Journal of Computational and Applied Mathematics 291 (2016) 94–102

Contents lists available at ScienceDirect

Journal of Computational and Applied Mathematics

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

Numerical simulation of primary atomization in diesel spray at low injection pressure

F.J. Salvador [a,*], J.-V. Romero [b], M.-D. Roselló [b], D. Jaramillo [a]

[a] CMT-Motores Térmicos, Universitat Politècnica de València, Camino de Vera s/n, Edificio 6D, 46022, Valencia, Spain

[b] Instituto Universitario de Matemática Multidisciplinar, Universitat Politècnica de València, Camino de Vera s/n, Edificio 8G, 2º, 46022, Valencia, Spain

ARTICLE INFO

Article history:
Received 15 October 2014
Received in revised form 23 March 2015

Keywords:
Diesel spray
Atomization
Gerris
DNS
VOF

ABSTRACT

Atomization involves complex physical processes and gas–liquid interaction. Primary atomization on diesel spray is not well understood due to the difficulties to perform experimental measurements in the near nozzle field. Hence computational fluid dynamics (CFD) has been used as a key element to understand and improve diesel spray.

A recent new code for incompressible multiphase flow with adaptive octree mesh refinement has been used to perform simulations of atomization at low injection pressure conditions. The multiphase flow strategy to manage different flows is the volume of fluid (VOF) method. The adaptive mesh allows to locally refine the mesh at each time step where a better resolution is needed to capture important gradients instead of using a static mesh with a fixed and high number of cells which, in turn, would lead to an unaffordable computational cost. Even with this approach, the cell number is very high to achieve a Direct Numerical Simulation (DNS) at reasonable computational cost. To reduce the computational cost, an idea has been explored, the possibility of setting a maximum number of cells of the domain. Following this idea, the code has been tested with different configurations to understand their effects on numerical stability, the change in different spray parameters and the benefits achieved in terms of execution time. The outcomes have been validated against a theoretical model.

1. Introduction

Atomization process in a spray has been an important issue for researchers during last decade, due to its presence in many industrial applications. In particular, this is extremely important in Diesel Engines, where combustion efficiency and pollutant formation are consequences of spray atomization and fuel–air mixing process [1–5].

As a result of these studies, several tools have been developed for modeling macroscopic spray behavior [6,7]. Nevertheless, there are still uncertainties related with internal nozzle flow and its link with spray formation and primary breakup [8–10].

Last decades have been characterized by a continuous increase in computational resources. For the study of diesel spray this increase allows to move forward to use more complex models for breakup, evaporation, coalescence, turbulence, etc.

* Corresponding author. Tel.: +34 963879659; fax: +34 963877659.
E-mail addresses: fsalvado@mot.upv.es (F.J. Salvador), jvromero@imm.upv.es (J.-V. Romero), drosello@imm.upv.es (M.-D. Roselló), dajacis@mot.upv.es (D. Jaramillo).

http://dx.doi.org/10.1016/j.cam.2015.03.044
0377-0427/

Nomenclature

c:	concentration
$\mathbf{D}$:	deformation tensor
D_0:	orifice diameter
f:	frequency
k:	curvature
L:	domain width
M:	momentum flux
$\mathbf{n}$:	normal vector
p:	pressure
Sc:	Schmidt number
$\mathbf{u}$:	velocity

Greek symbols:

α	shape factor
δ_s	Dirac distribution
θ_u	velocity spray angle
μ	dynamic viscosity
ρ	density
σ	surface tension coefficient

Subscripts:

f:	fuel
a:	air

In terms of turbulence modeling, the classes of models from lower to higher computational cost are: RANS (Reynolds averaged Navier–Stokes) [11,12], LES (large eddy simulations) [13,14] and DNS (direct numerical simulations) [15–17]. While RANS methods have been used along several decades, the use of LES models is more recent and even now the computational requirements for the use of DNS is still very high for study typical current conditions in diesel engines.

However, despite all the computational difficulties some researchers [16–18] have tried to use DNS approach for the study of Diesel sprays. Some basic procedures have been adopted by these researchers in order to be able to perform DNS simulations in sprays, such as, decreasing injection velocity and reducing the domain for studying only the first millimeters, and so, taking into account only primary atomization. It is also used an Adaptative Mesh Refinement (AMR) method to reduce the computational cos of simulations [17,18]. Even with this simplifications, in the present paper, the numerical cost to simulate around 8 mm of the spray has been around 2 months running over 32 CPUs on a blade server Fujitsu BX920.

The aim of this paper is to study the potential of a new code [19,20] to perform simulations of primary atomization in diesel sprays with DNS approach. For this purpose the same strategy used by other researchers described before [16–18] for reducing the computational time has been used: Low spray velocity, small domains (just to consider only the first atomization and breakup length) and the application of AMR algorithm.

The present paper has been split into 6 sections. In Section 2, a brief description of the numerical code will be performed. After that, in Section 3, a mesh sensitivity study performed over several parameters that define the mesh will be reported. In Section 4 the outcomes of the computational simulations will be validated against a theoretical model available in the literature (which in turns has been widely validated). After the validation, in Section 5, a study on the influence that the use of periodic perturbation in the injection velocity has on the results will be described. Finally, in Section 6 the main conclusions will be drawn.

2. Numerical code

For this study, the numerical code Gerris developed by Stéphane Popinet [19,20] has been used. This code solves Navier–Stokes equations with surface tension for incompressible flow (1)–(3)

$$\rho\left(\partial_t \mathbf{u} + \mathbf{u}\cdot\nabla\mathbf{u}\right) = -\nabla p + \nabla\cdot\left(2\mu\mathbf{D}\right) + \sigma k\delta_s\mathbf{n}, \tag{1}$$

$$\partial_t\rho + \nabla\cdot\left(\rho\mathbf{u}\right) = \mathbf{0}, \tag{2}$$

$$\nabla\cdot\mathbf{u} = \mathbf{0}, \tag{3}$$

where $\rho = \rho(\mathbf{x}, t)$ is the fluid density, $\mathbf{u} = \left(u_x, u_y, u_z\right)$ is the fluid velocity, p is the pressure field, $\mu = \mu(\mathbf{x}, t)$ is the dynamic viscosity, $\mathbf{D}$ is the deformation tensor, σ is the surface tension coefficient, k and $\mathbf{n}$ are the curvature and the normal vector