Swirling flow in a two-stroke marine diesel engine

Hemmingsen, Casper Schytte; Ingvorsen, Kristian Mark; Walther, Jens Honore; Mayer, Stefan

Published in:
Proceedings of the 26th Nordic Seminar on Computational Mechanics

Publication date:
2013

Document Version
Publisher's PDF, also known as Version of record

Citation (APA):
Hemmingsen, C. S., Ingvorsen, K. M., Walther, J. H., & Mayer, S. (2013). Swirling flow in a two-stroke marine diesel engine. In A. Logg, & K. A. Mardal (Eds.), Proceedings of the 26th Nordic Seminar on Computational Mechanics

Proceedings of the 26th Nordic Seminar on Computational Mechanics
A. Logg, K.A. Mardal (Eds.)
©Oslo, 2013

Swirling flow in a two-stroke marine diesel engine

Casper S. Hemmingsen[1], Kristian M. Ingvorsen[1], Jens H. Walther[1,2], and Stefan Mayer[3]

(1)Department of Mechanical Engineering, Technical University of Denmark, Denmark, jhw@mek.dtu.dk

(2) Computational Science and Engineering Laboratory, ETH Zürich, Switzerland, Walther@mavt.ethz.ch

(3)MAN Diesel & Turbo, Denmark, Stefan.mayer@man.eu

Summary. Computational fluid dynamic simulations are performed for the turbulent swirling flow in a scale model of a low-speed two-stroke diesel engine with a moving piston. The purpose of the work is to investigate the accuracy of different turbulence models including two-equation Reynolds-Averaged Navier-Stokes models and large eddy simulations. The numerical model represent the full three-dimensional geometry and the piston motion is modeled by compressing cells in the axial direction. The CFD predictions are compared to experimental results and a reasonable agreement is found.

Key words: CFD, RANS, LES, swirl, turbulence, two-stroke, marine diesel

Introduction

Low-speed two-stroke (LSTS) diesel engines are used to power the worlds largest marine vessels, such as tankers and container ships. When the piston approaches the bottom dead center (BDC) of the cylinder, it uncovers a series of angled scavenge ports in the cylinder wall. Fresh air is blown into the cylinder through the scavenge ports, thereby flushing the old combustion gas out through the exhaust valve located in the cylinder head. This gas exchange process is known as uniflow-scavenging. The angled ports induce a rotational motion to the incoming scavenge air, thereby creating a swirling flow.

Accurate computational fluid dynamics (CFD) simulations of the in-cylinder swirling flow is important for engine optimization and emission reduction. Recently, a database for CFD validation was established based on an experimental investigation of the flow in a dynamic scale model of a LSTS diesel engine [1]. The model has a moving piston but compression and combustion are neglected.

The purpose of the present work is to evaluate the accuracy of different turbulence models

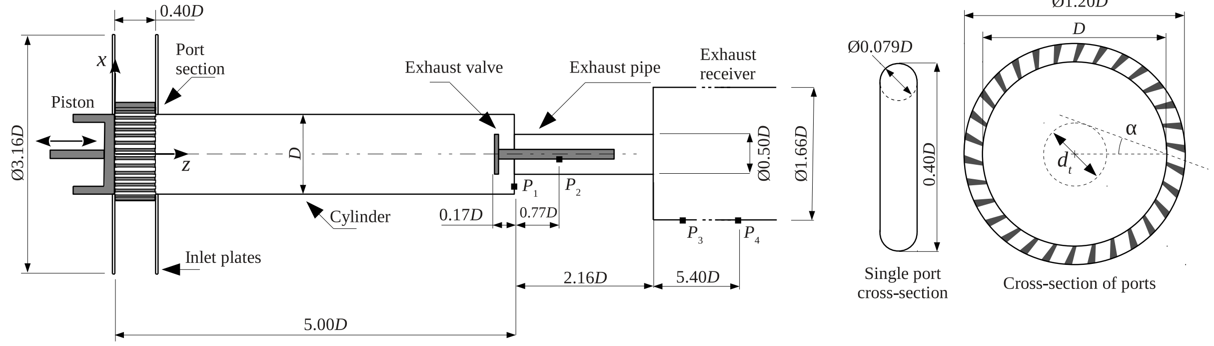

Figure 1. (a) Geometric of the main model, (b) details on the scavenge port geometry.

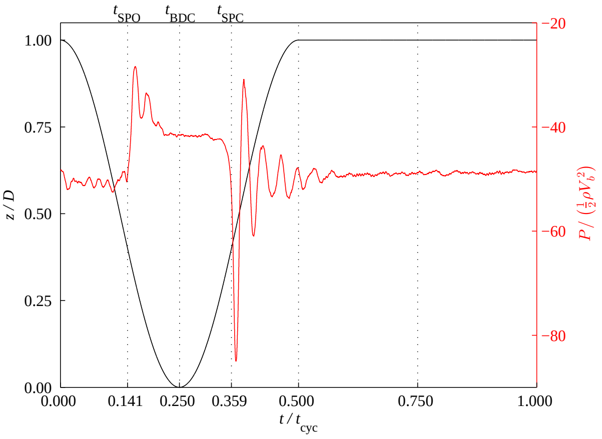

Figure 2. Piston motion (*black curve*) and outlet pressure (*red curve*).

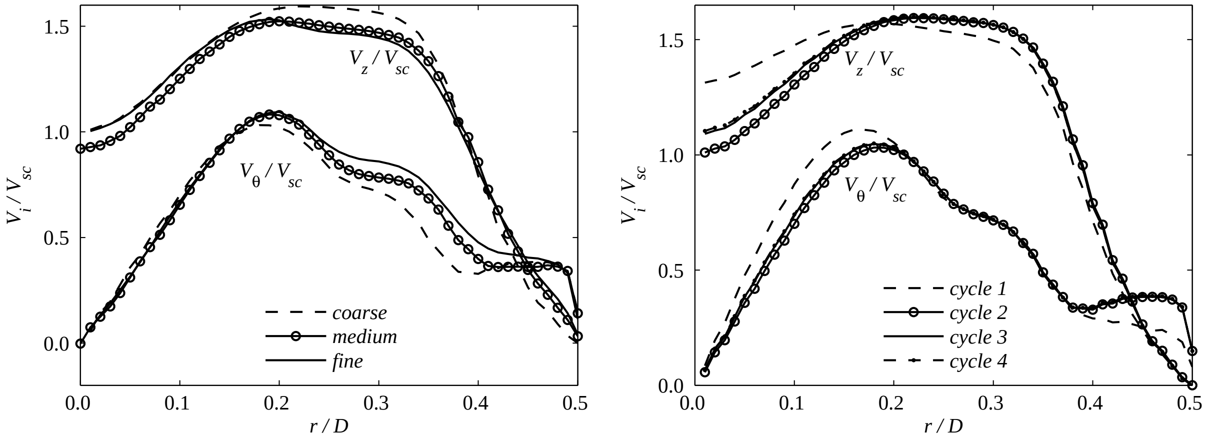

Figure 3. Numerical sensitivity study showing radial velocity profiles. (*left*) Spatial convergence, (*right*) cyclic convergence.

and to investigate the dynamics of the turbulent swirling flow. This is achieved by simulating the flow in the model using CFD and comparing the obtained predictions with the experimental results.

Methodology

The model geometry is shown in Figure 1 and described in detail in [1, 2]. The internal cylinder diameter is $D = 190\,\mathrm{mm}$ and the Reynolds number of the flow is $\mathrm{Re} = V_{sc}D/\nu = 50{,}000$, where ν is the kinematic viscosity and V_{sc} is the characteristic scavenge velocity. In the present work, a port section is used with 30 equally spaced ports and a port angle of $\alpha = 20°$. The piston motion is presented in Figure 2 together with the pressure measured at the model outlet (P_4 in Figure 1). The time is normalized with the total cycle time which is $t_{cyc} = 1.20\,\mathrm{s}$.

The simulations are performed with the commercial CFD code STAR-CCM+ version 8.02.008. A hybrid mesh is used with polyhedral cells in the main volume and prismatic cells on the wall. The mesh used for the simulations, referred to as the '*fine*' mesh, has approximately 5.2 million cells. A second order linear upwind scheme is used for the spatial derivatives and a second order implicit scheme is used for the temporal derivatives. Both Reynolds-Averaged Navier-Stokes turbulence models and large eddy simulations are investigated.

A sensitivity study is carried out in order to investigate the effect of the numerical parameters. Figure 3 shows the radial velocity profiles of tangential velocity V_θ and axial velocity V_z for

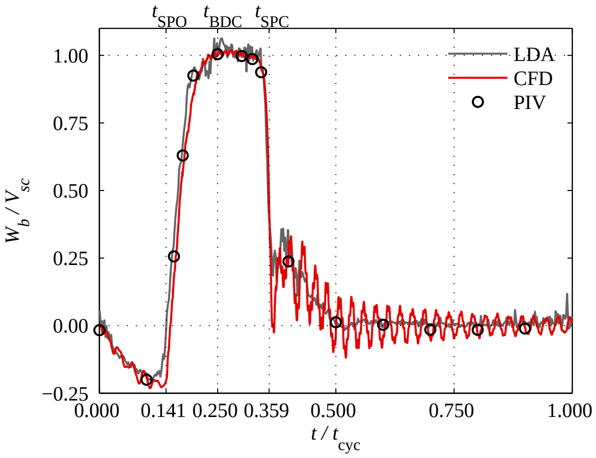

Figure 4. Comparison of predicted and experimental bulk velocity.

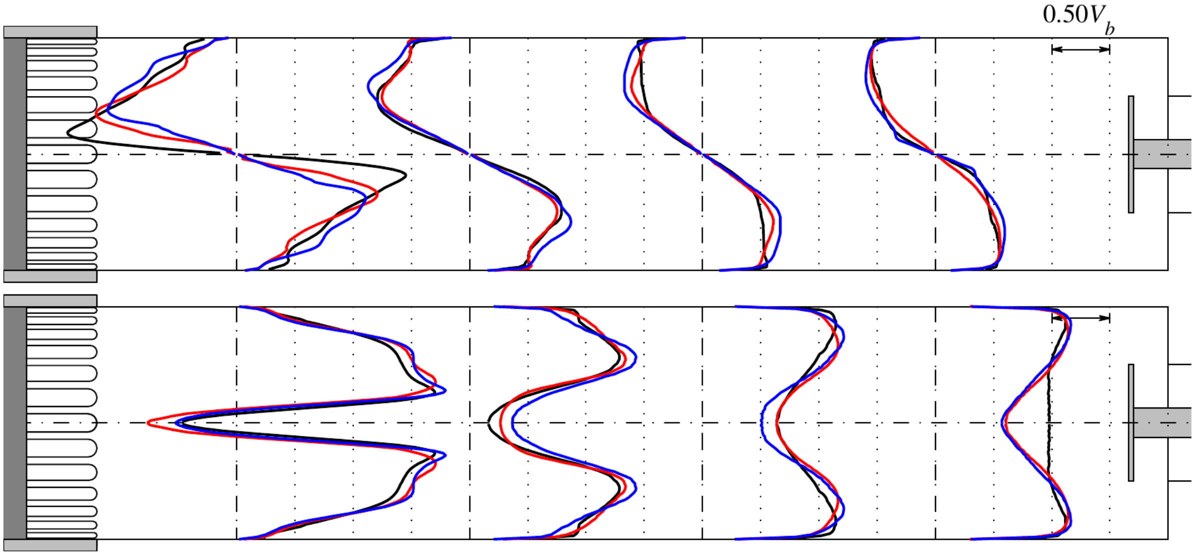

Figure 5. Comparison of radial velocity profiles. (*top*) tangential velocity V_θ, (*bottom*) axial velocity V_z. (*blue curves*) Spalart-Allmaras, (*red curves*) K-ω SST, (*black curves*) experiments.

different mesh sizes and cycles after simulation start. It is seen that the results obtained on the '*fine*' mesh can be considered mesh independent as no significant changes occur from the '*medium*' to '*fine*' mesh. Furthermore, it is seen that the flow becomes approximately periodic already after the second cycle.

Results

A comparison between the predicted and experiential bulk velocity W_b is presented in Figure 4. The numerical results are obtained with the Spalart-Allmaras turbulence model [4] and the experimental results are measured with particle image velocimetry (PIV) and laser Doppler anemometry (LDA). After the scavenge port closing, the CFD predicts large amplitude oscillations which do not exist in the experimental data. This is, however, a result of the experimental phase-averaging and therefore not a 'true' discrepancy. In general, it is concluded that the predicted bulk velocity is in good agreement with the experiments.

Figure 5 shows a comparison of the radial velocity profiles at $t/t_{cyc} = 0.30$ which is shortly after the piston reaches BDC and corresponds to 25% port closure. Profiles are presented

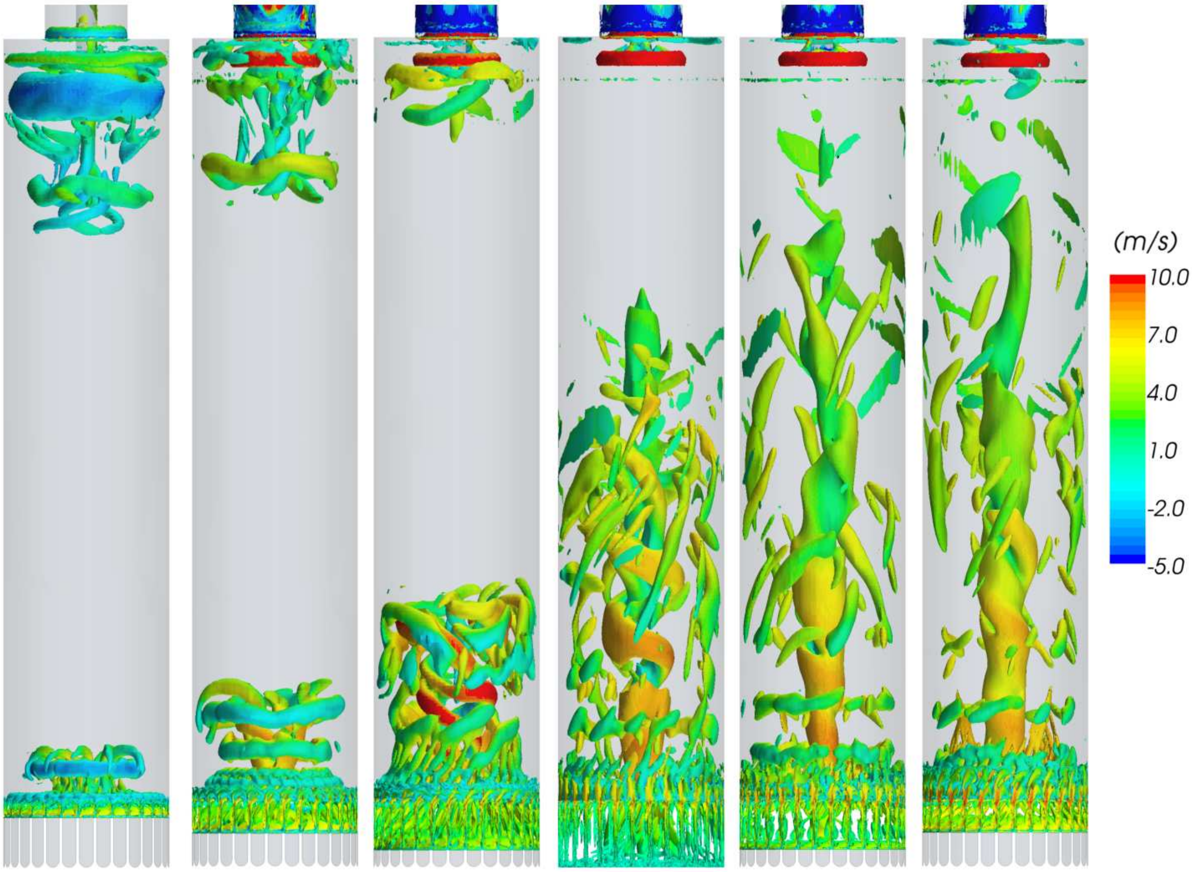

Figure 6. Visualization of flow structures during the scavenging period using iso-surfaces of the λ^2-criterion colored by the axial velocity.

for simulations using both the Spalart-Allmaras and the k-ω SST [3] turbulence model. The predictions obtained with the two turbulence models are in general similar and in reasonable agreement with the experimental data. The tangential velocity is underestimated near the ports and the axial velocity deficit is overpredicted in the top of the cylinder. Both models do, however, correctly predict the reversed axial flow at the centerline near the ports, which shows the occurrence of a vortex breakdown.

The numerical results are also used to investigate the complex flow dynamics of the turbulent swirling flow. In Figure 6 the unsteady flow structures are visualized using iso-surfaces of the λ^2-criterion colored by the axial velocity. In the start of the scavenging process, multiple vortex rings are formed, and later in the process, a coherent vortex core is established. When the ports are fully open, the vortex core has a pronounced helical shape.

References

[1] K. M. Ingvorsen, K. E. Meyer, J. H. Walther, and S. Mayer. Turbulent swirling flow in a dynamic model of a uniflow-scavenged two-stroke engine. *Exp. Fluids*, page submitted, 2013.

[2] K. M. Ingvorsen, K. E. Meyer, J. H. Walther, and S. Mayer. Turbulent swirling flow in a model of a uniflow scavenged two-stroke engine. *Exp. Fluids*, 54:1494, 2013.

[3] Florian R. Menter. Zonal two equation k-ω turbulence models for aerodynamic flows. *AIAA J.*, 1993. Paper 93-2906.

[4] P. R. Spalart and S. R. Allmaras. A one-equation turbulence model for aerodynamic flows. *Rech. Aerospatiale*, 1:5–21, 1994.