Simulation Modelling Practice and Theory 41 (2014) 73–86

Contents lists available at ScienceDirect

Simulation Modelling Practice and Theory

journal homepage: www.elsevier.com/locate/simpat

Simulation and experimental validation of the flow field at the entrance and within the filter housing of a production spark-ignition engine

Carmine De Bartolo, Angelo Algieri *, Sergio Bova

Department of Mechanical, Energy and Management Engineering – DIMEG, University of Calabria, Via P. Bucci – Cubo 46C, 87036 Arcavacata di Rende, CS, Italy

A R T I C L E   I N F O

Article history:
Received 13 July 2013
Received in revised form 23 October 2013
Accepted 21 November 2013
Available online 13 December 2013

Keywords:
Intake system
Air filter
Internal combustion engines
Porous medium model
CFD
Laser Doppler Anemometry

A B S T R A C T

The paper aims to analyse the flow field at the entrance and within the filter housing of a production four-cylinder, spark-ignition engine during the intake phase. To this purpose a computational fluid dynamic (CFD) analysis was carried out adopting a finite volume code, while an experimental activity was performed at a steady flow rig to validate the computational model.

The comparison between numerical data and experimental measurements showed a good agreement and it demonstrated the capabilities of the proposed CFD model to predict in detail the flow field within a complex production automobile component that largely influences the efficiency and reliability of actual internal combustion engines (ICEs).

1. Introduction

Nowadays, a profound understanding of the intake system of internal combustion engines is fundamental to design, develop and optimize new high efficiency engines [1–5]. The improvement of the breathing efficiency, in fact, guarantees both a reduction in emissions and in fuel consumption (to meet the more and more severe regulations on exhaust emissions) and a significant increase in engine performances.

The variety and complexity of fluid dynamic phenomena that take place within the modern engine intake systems put forward the advantage of a design and an optimisation that are based on an integrated numerical–experimental methodology.

The study of the ICEs by means of computational fluid dynamics (CFD) is a widespread technique due to the relevant increase in the computational power and the availability of commercial codes able to analyse flow field in complex geometries [6–11]. Theoretically, all the flows can be simulated by solving the Navier–Stokes equations exactly. However the direct numerical simulation (DNS) technique can be used only for low Reynolds number, owing to the high computational efforts to resolve the whole spectrum of turbulent scales. Alternative approaches to DNS technique are Large Eddy Simulations (LES) and Detached Eddy Simulations (DES) [12–14]. Nevertheless, these models turn out to be highly time consuming when the analysis has to be extended to the industrial applications because of the more complex nature of both the geometry and the flow. For these reasons three-dimensional fluid dynamic models in ICEs are often based on Reynolds Averaged Navier Stokes equations (RANS), due to the higher robustness and to the lower computational cost

* Corresponding author. Tel.: +39 0984 494665; fax: +39 0984 494673.
E-mail addresses: c.debartolo@unical.it (C. De Bartolo), a.algieri@unical.it (A. Algieri), sergio.bova@unical.it (S. Bova).

1569-190X/$ - see front matter
http://dx.doi.org/10.1016/j.simpat.2013.11.012

Nomenclature

Symbols

B	bore
D	diameter
I	turbulence intensity
k	turbulent kinetic energy
L	stroke
$\dot{m}$	mass flow rate
p	pressure
R	inertial resistance coefficient
U	flow velocity
α	permeability of the medium
ε	dissipation rate
λ	laser wavelength
$\Delta\eta$	porous medium thickness
Δp	pressure drop
μ	dynamic viscosity
ρ	fluid density
τ	Reynolds stress

Acronyms

CFD	computational fluid dynamics
DES	Detached Eddy Simulations
DNS	direct numerical simulation
LDA	Laser Doppler Anemometry
LES	Large Eddy Simulation
ICE	internal combustion engine
PIV	Particle Image Velocimetry
RANS	Reynolds Averaged Navier Stokes

[15–18]. In this framework all the turbulence scales are modelled, with considerable reduction in the computational effort but, also, in the numerical accuracy. The shortcomings of RANS models for ICE applications have been broadly discussed in the literatures [19–22] and their numerical validation is necessary especially for industrial applications.

At the same time, steady flow testing is a widely adopted procedure to study the fundamentals of the intake process, to analyse complex flow fields and to validate numerical codes, owing to its relative simplicity and the proper simulation of the real intake phase [23–27]. To this purpose, mass flow rate and pressure drop measurements are adopted to provide global information on engine head breathability [28–30], while Laser Doppler Anemometry (LDA) [31,32] and Particle Image Velocimetry (PIV) [33,34] techniques are used to define the fluid dynamic behaviour of internal combustion engines.

The analysis of the relations between the fluid dynamic behaviour of each individual element and its geometry is fundamental to clearly identify the individual contributions of each element to the overall engine performance [35–38]. This process takes place within a general vehicle development programme, based on a systematic tuning of the components that optimise the engine performances and breathing characteristics. In the last few years, several studies have been carried out to analyse in detail the in-cylinder flow during the intake phase and to evaluate its dependence on the geometry and the operating conditions of the engines [39–41]. Conversely, few quantitative studies have been performed on other components that largely influence the efficiency and reliability of actual ICEs.

Specifically, a detailed fluid dynamic characterisation of air cleaner system is a fundamental key to improve engine performance and durability, and to reduce the noise emission and the dust concentration to an acceptable level [42–44].

Often, underhood space utilisation represents the major constraint in designing the filtering system [42]. As a consequence, the filter housing geometry tends to produce a very complex flow field, with large-scale separation and recirculation [43], and high aerosol velocities through the primary filter element. Specifically, non-uniform flow distributions significantly reduce the filter efficiencies [44] and could produce propagation of pressure waves through the system with noise emission to the surrounding environment [45]. Furthermore, high velocities may cause dust re-entrainment and increase the amount of dust penetrating the filter [46].