Applied Mathematical Modelling 40 (2016) 6293–6309

Contents lists available at ScienceDirect

Applied Mathematical Modelling

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

Validation and optimization of the thermal cycle for a diesel engine by computational fluid dynamics modeling

Wojciech Tutak, Arkadiusz Jamrozik*

Czestochowa University of Technology, 42-201 Czestochowa, Poland

ARTICLE INFO

Article history:
Received 30 August 2013
Revised 3 February 2016
Accepted 18 February 2016
Available online 3 March 2016

Keywords:
Combustion
Diesel engine
Mesh
Modeling

ABSTRACT

In this study, we developed a computational fluid dynamics (CFD) model of a turbocharged diesel engine (1CT107) powered by diesel fuel. The engine was part of the power generating set in the portable version. In the simulation tests, we analyzed the impact of the ignition timing on the thermodynamic parameters and emissions of toxic components. We verified the model of the test engine and it was then used to optimize the thermal cycle for the test engine. We found that the engine model had acceptable accuracy and it was suitable for emissions modeling. Under the full load, the NO emissions were 2.2 g/kWh, which satisfied the EURO IV criteria. As the load increased, the soot emissions also increased. This model also confirmed that the dynamics of soot formation were the opposite of NO formation. In summary, CFD modeling provides a powerful tool for optimizing the internal combustion engine in terms of both the thermodynamic parameters and emissions. The model of the test engine was produced using AVL FIRE.

1. Introduction

Compression ignition engines are used widely because of their high thermal efficiency and low CO_2 emissions. Exhaust emissions are the main negative feature of diesel engines: nitrogen oxides (NO_x) and soot or particle matter. Studies are being conducted to reduce the toxicity of engine exhaust emissions by focusing on the application of high pressure fuel injection systems, turbocharging, exhaust gas recirculation, or new alternative fuels [1–5].

Studies based on numerical simulations using advanced mathematical models have been reported widely recently. Engine simulation models are valuable tools for researchers and engineers, where the aim is to design engines that comply with strict emissions legislation while maintaining high performance. In recent years, many simulation models have been developed for internal combustion engines, which can be divided into two main categories: phenomenological and computational fluid dynamics (CFD) [6]. The main difficulties when using CFD models as an everyday tool for engine simulations are the very high computational time requirements for engine cycle simulations and the high demands of users. Much progress has been made in both of these areas during the last decade, but CFD models fundamentally have very high demands in terms of CPU power [7]. The development of numerical modeling has been facilitated by increases in computational power, which allows the modeling of flow processes as well as combustion in three dimensions [8–10]. Thus, in the present study, we attempted to optimize this process. The main aim was to achieve good agreement between the modeling results and those obtained in experiments, while also minimizing the computational time required. Many previous studies have addressed the

* Corresponding author. Tel.: +48 343250541; fax: +48 343250555.
E-mail address: jamrozik@imc.pcz.czest.pl, arkadiuszjamrozik@yahoo.pl, tutak@imc.pcz.czest.pl (W. Tutak).

http://dx.doi.org/10.1016/j.apm.2016.02.021

Abbreviations

CA	crank angle, degrees
BDC	bottom dead center, degrees
TDC	top dead center
IMEP	indicated mean effective pressure, MPa
ITE	indicated thermal efficiency, %
HRR	heat release rate, J/degree
U	velocity
x	coordinate
t	time
g_i	specific body force
k	turbulent kinetic energy
mf	mass fraction
f	elliptic relaxation function
P_k	production of turbulent kinetic energy
T	turbulent time scale
L	turbulent length scale
V	cylinder volume
p	cylinder pressure

Greek

φ	the corresponding property
$\dot{\gamma}_m$	internal source per unit mass of heat due to chemical reaction
$\dot{\gamma}_A$	diffusion flux of heat through the control surface
ρ	fluid density
δ_{ij}	unit tensor
μ	viscosity
$\dot{q}_g$	heat flux
Φ	instantaneous component
$\tilde{\varphi}$	fluctuation component
C_μ	coefficient used as a constant value
ε	turbulent kinetic energy dissipation rate
ζ	normalized velocity scale
γ	specific heat ratio

modeling of a piston engine, but most focused on the cylinder without the intake and exhaust ports, and valves. The initial movement of the charge is imposed artificially [6,11–14]. In order to determine the initial conditions that determine the initial charge motion and the initial level of turbulence intensity, it is necessary to produce a basic physical model of the intake stroke [10,15].

Many studies have used KIVA [13,16], so we also employed this program to model the engine thermal cycle. It should be noted that although this method uses structural mesh cells, they are characterized as relatively large mesh cells due to the specificity of the program, rather than the pre-processor. This method is used primarily to model only the cylinder while the valves are closed. In order to obtain results that are very similar to the experimental values, a sufficiently dense model is required in mesh computing.

Many researchers have assumed an arbitrary mesh size without optimizing the size, but the mesh density affects the calculation performance. However, some studies have addressed this process, e.g., Shi et al. [17] obtained results using three meshes with different mesh densities, which affected the results, where the time required by the CPU-based solvers increased in a linear manner with the number of mesh points (ranging from 1700 to 14,960 mesh points at the engine bottom dead center (BDC)). They used a Tesla C2050 GPU and the calculation was performed on approximately 13 parallel 2.8 GHz CPUs. Viggiano and Magi [18] investigated five meshes with different densities to model a HCCI engine, where simulations were performed using various mesh resolutions in order to assess the mesh independence of the results. The computations for the baseline were conducted with five mesh resolutions and the results were obtained in terms of the pressure trace and heat release rate (HRR). The results indicated that there was a very slight increase in the ignition delay when a small number of cells were used for the top dead center (TDC), as well as higher values for the pressure during combustion when using lower resolution near the walls. These results showed that suitable values for the pressure trace and HRR could be obtained using a relatively coarse number of mesh points. However, this was not the cases for emissions, where a higher mesh resolution was required [18]. This study did not consider the intake and exhaust valves. Kim and Park [19] presented the results obtained by compression ignition engine model prepared using KIVA code, where they employed two grids with