International Journal of Heat and Fluid Flow xxx (xxxx) xxx–xxx

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Direct numerical simulation of the flow in the intake pipe of an internal combustion engine

G.K. Giannakopoulos*[a], C.E. Frouzakis[a], K. Boulouchos[a], P.F. Fischer[b,c], A.G. Tomboulides[c,d]

[a] Aerothermochemistry and Combustion Systems Laboratory, ETH Zurich, Switzerland
[b] Department of Computer Science, University of Illinois, Urbana-Champaign, IL, U.S.A.
[c] Mathematics and Computer Science Division, Argonne National Laboratory, Chicago, IL, U.S.A.
[d] Department of Mechanical Engineering, Aristotle University of Thessaloniki, Greece

ARTICLE INFO

Keywords:
Direct numerical simulation
IC engine intake pipe
Entry flow in curved pipe
Flow around valve
Spectral element method

ABSTRACT

The incompressible flow in the intake pipe of a laboratory-scale internal combustion engine at Reynolds numbers corresponding to realistic operating conditions was studied with the help of direct numerical simulations. The mass flow through the curved pipe remained constant and the valve was held fixed at its halfway-open position, as is typically done in steady flow engine test bench experiments for the optimization of the intake manifold. The flow features were identified as the flow evolves in the curved intake pipe and interacts with the cylindrical valve stem. The sensitivity of the flow development on the velocity profile imposed at the inflow boundary was assessed. It was found that the flow can become turbulent very quickly depending on the inflow profile imposed at the pipe inlet, even though no additional noise was added to mimic turbulent velocity fluctuations. The transition to turbulence results from competing and interacting instability mechanisms both at the inner curved part of the intake pipe and at the valve stem wake. Azimuthal variations in the local mass flow exiting the intake pipe were identified, in agreement with previously reported measurement results, which are known to play an important role in the charging motion inside the cylinder of an internal combustion engine.

1. Introduction

The gas motion in the interior of internal combustion engines (ICE) has a strong impact on the efficiency and engine-out emissions, through its effect on mixing, heat transfer and combustion processes (Heywood, 1988; Lumley, 1999). In the continuing effort to improve the efficiency and reduce pollutant emissions to conform with stringent environmental regulations and to introduce novel combustion concepts, the optimization of the geometry of the intake and exhaust ports/valves, piston and cylinder has become an essential part of ICE development. In particular, the flow through the intake ports can affect the cylinder charge filling and produce variations in the in-cylinder flow pattern, which can eventually contribute to increased cycle-to-cycle variability (CCV).

Typically, the study and optimization of the intake manifold aiming at enhancing the generation of the in-cylinder swirling motion, and the measurement of the valve's discharge coefficient, is carried out in steady flow engine test benches at fixed valve positions (e.g. Jones, 1966; Heim and Ghandhi, 2011). Despite recent progress in optical diagnostics techniques, the inability to optically access the inlet port and valve regions and limitations in scanning of the flow field with sufficient spatial resolution, restrict the detailed investigation of the flow in these areas. Laser Doppler anemometry (LDA) using acrylic plastic replicas of the engine with liquid (Tindal et al., 1988) or air (Bicen et al., 1985) as the working fluid have been employed in the experiments. Hot-wire anemometry has also been used to provide the three velocity components at the exit plane of the intake valve in a steady-flow rig (Khalighi et al., 1986), but the interference of the hot-wire probe with the flow makes this technique a less attractive choice. In a recent study, (Freudenhammer et al., 2014) employed an advanced non-intrusive diagnostic technique based on magnetic resonance velocimetry (MRV) to investigate the volumetric flow within the intake and cylinder during flow induction in a polyamide model of a single-cylinder optical engine. Water was used as the working fluid and the measurements were performed at Reynolds numbers of 22,500 and 45,000 based on the bulk flow velocity and the inlet pipe diameter. The flow revealed significant variations in the radial velocity through the valve curtain, caused by large recirculation regions in the valve seat and downstream of the valve stem. The local mass flow entering the cylinder was found to decrease by 50% in these regions, thus having large implications on the cylinder charge filling and producing variations of the in-cylinder flow pattern.

* Corresponding author.
E-mail address: georgeg@lav.mavt.ethz.ch (G.K. Giannakopoulos).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2017.09.007
Received 14 May 2017; Received in revised form 13 September 2017; Accepted 13 September 2017
0142-727X/

From a more fundamental point of view, the complex flow in the intake pipe of an ICE can be considered as a combination of the flow through a curved pipe and that past a surface-mounted circular cylinder representing the valve stem. A fluid flowing through a curved pipe is subjected to centrifugal forces ($\sim U^2/R$, where U is the velocity and R the radius of curvature) which act stronger on the more rapidly flowing central part of the pipe than close to the walls. As a result, the location of the maximum axial velocity is deflected from the center towards the outer pipe wall. The generated adverse pressure gradient in combination with the centrifugal forces creates a secondary motion in the cross-stream plane, which results in the formation of axially-oriented counter-rotating cells. Depending on the Reynolds number, the ratio $\kappa = R_p/R_c$ between the pipe radius R_p and the radius of the pipe centerline R_c, and the velocity profile at the curved segment entry point, different secondary motion patterns can be observed.

The secondary flow development in curved pipes has been studied extensively, mainly due to its relevance to many industrial applications, biological flows (blood flow through the cardiovascular system), and flow in rivers. For comprehensive reviews on this flow the reader is referred to Berger et al. (1983); Ito (1987) and the references therein.

Dean (1927) was the first to study theoretically the fully developed laminar flow in curved pipes with small curvature and showed that the relation between the flow rate and the pipe curvature depends on a single parameter, later called the Dean number $De = Re\sqrt{\kappa}$, where the Reynolds number Re is based on the pipe cross-section diameter and - the bulk velocity. A solution based on a series expansion in terms of De was derived which demonstrated a flow field exhibiting a pair of symmetrical, axially-oriented counter-rotating vortices, known as Dean vortices. Over the years, the work was extended to study flows at larger De (Dennis and Ng, 1982; Yanase et al., 1989) and curvature ratios (Siggers and Waters, 2005). It was found that there exists a critical Dean number above which, in addition to the symmetric pair of counter-rotating Dean vortices, a secondary flow pattern consisting of additional vortex pairs can appear. Other studies have focused on the investigation of the laminar entry flow into curved pipes imposing a parabolic (Bovendeerd et al., 1987) or uniform (Agrawal et al., 1978; Soh and Berger, 1984) velocity profile at the inlet. It was found that the inflow condition affects the distance required for the flow to reach a fully developed state, and significant differences in the secondary flow structures were observed before reaching this state.

The continuously increasing computational capabilities has allowed the in-depth study of transitional and turbulent flows in curved and toroidal pipes by means of large-eddy (LES) and direct numerical simulations (DNS). Boersma and Nieuwstadt (1996) performed LES, while Hüttl and Friedrich (2000, 2001) employed DNS to study the influence of curvature on the mean and fluctuating flow. It was found that the turbulent fluctuations are drastically reduced by curvature compared to the flow in a straight pipe. A flow regime map was proposed by Piazza and Ciofalo (2011) based on the Reynolds number and the curvature parameter κ, delineating regimes with laminar, fully-turbulent and quasi-periodic secondary flows. The effect of curvature on the stability of the flow in toroidal pipes was studied by Canton et al. (2015) using linear stability analysis, which showed that the critical modes are located in the region of the Dean vortices. A recent study by Noorani et al. (2013) considering Reynolds numbers up to $Re = 11,700$ and different values of κ reported the full Reynolds stress tensor, as well as kinetic energy budgets and friction factors.

In the flow past a surface-mounted cylinder, the recirculation bubble on the windward side of the cylinder creates an adverse pressure gradient on the flow and leads to the thickening of the boundary layer and a three-dimensional separation. The lower regions of the separated boundary layer roll up to form a number of coherent horseshoe vortices which wrap around the cylinder (Baker, 1979; 1980). The structure, number and stationarity of the horseshoe vortices around circular cylinders has been extensively studied in wall-bounded domains by Pattenden et al. (2007) and Kirkil and Constantinescu (2015).

Despite its importance on the in-cylinder flow development, only a few numerical studies have been devoted to the detailed investigation of the flow in the intake pipe of an ICE. Inagaki et al. (2010) used large eddy simulations to analyze steady intake flows in an engine-like geometry consisting of an intake port, a fixed valve and a cylinder and compare it against the predictions of two RANS models and LDV measurements in the cylinder and upstream of the intake bend. The LES model provided better agreement with the measurements in the cylinder, but the details of the flow in the intake could only be qualitatively compared to the RANS models. Most studies focused on the evolution and the characterization of the turbulent flow field inside the cylinder during (usually motored) engine operation. In this work, high-fidelity direct numerical simulations of the flow in the intake pipe of the laboratory-scale TCC-III engine (Schiffmann et al., 2016) at realistic operating conditions is performed. The complex evolution of the velocity and vorticity fields is investigated as the fluid flows through the pipe and interacts with the valve stem before entering the cylinder. Three Reynolds numbers ($Re = 16,000, 33,000$ and 50,000) are considered and the velocity profile at the pipe inlet is assumed to take a parabolic or a power-law profile.

Although the configuration studied is the real geometry of the TCC-III engine intake pipe, the aim of this paper is the fundamental understanding of the flow physics under two well defined inflow velocity boundary conditions, which are not realistic for the flow at the considered Re. However, in addition to the insights gained for a complex geometry that is relevant for applications and is not typically investigated by DNS, the generated high-quality database will be helpful for the validation of existing engineering RANS and LES models.

The paper is structured as follows: following the presentation of the geometry and the numerical approach in Section 2, the main results in Section 3 start with the discussion of the effect of the inflow velocity profile and the Reynolds number on the main flow structures quantified through the vorticity field, and continue with the quantitative description of the secondary flow upstream and downstream of the valve stem. The conclusions are summarized in Section 4.

2. Methodology

The conservation equations of mass and momentum are discretized in space using the spectral element method (SEM), a high-order method weighted-residual technique for the solution of partial differential equations (Patera, 1984; Deville et al., 2002). The computational domain is discretized into curved conforming hexahedral elements and the solution is expanded into Nth order tensor product Lagrange polynomials, based on the Gauss–Lobatto–Legendre (GLL) quadrature points in each element. The SEM offers both the geometric flexibility needed to accommodate the complex geometries of ICEs and the high-order accuracy of spectral methods make it very well suited for the study of laminar, transitional and turbulent flows (c.f. Fischer et al., 2002). The discretized equations are integrated in time with Nek5000 (Fischer et al., 2008), a highly-efficient and massively-parallel incompressible flow solver. This choice is justified since the maximum calculated Mach number is $Ma \approx 0.1$ and the flow can be considered as incompressible. The code has been validated for the multi-cycle flow in a fixed-valve, moving-piston assembly by comparison with experimental data (Schmitt et al., 2014a,b) and has been used to study the effect of compression on the flow, mixing and heat transfer to the cylinder walls (Schmitt et al., 2015a,b).

The computational domain consists of the intake runner of the TCC-III engine (Schiffmann et al., 2016), a circular cross-section 90° elbow with a diameter $D = 25.4$ mm (Fig. 1a) and centerline pipe radius of $R_c = 60$ mm. The domain includes the valve body, with a valve stem of diameter $d_s = 6.99$ mm at the top and the upper valve surface at the bottom. As already noted, the valve is fixed at its half-way open position, at a distance 4.45 mm away from the valve seat. The computational mesh was generated using the (CUBIT, 2015) geometry and mesh