Applied Energy 123 (2014) 37–46

Contents lists available at ScienceDirect

Applied Energy

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

# Numerical analysis of the scavenge flow and convective heat transfer in large two-stroke marine diesel engines

E. Sigurdsson [a], K.M. Ingvorsen [a], M.V. Jensen [a], S. Mayer [b], S. Matlok [b], J.H. Walther [a,c,*]

[a] Department of Mechanical Engineering, Technical University of Denmark, Nils Koppels Allé Building 404, 2800 Kgs. Lyngby, Denmark
[b] MAN Diesel & Turbo SE, Teglholmsgade 41, 2450 Kbh. SV, Denmark
[c] Chair of Computational Science, ETH Zürich, Clausiusstrasse 33 ETH-Zentrum, CLT F 11, CH-8092 Zürich, Switzerland

## HIGHLIGHTS

- An accurate CFD model of the scavenge flow in a marine diesel engine is developed.
- Conjugate heat transfer calculations are performed for the piston crown.
- The combustion is accounted for by a novel heat release approach.
- A detailed analysis of the scavenging flow is presented.
- The temporal and spatial temperature distribution in the piston crown is presented.

## ARTICLE INFO

*Article history:*
Received 20 August 2013
Received in revised form 6 December 2013
Accepted 7 February 2014
Available online 12 March 2014

*Keywords:*
Uniflow scavenging
Two-stroke
Heat-transfer
Swirl
CFD
Marine diesel engines

## ABSTRACT

A novel computational fluid dynamics (CFD) model is presented for the study of the scavenging process and convective heat transfer in a large two-stroke low-speed uniflow-scavenged marine diesel engine. The engine is modeled using a fully resolved 12° sector, corresponding to one scavenge port, with cyclic boundaries in the tangential direction. The CFD model is strongly coupled to experiments and effectively provides a high order “interpolation” of the engine processes through the solution of the Reynolds-Averaged Navier–Stokes (RANS) equations subject to boundary conditions obtained through experiments. The imposed experimental data includes time histories of the pressure difference across the engine and the heat release during combustion. The model is validated by a numerical sensitivity analysis and through a comparison of model predictions and experimental data, which shows a good agreement. The results show an effective scavenging and a low convective heat loss in agreement with experimental data for large marine diesel engines.



## 1. Introduction

The International Maritime Organization has defined new regulations for marine engines which aim at reducing $NO_x$ emissions by 75% in many coastal areas from 2016. At the same time, a continued increase in the fuel oil price is expected. These factors call for increased control and improvement of emission levels and engine efficiency.

A key engine process for two-stroke engines, both in terms of emission levels and fuel efficiency, is the scavenging process. The uniflow method is the most efficient scavenging method for two-stroke engines [1] and has also been considered for free piston engines [2] and opposed piston engines [3]. In large two-stroke low-speed marine diesel engines the uniflow method is used in a configuration with angled scavenge ports located in the cylinder liner and a single centered exhaust valve in the cylinder head. As the piston uncovers the ports the fresh charge is blown into the cylinder creating a swirling flow that forces the combustion gases out through the exhaust valve while cooling the combustion chamber surfaces. The swirling motion improves the scavenging and enhances the mixing of air and fuel during the combustion phase.

*Abbreviations:* BDC, bottom dead center; CFD, computational fluid dynamics; EVC, exhaust valve closing; EVO, exhaust valve opening; LUD, linear upwind difference; MARS, monotone advection and reconstruction scheme; RANS, Reynolds-Averaged Navier–Stokes; SPC, scavenge port closing; SPO, scavenge port opening; TDC, top dead center; UD, upwind difference.

* Corresponding author at: Department of Mechanical Engineering, Technical University of Denmark, Nils Koppels Allé Building 404, 2800 Kgs. Lyngby, Denmark. Tel.: +45 4525 4327; fax: +45 4588 4325.

*E-mail address:* jhw@mek.dtu.dk (J.H. Walther).

http://dx.doi.org/10.1016/j.apenergy.2014.02.036
0306-2619/

Optimization of the scavenge flow can therefore lead to reduced emissions and fuel consumption as well as reductions in heat loss and thermal loads.

Despite the importance of the scavenging process, the details of the in-cylinder flow processes are poorly understood. Due to the size of two-stroke low-speed marine diesel engines, experimental investigations have in general been limited to simplified scale models, see e.g. [4,5] and references therein. As experimental work often has significant practical limitations as well as being both time-consuming and expensive, it is desirable to develop numerical tools capable of predicting the in-cylinder processes.

Numerical modeling of the in-cylinder flow in uniflow-scavenged engines has been the subject of earlier works and include steady-flow simulations [6–9], 2D-axisymmetric simulations [10–15], sector simulations without scavenge port geometry [16,17], and full 3D simulations [18–20]. The steady-flow simulations must be considered to provide only a coarse approximation of the flow in a running engine, and have primarily been used for validation by comparison with experimental velocity profiles. The 2D-axisymmetric simulations are the least computational expensive, but the poor representation of the scavenge port geometry is known to lead to large errors in the predictions [16]. The sector approach provides a reasonable compromise between the accuracy of the geometrical representation and the computational cost by assuming rotational symmetry. The full 3D-simulations are the only approach that allows asymmetric phenomenons to be studied. However, the large computational cost of the full 3D simulations generally necessitates reductions of the spatial resolution, which can lead to considerable reductions of the accuracy. In the works where the scavenging process is studied for direct injection engines under fired conditions, only a part of the engine cycle is simulated, see e.g. [11–13,16,17,21]. In these cases, the simulations are started at exhaust valve opening (EVO) or scavenge port opening (SPO), and are sensitive to the initial fields, which must be estimated. These fields include the in-cylinder distribution of the temperature, velocity, and turbulence which in general are unknown.

In the present work a computational fluid dynamics (CFD) model of a large two-stroke low-speed uniflow-scavenged marine diesel engine is developed. The simulated engine is the MAN Diesel & Turbo 4T50ME-X research engine, which is a four cylinder two-stroke marine diesel engine with electronically controlled exhaust valve and fuel injection and has recently been modified to give optical access into the combustion chamber [22]. The cylinder bore is $B = 0.50$ m, the stroke is $S = 2.2$ m, and the engine speed is $N = 123$ rpm at full load conditions. The cylinder has 30 equally spaced scavenge ports with port angles of $\phi_{sc} = 20^\circ$ with respect to the radial direction. The main specifications for the research engine at full load conditions are presented in Table 1. The characteristic scavenge flow velocity given in the table is defined by $U_{sc} = m_{sc}/(\rho_{sc} t_{sc} A_{cyl})$, where $\rho_{sc}$ is the fluid density at scavenge box conditions and $A_{cyl} = B^2\pi/4$ is the cross-sectional area of the cylinder. The Reynolds number for the scavenging process is $\mathrm{Re} = U_{sc}B/\nu_{sc} = 2.5 \times 10^6$, where $\nu_{sc}$ is the kinematic viscosity at scavenge box conditions. The large Reynolds number indicates that the flow is fully turbulent. The flow resistance over the engine is given by the non-dimensional Euler Number $\mathrm{Eu} = (\overline{P_{sc}} - \overline{P_{exh}})\big/\left(\rho_{sc}U_{sc}^2\right) = 14.5$, where $\overline{P_{sc}}$ is the mean scavenge box pressure and $\overline{P_{exh}}$ is the mean exhaust receiver pressure. The blowdown process starts at exhaust valve opening (EVO) which occurs at a crank angle of 68° before the piston reaches bottom dead center (BDC). The scavenge port opening (SPO) and scavenge port closing (SPC) is 37° before and after the BDC, respectively. The scavenging process ends at exhaust valve closing (EVC) 89° after BDC.

For the simulations only one engine cylinder is considered and the geometry is simplified by assuming that the scavenge box and exhaust duct are rotational symmetric. These assumptions make it possible to represent the engine geometry using a 12° sector, corresponding to one scavenge port, with cyclic boundaries in the tangential direction. The combustion is accounted for by implementing a time dependent heat source based on the experimentally determined heat release. Compared to earlier works, this new approach allows for continuous simulation of the full engine cycle and makes the model results independent of the initial fields. The in-cylinder pressure, velocity, temperature, and concentration fields are predicted through the solution of the Reynolds-Averaged Navier–Stokes (RANS) equations. The purpose of the present work is to use the CFD model to gain knowledge on primarily the scavenging process but also the convective contribution to the in-cylinder heat transfer.

The model is validated by a sensitivity analysis of the numerical parameters and through comparison of model predictions and experimental data. The mixing of the cylinder gases is examined and the concentration of fresh charge in the cylinder at exhaust valve closing is estimated. The transient behavior of the angular and axial momentum in the engine cylinder is studied over the full engine cycle. Convective heat transfer between cylinder gases and combustion chamber surfaces is investigated and the total in-cylinder heat loss is estimated. Conjugate heat transfer computations are performed for the upper part of the piston crown and surface temperature variations through the engine cycle are studied.

**Table 1**
Main specifications for the 4T50ME-X research engine at full load conditions.

| Parameter | Sym. | Unit | – |
|---|---|---|---|
| Bore | $B$ | m | 0.50 |
| Stroke | $S$ | m | 2.20 |
| Connecting rod | $L$ | m | 2.885 |
| Compression volume[a] | $V_c$ | $m^3$ | 0.02653 |
| Geometric compression ratio | $CR$ | – | 17.28:1 |
| Engine speed | $N$ | rpm | 123 |
| Number of scavenge ports | $n_{port}$ | – | 30 |
| Scavenge port angle | $\phi_{sc}$ | ° | 20.0 |
| Scavenged mass pr. cycle | $m_{sc}$ | kg | 2.07 |
| Temperature in scavenge box | $T_{sc}$ | K | 312 |
| Mean scavenge box pressure | $\overline{P_{sc}}$ | bar | 3.68 |
| Mean exhaust receiver pressure | $\overline{P_{exh}}$ | bar | 3.28 |
| Open port period | $t_{sc}$ | s | 0.10 |
| Characteristic flow velocity | $U_{sc}$ | m/s | 26.3 |

[a] Uncertain estimate due to compression of connecting rod and thermal expansion of cylinder cover.

## 2. Numerical model

The engine simulations are performed using the commercial CFD code STAR-CD version 4.12.010 solving the RANS equations using a finite volume discretization. The computational domain is a 12° sector corresponding to a single scavenge port and represents the engine geometry from the scavenge box inlet to the exhaust diffuser outlet (Fig. 1). For simplicity, the piston, cylinder head, exhaust valve, and scavenge port geometries are modeled using plane surfaces with sharp edges. However, care is taken, to ensure that the compression volume and the effective flow areas in the model are the same as in the physical engine. The mesh consists of approximately 174,000 cells corresponding 5.2 million cells for the full 360° geometry. The mesh resolution in the cylinder volume is 76 cells in radial direction, 8 cells in tangential direction, and 172 cells in axial direction. The mesh is gradually refined toward the walls in radial and axial directions ensuring $y^+$ values throughout the engine cycles within the limits of the applied turbulence models ($y^+ < 250$). Additional details and visualizations of the mesh are given in [23]. The motion of the piston and exhaust valve is mod-