

Advanced Computational Aero-Acoustic Simulation of Complex Automotive Exhaust Systems

2011-01-0503

Published
04/12/2011Julien Caradonna
Faurecia Emissions Control Technologies

Copyright © 2011 SAE International

doi:10.4271/2011-01-0503

ABSTRACT

In the automotive sector, the time to market has become increasingly important. Consequently, powertrain systems require specific exhaust systems solutions to meet engine performance, pollutant emissions and acoustic targets delivered in a shorten time period. In this context, exhaust system suppliers need to constantly update their development process and according to project demands, tail-pipe noise has to be managed with advanced tools and methodologies. Flow generated noise has a broad band character and depending on the product design, some tonal frequencies could appear and produce a whistling noise. In order to anticipate and solve all these sound quality problems, an innovative computational aeroacoustic methodology has been developed and validated for a large range of exhaust system products. The overall approach has the advantage of making possible an aeroacoustic analysis for complex industrial models in a realistic time framework since it relies upon fast and reliable CFD analysis and on propagation tools that capture most the meaningful physics. Besides the modularity of the approach is a key factor for real industrial application. The main purpose of this paper is to discuss an advanced technique for computational aeroacoustic simulations of exhaust systems products, combining a standard CFD method with an innovative acoustic generation and propagation approach. Results will be shown comparing numerical predictions to experimental data for isolated products, such as tail pipes and silencers, and also for an integrated complex exhaust system.

INTRODUCTION

Exhaust systems development procedures need to be continuously improved in terms of accuracy and speed. Acoustic management is one of the main functions of an exhaust line. Within the engine range of components dealing

with exhaust, the turbocharger is the main protagonist of the downsizing process. The turbocharger increases the mass flow rate and temperature throughout the exhaust system. As a consequence, exhaust gas increases the velocity magnitude and this leads to a rising of the Mach number, which is the dimensionless ratio between the gas velocity and the speed of sound of the propagating medium. Therefore the flow noise has an impact which can not be neglected, because the overall noise value is strongly dependent on the Mach number. Tail pipe noise is the sum of engine order noise and flow noise. Engine order noise is mainly generated by pulsating gas pressure gradient produced by opening and closing of the exhaust valves and it's characterized by a dominant firing order and its harmonics. Flow noise is instead produced by the impact between the flow and silencer internal components, and the jet noise, which is produced when the flow expands into atmosphere. In order to meet customer targets, tail pipe noise needs to be attenuated and modeled with the assistance of advanced simulation tools. This work describes an innovative approach that can be applied to the prediction of the flow noise generated by exhaust system components. The basic idea of the formulation combines a stochastic reconstruction of the aeroacoustic sources with a multi-domain discretization of the convective Helmholtz equation for sound propagation. In this paper the overall method will be introduced and described and some results will be shown comparing numerical calculations to experimental data for different exhaust system products, such as tail pipes and silencers, and also for an integrated complex exhaust system.

THEORETICAL CONTEXT

The simulation of flow induced noise has been subject of a lot of investigations during several decades, and a large number of different methods have been studied with time. Two big families of approaches can be identified. The first

one consists in the estimation of the acoustic sources, the acoustic propagation, and the flow field at the same time with advanced CFD analyses, while in the second one the noise generation and propagation are evaluated at different times with different tools using some kind of acoustic analogy. The first approaches, usually referred as CAA, are principally capable to capture at the same time both aerodynamics and acoustics and so can in principle model all the complex interaction effects, but are severely limited in their practical application by a number of different problems generated mainly by the large difference in the length and time scales of noise generation and propagation phenomena. Consequently CAA is never used for the full computational domain but it is used only in a small region where noise generation and non linear propagation effects can be important and then is coupled with an integral method (permeable FWH, Kirchhoff) for sound propagation in the rest of the domain up to the observer. Even with this approach the CAA calculations are limited to very simple problems and therefore different levels of CFD simulations have been introduced, such as LES or DES formulations, with the target to numerically capture only what is required for the specific problem and model the rest reducing in this way the computational efforts, but in any case almost any practical 3D problems of any industrial interest remain not affordable with these kind of approaches.

ACOUSTIC SOURCES RECONSTRUCTION

In the last decade there has been a new interest towards acoustic analogies approaches in particular in the context of stochastic approaches. The main idea of the acoustic analogy assumes that the acoustic field cannot modify the aerodynamic field, and so the two can be described with different tools. The fluid dynamic differential equations are therefore recast in a form in which we have on the left hand side a wave like operator that can be solved efficiently and on the right hand side we have instead a source term that is assumed to depend only on the fluid dynamic quantities. The approach applied in the present work, which has been implemented in the commercial platform VNoise, distributed by Scientific and Technical Software, is based on a classical stochastic approach like SNGR for source reconstruction. The basic idea of the stochastic approaches is to reconstruct the full spectra of the unsteady source terms starting from a simple steady state CFD RANS analysis that provides mean values for key parameters such as turbulent kinetic energy and other gas dynamic quantities. The average fluid quantities are then used to reconstruct the unsteady fluid quantities generating a number of Fourier components with random phase and direction and amplitudes designed in such a way to reconstruct the mean quantities provided by the RANS analysis. Once the sources are available they are usually inserted in linearized Euler equations or acoustic

perturbation equations to propagate and to consider the effect of the presence of solid boundaries.

NOISE PROPAGATION

This formulation is based on a classical stochastic approach like SNGR for source reconstruction, but differs from the previous works for the way in which the sources are propagated once they have been reconstructed. The use of the linearized Euler equation presents in fact some drawbacks that have severely limited the applicability of these kinds of approaches. It is well known that linearized Euler equations are subject to instability waves that can become much larger than the true acoustic field and that make therefore the method not usable especially in an industrial environment where the reliability of any simulation tool is a fundamental requirement. These problems can be partially attenuated by the use of the acoustic perturbation equations (APE) introduced by Ewert, that are a sort of modified Euler equation obtained by dropping some eigenvalues components that can more easily generate the instability waves. It can be demonstrated that the APE are much less subject to stability problems, but on the other side they lose some of the physics contained in the linearized Euler equation becoming substantially equivalent to a standard convective scalar wave equation, while maintaining the high computational cost. The approach followed in the present works is instead different since the basic idea is to try to use a scalar convective wave equation in the frequency domain in order to perform propagation. This presents several potential advantages in respect of time domain approaches based on linearized Euler or APE. First of all the convective scalar wave equations are completely free of any instability and can be solved much more efficiently than APE or linearized Euler. Obviously the convective scalar wave equation loses the capability to describe some physical phenomena that can be captured by linearized Euler, but it actually contains most of the physics required to describe a wide range of phenomena. Besides the results obtained are substantially equivalent to what can be obtained with the various APE at a much reduced cost. So this approach can be potentially very advantageous in an industrial environment having the capability to capture a wide range of phenomena at a reduced computational cost and without any stability or reliability problem. CFD results are used as a starting point for flow noise simulations. In order to permit to use a scalar convective wave equation as a propagation tool for the reconstructed sources, a new source coupling boundary condition has been developed. According to the gas dynamic field the computational domain is divided in several independent domains as sketched in [figure 1](#) below, that shows a duct with an incoming flow that impinges on a generic obstacle producing and radiating noise.

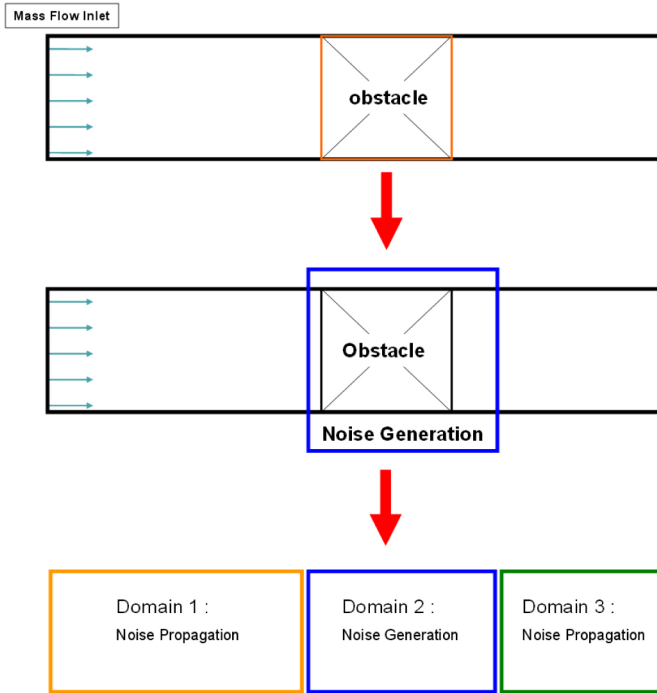


Figure 1. Domain subdivision process

As a first step we identify three different domains playing different roles. In particular in the central domain we have most of the generated acoustic energy while in the other two domains we have substantially only propagation effects.

We consider for each of the three domains the convective wave equation written in terms of pressure:

$$(1 - M^2) \nabla^2 p + k^2 p - 2ik\bar{M} \cdot \nabla p = 0 \quad (1)$$

The three domains are independently discretized. We can use both an integral (BEM) or a field approach (FEM) to discretize the three domains since the essence of the formulation remain unchanged.

We define the total pressure p^i of domain i as being the sum of two components: being p_s^i the pressure due to the aero-acoustic sources placed inside the domain i , and p_r^i is the rest of the pressure.

$$p^i = p_r^i + p_s^i \quad (2)$$

It is important to note that p_s^i is defined as the *free-field* pressure generated by the sources inside the i^{th} domain, in other words it is the pressure that will be generated by the

sources inside domain i^{th} in the absence of any boundary wall.

It is therefore possible to evaluate p_s^i using for example the Lighthill's acoustic analogy writing:

$$p_s^i = \int_V G \frac{\partial^2 T_{ij}}{\partial x_i \partial x_j} dV \quad (3)$$

Being G the convective green functions. Since p_s^i naturally satisfies eq (1), this can be rewritten for each domain only in terms of pressure p_r^i as:

$$(1 - M^2) \nabla^2 p_r + k^2 p_r - 2ik\bar{M} \cdot \nabla p_r = 0 \quad (4)$$

It is therefore possible to solve for each domain equation (4) with the proper boundary conditions. In particular we can have two kind of surfaces on which apply different boundary conditions, the physical walls that delimit the domain and the interface surfaces between two domains.

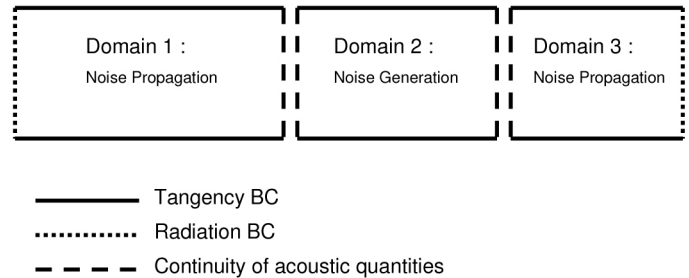


Figure 2. Boundary conditions for the three domains. Solid Thick lines represent surfaces on which tangency BC is applied, dashed thin lines are fictitious surfaces on which continuity of the acoustic quantities applied and that are here represented with the source coupling BC, and dotted lines are physical boundaries on which other BC apply rather than tangency (i.e. radiation)

For example considering the source domain (domain 2) showed in figure 2 above we have two following boundary conditions:

- Tangency BC on physical surfaces (thick lines):

$$\frac{\partial p_r^2}{\partial n} = -\frac{\partial p_s^2}{\partial n} \quad (5)$$

- Source coupling BC on interfaces between domains (dashed thin lines). For example for interface between domains 1 and

2 the continuity of the acoustic quantities (pressure and velocity) writes as:

$$\frac{\partial p_r^2}{\partial n} + \frac{\partial p_s^2}{\partial n} = - \left(\frac{\partial p_r^1}{\partial n} + \frac{\partial p_s^1}{\partial n} \right) \quad (6)$$

$$p_r^2 + p_s^2 = p_r^1 + p_s^1 \quad (7)$$

Where [equations \(6\)](#) and [\(7\)](#) represent respectively continuity of the acoustic velocity and the pressure (The minus in [eq \(6\)](#) is due to the change of normal orientation for interface between domains 1 and 2). Similar relations can be written between domains 2 and 3.

EXPERIMENTAL SET-UP

To validate the results obtained with the proposed methodology, tests were performed on a cold flow noise bench. This bench is located at the Test Department of Faurecia Emissions Control Technologies R&D Center based in Bavans, France.

The test rig allows measuring the sound pressure level and the noise spectrum generated by the test part at one or more microphone positions simultaneously. In the present paper results are analysed within the frequency range included between 400 Hz and 10.000 Hz. This range permits to obtain an accurate evaluation of the investigated physical phenomenon.

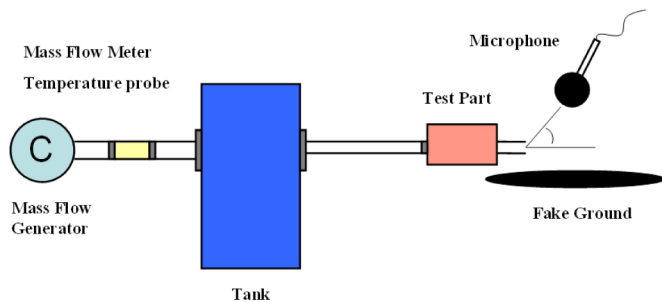


Figure 3. Sketch of the measurement setup

A volumetric compressor generates the mass flow rate, while mass flow and temperature measurements ensure the respect of the requested operating point. All pulsating phenomena are completely damped by the relaxation tank placed between the compressor and the test part. In order to take into consideration all the reflective effects a fake ground is placed at 28 cm behind the tail pipe origin point. One or more microphones are located at customer standard positions for the frequency spectrum and overall level measurement.



Figure 4. General overview of the experimental test set-up for the silencer

Table1. Microphone standard positions and related space locations

n°	Distance [cm]	Plane	Angle [deg]
1	14	Horizontal	45
2	15	Horizontal	0
3	25	Vertical	45
4	50	Vertical	45

Two families of microphone positions can be identified. Microphones n° 1 and n° 2 are considered as near field microphones, while n°3 and n° 4 are considered as far field microphones. Simulations and experimental results will be shown in two different graphs, the first one related to a near field microphone (n°1) and the second one related to a far field microphone(n°4).

RESULTS AND CORRELATIONS

In order to validate the proposed approach a simple application case has been submitted to the complete simulation loop. The computational fluid dynamics domain is composed of a straight pipe with a valve placed inside. The valve represents an obstacle, generating sources of noise. Computational fluid dynamics simulations have been carried out using the commercial software STAR-CD, version 3.26, distributed by CD-adapco. CFD results have been imported in the commercial code VNoise, which implemented the method described in [figure 1](#), obtaining a source domain in which the turbulent kinetic energy has an important component, and