accuracy of the method adopted, whereas considerable differences can be noted when the prediction of acoustic performances is addressed.

EXTENSION TO SECOND ORDER ACCURACY

For this reason the extension of the HLLC approximate solver has been developed according to the Weighted Average Flux (WAF) theory [6]. This approach has its origin in the random flux method, which determines the solution at the intercell on the basis of statistical considerations. Conversely, the WAF method follows a deterministic approach to determine the solution. There are two versions of the WAF: in one version the intercell flux is evaluated as an integral of the averaged physical flux across the full structure of the Riemann problem, in the other version the flux is computed from an averaged state at the intercell region [6]. In this study the first approach has been adopted. According to this choice, the WAF flux is then expressed as follows:

$$\mathbf{F}_{i+\frac{1}{2}} = \frac{1}{\Delta x} \int_{-\frac{1}{2}\Delta x}^{\frac{1}{2}\Delta x} \mathbf{F}(\mathbf{W}_{i+\frac{1}{2}}(x, \frac{\Delta t}{2})). \quad (4)$$

where $\mathbf{W}_{i+\frac{1}{2}}$ is the approximate solution of the Riemann problem at the intercell region.

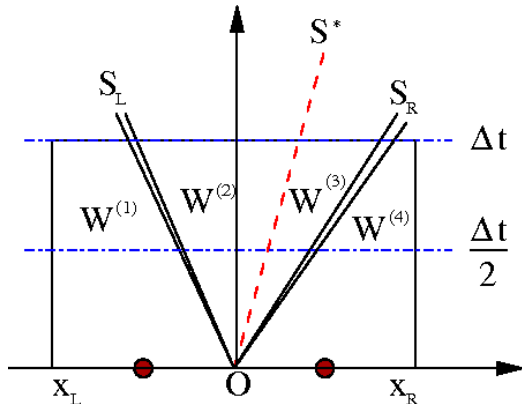


Figure 1: Time and space subdivision of the Riemann problem in the WAF approach

The evaluation of the integral, referring to the temporal and spatial subdivision of the domain, as shown in Figure 1, yields to the following expression of the intercell flux:

$$\mathbf{F}_{i+\frac{1}{2}} = \sum_{k=1}^{N+1} \beta_k \mathbf{F}_{i+\frac{1}{2}}^{(k)} \quad (5)$$

$$\mathbf{F}_{i+\frac{1}{2}}^{(k)} = \mathbf{F}(\mathbf{W}_{i+\frac{1}{2}}^{(k)}) \quad (6)$$

where N is the number of regions which the control volume is divided into by the three discontinuities S_L , S_R and S^* . The term $\mathbf{W}^{(k)}$ indicates the value of the conserved variables given by the solver used to find the solution of the Riemann problem. In this case we will consider the solution given by the approximate HLLC solver. The β_k coefficients, also defined as

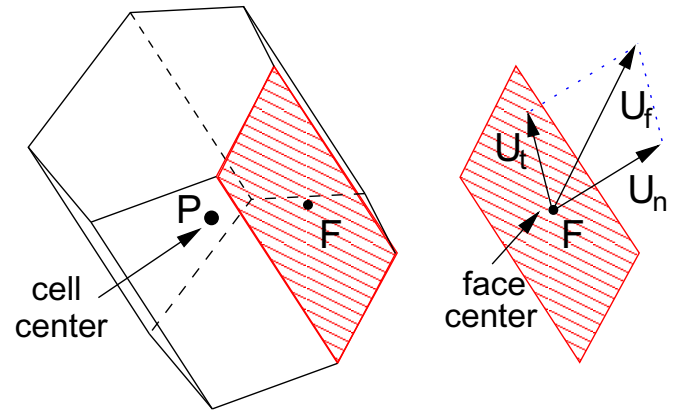


Figure 2: Generic cell face arbitrarily oriented in the space. The decomposition of the velocity vector is highlighted.

weighting factors, are calculated as functions of the wave Courant number coefficients c_k as follows:

$$\beta_k = \frac{1}{2}(c_k - c_{k-1}), \quad c_k = \frac{\Delta t S_K}{\Delta x} \quad (7)$$

With the contribution of some algebra, the flux of the variables becomes the following:

$$\mathbf{F}_{i+\frac{1}{2}}^{WAF} = \frac{1}{2}(\mathbf{F}_i + \mathbf{F}_{i+1}) - \frac{1}{2} \sum_{k=1}^N c_k \Delta \mathbf{F}_{i+\frac{1}{2}}^{(k)} \quad (8)$$

where

$$\Delta \mathbf{F}_{i+\frac{1}{2}}^{(k)} = \mathbf{F}_{i+\frac{1}{2}}^{k+1} - \mathbf{F}_{i+\frac{1}{2}}^k. \quad (9)$$

The extension to the second order of the HLLC is particularly needed for the prediction of acoustic performances of silencing devices. In particular, the first order approach can give a sufficiently accurate prediction of transmission loss for frequencies below 1200Hz, without recurring to an excessively refined calculation grid.

1D MODEL

The 1D model adopted for the simulation is a research code developed by the authors (GASDYN) [7, 8]. In the 1D model, the conservation equations of mass, momentum and energy are expressed in a strong conservative form and applied to the simulation of complex systems of pipes, with source terms to account for the friction and the heat transfer between the gas and the walls:

$$\frac{\partial (\rho F)}{\partial t} + \frac{\partial (\rho u F)}{\partial x} = 0 \quad (10)$$

$$\frac{\partial (\rho u F)}{\partial t} + \frac{\partial (\rho u^2 F + p F)}{\partial x} = p \frac{\partial F}{\partial x} - \frac{2}{D} \rho u |u| f_w F \quad (11)$$

$$\frac{\partial (\rho e_0 F)}{\partial t} + \frac{\partial (\rho e_0 u F + u p F)}{\partial x} = \rho (u \frac{\partial F}{\partial x} + F \frac{\partial u}{\partial x})$$

The system of equations is closed with the perfect gas equation of state. The set of equations is then solved applying explicit Godunov type methods based on the HLLC Riemann solver and on its extension to the second order accuracy (WAF) [6]:

$$W_i^{n+1} = W_i^n + \frac{\Delta t}{\Delta x} \left(F_{i-\frac{1}{2}} - F_{i+\frac{1}{2}} \right) - \Delta t C_i^n \quad (13)$$

where the intercell fluxes $F_{i\pm\frac{1}{2}}$ can be evaluated according to a first order approach (see Eq. A-6 in appendix) or to its second order formulation (Eq. 8). Moreover, the source term C is discretized according to the stencil proposed by other authors [5].

MULTI-DIMENSIONAL MODEL

The OpenFOAM code was adopted for the multidimensional simulation of the non-linear Euler set of equations. OpenFOAM [9, 10] is a general purpose finite volume solver, written in C++ object oriented programming language, and provides the user with suitable libraries to discretize partial derivatives operators applied to arbitrarily oriented meshes. The WAF solver was implemented in the OpenFOAM code to ensure the same discretization of the transport term over the 1D and 3D calculation domains and to enhance the accuracy of the previously implemented HLLC approximate solver. The Euler equations have been adopted to describe the unsteady flows also in the multidimensional domain, since at high Reynolds numbers (typical of intake and exhaust systems) the problem is dominated by convection, making viscous phenomena negligible [11]. Consequently the WAF Riemann solver has been exploited to calculate the transport term of Eq. 1-3.

Considering the conservation of a generic property ϕ :

$$\frac{\partial \rho \phi}{\partial t} + \nabla \cdot (\rho \phi \vec{U}) = 0, \quad (14)$$

the transport term is integrated over the cell volume and, by means of the Gauss theorem, reduced to a surface integral:

$$\oint_S \rho \phi \vec{U} \cdot \vec{n} dS = \sum_{f=1}^n \rho \phi_f U_f S_f. \quad (15)$$

On the basis of the finite volume discretization, the surface integral becomes a summation of the volumetric fluxes of the conserved variables through the intercell faces, evaluated at the face center (Eq. 15). Moreover, instead of recurring to different interpolation techniques, the HLLC and the WAF Riemann solvers have been exploited to evaluate directly the volumetric flux at the intercell face:

$$\sum_{f=1}^n \rho \phi_f U_f S_f = \sum_{f=1}^n F_f^{HLLC, WAF}. \quad (16)$$

For each intercell face it has been defined an owner cell and a neighbor cell. According to this convention, the Riemann problem has been located at the center of the intercell face, so that the owner cell centroid becomes the left state and the neighbor the right one, as shown in Figure 2. The velocity vector $\vec{U}_f$ has been decomposed into two components: one a normal to the cell face, $\vec{U}_n$, and another laying in the face plane, $\vec{U}_t$. The former is responsible of the net flux across the face of the conserved variable the latter, giving no contribution to this flux, is advected as a fluid property:

$$\begin{aligned} \vec{U}_n &= \vec{n}(\vec{U}_f \cdot \vec{n}), \\ \vec{U}_t &= \vec{U}_f - \vec{n}(\vec{U}_f \cdot \vec{n}). \end{aligned} \quad (17)$$

NETWORK OF PIPES

In a previous work the authors [12] pointed out that in particular cases the fully 1D approach fails in predicting the silencer performances, due to non-planar wave motion or transversal resonance effects which can only be captured by means of multidimensional CFD models. The case of the expansion chamber with an offset between the inlet and the outlet pipes (as shown in Figure 3) is the most clarifying example. This limitation can be overcome only by the adoption of 1D-multiD coupling techniques, as will be shown in this work.

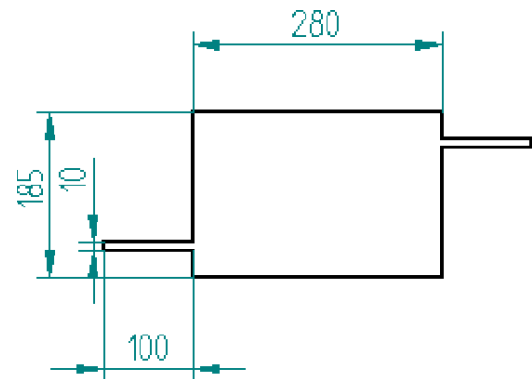


Figure 3: Expansion chamber with offset inlet and outlet pipes.

However, a different approach has been described by other researchers, which can be classified halfway between the fully 1D and the CFD calculations. The idea is to study complex muffler geometries generating 2D or 3D equivalent schematics of the muffler recurring to a grid of 1D elements. A similar method was also adopted by the authors to model geometries characterized by evident multidimensional effects, for instance the Diesel Particulate Filter [13]. As shown in Figure 4, the expansion chamber with offset inlet and

outlet pipes has been modeled with an acoustically equivalent 2D network of 1D elements, each one representing a portion of the muffler.

The generation procedure of such a configuration can require a considerable effort, especially if it has to be done by hand. Commercial codes, however, are provided with fully automatic procedures to generate the schematic starting from a CAD or STL file. Although this approach allows to preserve the global geometry, namely the muffler volume and shape, it may yield to an inconsistency with the real flow path covered by the gas, or the pressure wave, when it travels inside the expansion chamber.

Figure 4 shows, along with the equivalent 2D schematic of the muffler, two different pathways, from the inlet (point 1) to the outlet (point 2), which can be covered by the gas flow, namely path A and path B. Both of them represent the shortest way available for the wave to get from the inlet to the outlet of the expansion chamber. As can be noticed all the paths from point 1 to point 2 have the same length, the issue, however, is that this length is much longer than the real one traveled by the wave. This means that some resonance effects cannot be correctly captured since the main variable, the length of the chamber, does not correspond to the real one. A clear example, as shown in Figure 5, is the first passband frequency, which is dependent on the chamber length according to the well known expression $f_0 = a/2L$. The prediction given by the network of pipes calculates the first pass band at the frequency of 381Hz, while, as correctly predicted by the CFD simulation, it should occur at 612Hz. Hence the longitudinal resonance cannot be captured correctly, due to the longer path covered by the pressure waves. Besides, Figure 5 points out an interesting result, since the behavior of

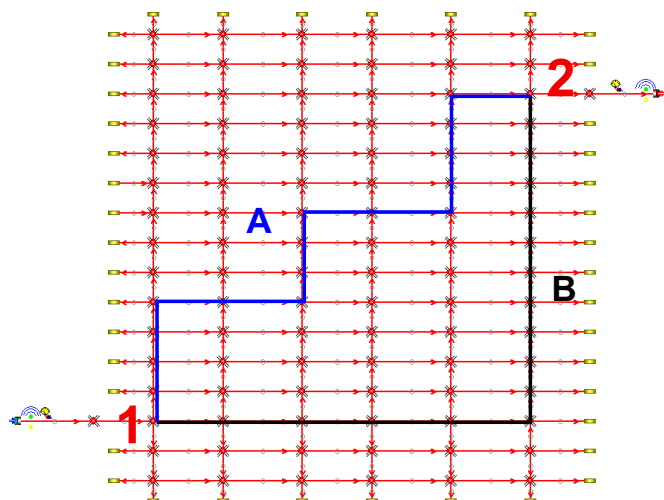


Figure 4: Equivalent model of the expansion chamber composed by a 1D pipe grid: the flow paths are highlighted.

the predicted transmission loss seems to capture the transversal resonance effects that the simple 1D approach could not predict. In particular, the two plots show some similarities: they have the same number of transparency and abatement peak frequencies, in addition they show the same trend on a different scale. Unfortunately they are not scaled by a constant factor, thus leading to the conclusion that an automatic procedure could easily fail in the attempt of generating the acoustically equivalent 1D model from the 2D or 3D geometry [14].

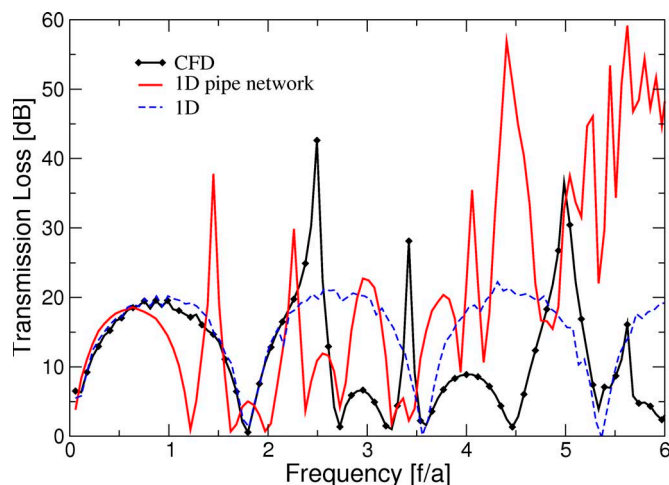


Figure 5: Offset chamber transmission loss: CFD versus fully 1D with pipe network calculations.

COUPLING METHOD

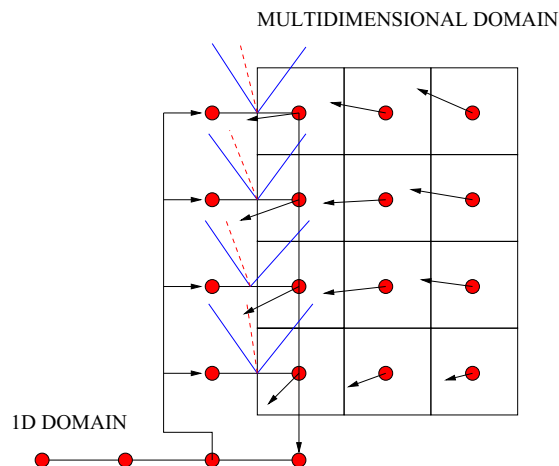


Figure 6: Procedure adopted for the integration of the two codes. The solution of the local Riemann problem is pointed out.

The attempt to adopt a net of 1D pipes to represent a 3D geometry to model a specific silencer characterized by a rather complex shape, can lead to configurations which may be rather different from the studied geometry but with similarities in the transmission loss. This way of proceeding, however, may reveal