Computers & Fluids 105 (2014) 113–124

Contents lists available at ScienceDirect

Computers & Fluids

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

A comprehensive modeling study of in-cylinder fluid flows in a high-swirl, light-duty optical diesel engine

Federico Perini [a,*], Paul C. Miles [b], Rolf D. Reitz [a]

[a] Engine Research Center, University of Wisconsin–Madison, Madison, WI 53706, United States
[b] Combustion Research Facility, Sandia National Laboratories, Livermore, CA 94550, United States

ARTICLE INFO

Article history:
Received 14 September 2013
Received in revised form 1 August 2014
Accepted 3 September 2014
Available online 16 September 2014

Keywords:
Engine CFD
Swirl modeling
Light-duty diesel engine
PIV
Swirl center
RANS
KIVA

ABSTRACT

The effectiveness of computational fluid dynamics modeling as a tool for researching fuel-lean, low temperature engine combustion strategies relies on its capability to capture the local fluid flow properties that affect spray dynamics, mixture preparation and ignition kinetics. In this study, a comprehensive model of an optically accessible, single-cylinder light-duty diesel engine was developed for engine combustion research. The computational model includes the realistic combustion chamber and ducts geometries. Variable orientation throttles in the intake ducts were modeled to reproduce variable swirl generation. Full induction stroke calculations were run over a portfolio of intake swirl conditions, and validated against extensive measurements featuring global intake swirl ratios, in-cylinder particle image velocimetry (PIV)-measured velocity fields and swirl centers. The results showed good agreement with the experiments, and the model was used to understand the effects of different swirl generation strategies on the in-cylinder flow field. The implications of using simplified sector mesh geometries on the predictiveness of in-cylinder flow and turbulence quantities are described.

1. Introduction

Energy efficiency and environmental sustainability concerns are driving internal combustion engine research towards looking for more effective combustion strategies that can maintain extremely low emissions levels in-cylinder, while achieving very high indicated efficiencies [1]. Homogeneous-charge, partially-premixed and reactivity-controlled compression ignition (HCCI, PPCI, RCCI) combustion strategies rely on forming locally ignitable mixtures at typically highly dilute and low temperature limits [2]. All of these strategies present a number of challenges from the modeling point of view. In partially-premixed compression ignition (PPCI) combustion, in particular, fuel is directly injected into the combustion chamber earlier than in conventional diesel combustion modes, to allow for premixed mixture preparation prior to ignition. Dilute, low-temperature charge conditions and high swirl ratios enable the feasibility of this combustion mode, allowing time for some degree of premixing to be achieved for all the fuel; however, the heterogeneous presence of locally fuel-lean and locally fuel-rich mixtures at the ignition can lead to significant engine-out carbon monoxide (CO) and unburned hydrocarbons (UHC) emissions arising from bulk overly-lean regions that do not completely ignite [3].

High-speed, direct-injection diesel engines (HSDI) are particularly suitable for applying low-load, partially premixed combustion strategies, due to their swirl-tailored-intake-duct designs that allow high swirl ratios to be achieved within the combustion chamber, fostering fuel–air mixing before ignition of the richest regions occurs [4,5]. The light-duty optical diesel engine facility at the Sandia National Laboratories (SNL) has provided significant insight into the mechanisms that rule the sources of UHC and CO emissions arising from low-load PPC combustion [3–13]. In the studies, a Planar Laser Induced Fluorescence (PLIF) based experimental technique was developed to analyze local mixture formation by evaluating the local distribution of equivalence ratios throughout most of the combustion chamber volume. Experiments were performed at different horizontal planes that spanned the squish region, the piston rim region where jet impingement occurs, and the deep in-bowl region which is affected by bowl shape design. A comprehensive review of these local equivalence ratio measurements for PPC engine operation can be found in [14]. Detailed studies on the fluid flow properties of this engine facility were carried out by Opat [15] and Petersen and Miles [16]. These comprehensive experimental studies provide a unique quantitative validation opportunity for engine computational fluid dynamics (CFD) models.

---

* Corresponding author. Tel.: +1 608 658 0985.
E-mail address: perini@wisc.edu (F. Perini).

http://dx.doi.org/10.1016/j.compfluid.2014.09.011

Widely adopted sector mesh engine simulations have not been able to capture local mixture preparation and jet-to-jet differences that are responsible for most of UHC and CO emissions [5,17], even when excellently reproducing the overall in-cylinder ignition behavior. For this reason, the focus of the present study was a thorough description of the engine fluid flow properties, using a comprehensive computational model of the experimental engine facility. Validations are performed against flow-bench swirl and in-cylinder PIV measurements. The importance of correct fluid flow modeling for high-speed, direct-injected diesel engines, and the usefulness of PIV measurements as a tool for understanding the fluid flow structures and to support the modeling, are well established [18–24]. The model was used to study the implications of the interactions between the flow field and temperature stratification on the formation of pollutants in globally lean, low-temperature combustion strategies.

The paper is structured as follows. First, the computational model is described with a focus on the modeling of the variable-swirl generation throttles. Then the fluid flow predictions at a number of swirl conditions against measurements of swirl ratios, local velocities, and swirl center positions, are validated for full induction stroke calculations in motored engine operation. Finally, the validated model is used to study the effects of different throttling strategies on the in-cylinder fluid flow structure, and compared with sector mesh simulations. The results show the good accuracy of the Reynolds-averaged Navier Stokes (RANS) approach in capturing ensemble-averaged flow field properties, the significant differences in the average velocity field caused by different throttling strategies, and the substantial lack of resolution of simplified sector mesh modeling at capturing local average and turbulent quantities.

2. Engine model development

The engine is derived from a modified 1.9L General Motors (GM) production engine. The optical piston assembly retains the full geometric properties of the production piston, with slight differences in the top land ring crevice volume, fitted for experimental imagery [7]. As a basis for the fluid flow calculations, an unstructured version of the KIVA family of codes was used [25–27]. The code solves the RANS ensemble-averaged Navier–Stokes equations on complex engine geometries using an arbitrary Lagrangian–Eulerian (ALE) scheme. Fluid flow turbulence is accounted for by means of the ReNormalization Group (RNG) closure for the two-equation $k-\epsilon$ model [28]. In order to match the non-negligible compressibility of the optical piston assembly, a previously validated static compressibility model for the connecting rod was used [5]. The code also features additional models for efficient combustion chemistry calculations, including a sparse analytical Jacobian chemistry code for chemical kinetics (`SpeedCHEM`) [29,30] and a high-dimensional cell clustering algorithm [31,32].

The computational grid of the single cylinder engine features 559,867 cells and 585,189 vertices, and was developed based on the detailed geometry files of the combustion chamber, cylinder head, intake and exhaust ducts and runners, as shown in Fig. 1. The intake section was modeled up to the intake plenum, including the merging cross section ducts at the intake plenum flange. The chamfered piston geometry, including the bowl shape, valve cutouts and top land ring crevice was obtained from the optical piston assembly. The unstructured grid capability kept the number of cells in the discretized geometry as low as possible, while resolving the fluid flow in the near-valve regions and at the walls. Circular geometric features, including the piston bowl, cylinder liner, as well as the valves and their seats, were modeled making use of o-grid structures that allow body-fitted meshes with high-quality and optimal aspect-ratio cells that avoid non-convex or degenerate cells at the walls that could cause solution convergence problems. Finally, in order to achieve a numerically converged prediction of the intake mass flux through the valves, the code was modified to allow an arbitrary number of cell layers at the valve opening, to describe the valve skirt, so that the valve profiles could be faithfully

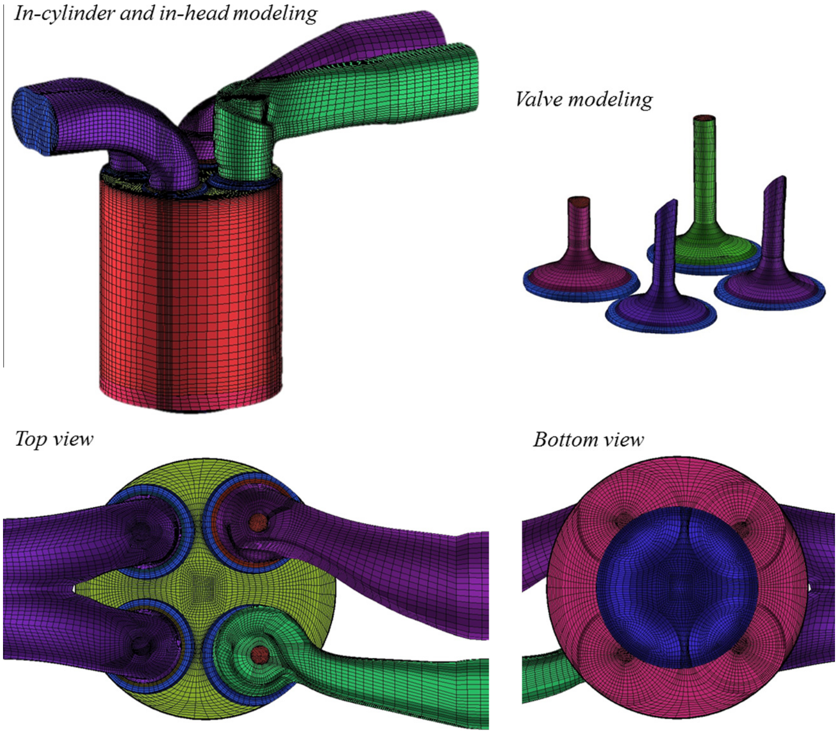

Fig. 1. Meshing details for the combustion chamber and head regions.