Chemical Engineering Science 131 (2015) 41–48

Contents lists available at ScienceDirect

Chemical Engineering Science

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

Large eddy simulation of a high-pressure homogenizer valve

Javad Taghinia*, Md Mizanur Rahman, Timo Siikonen

Aalto University, School of Engineering, FI-00076, Finland

HIGHLIGHTS

- LES is applied for the first time to simulate the flow in HPH valve.
- Two zero-equation SGS models (RAST and DSM) are used in this study.
- Both models produced relatively accurate results compared to experiment.
- RAST model showed slightly better performance than DSM.

ARTICLE INFO

Article history:
Received 18 October 2014
Received in revised form
11 March 2015
Accepted 27 March 2015
Available online 2 April 2015

Keywords:
High-Pressure-Homogenizer valve
Large eddy simulation
Sub-grid-scale model
Turbulent flow

ABSTRACT

A detailed understanding of the flow behavior inside a high pressure homogenizer (HPH) valve has a vital importance in designing and optimizing the systems in terms of energy and performance. A large Eddy simulation (LES) method is used in this study to investigate the flow structure in an HPH valve. The current paper utilizes two zero-equation subgrid-scale models: namely the RAST (Rahman–Agarwal–Siikonen–Taghinia) and DSM (dynamic Smagorinsky model). The performance of these two models and their predictions are compared with experimental data available in the literature. Computations dictate that an LES can reproduce the accurate information in terms of main parameters that are necessary in designing and optimizing of the homogenizing process. Comparisons demonstrate that both models are capable of predicting the turbulent-flow structures at the gap exit which are in a good agreement with measurements. However, the RAST model shows a slight superiority over the performance of DSM.

1. Introduction

The homogenization is a mechanism responsible for the subdivision of particles into very small sizes to create an emulsion for further processing. This procedure is widely used in food and dairy industries in which the homogenization plays a vital role in improving the product quality and taste. This process occurs in a homogenization valve which is the main component of an high pressure homogenizer (HPH) device. The flow goes through a very narrow gap creating high acceleration and turbulent structures that are essential in creating the dispersion of particles. Therefore, a clear understanding of the flow inside the valve domain provides a critical information in designing and optimization of homogenization devices in terms of energy efficiency.

One of the main approaches in numerical investigation of the fluid flow is the computational fluid dynamics (CFD). This approach provides a detailed insight into the turbulent-flow structure and main flow parameters such as the velocity, pressure and kinetic energy distributions. The early studies are carried out by Kleinig and Middelberg (1997) and Stevenson and Chen (1997). They investigated the flow field inside the homogenizer valve with a k–ϵ Reynolds-Averaged-Navier–Stokes (RANS) turbulence model and found fairly good results. There are a few studies using CFD for an HPH valve in the literature. The majority of these studies deals with a two-dimensional flow in HPH valves with different variants of k–ϵ of an RANS approach (Miller et al., 2002; Kelly and Muske, 2004; Floury et al., 2004; Köhler et al., 2007; Steiner et al., 2006; Raikar et al., 2009; Casoli et al., 2010). Håkansson et al. (2012) performed numerical studies based on standard RNG and realizable k–ϵ models for a three-dimensional HPH valve. They concluded that the standard k–ϵ model is unable to accurately predict the flow field close to the gap. They also reported that all applied turbulence models failed to reproduce the correct kinetic energy distribution at the gap entrance.

The above-mentioned studies consider the existing information on the flow behavior in HPH valves based on RANS approaches which provide a general description of the flow inside the valve due to their averaging nature. The applied "steady-state" RANS cannot provide a detailed understanding of instantaneous velocity and/or fluctuations due to the time averaging procedure.

With the developing computational tools and power, the LES is an ideal and powerful approach in modeling of transient flows. However, an LES requires higher computational costs and grid

* Corresponding author.
E-mail address: javad.taghiniaseyedjalali@aalto.fi (J. Taghinia).

http://dx.doi.org/10.1016/j.ces.2015.03.041

resolution than that of RANS; however, its ability to capture the flow structures at a wide range of turbulent scales makes it a promising alternative approach compared to RANS in simulating complex flows. The application of LES has attracted an increasing interest in industrial design and modelings. The studies based on an LES in analyzing the flow in valves are inadequate and there is rarely reported research concerning the LES approach in the simulation of flow field in HPH valves.

The LES decomposes the flow structures to a large scale and a sub-grid scale (SGS). The larger eddies are solved directly while the SGS part is modeled. The criterion for this scale-decomposition is usually a grid-based scheme, serving as a filter. The main differences among LES approaches are in the applied SGS models, determining the mechanism to reproduce small scales based on various methods. The most common SGS models are Smagorisnky model (SM) (Smagorinsky, 1963) and dynamic Smagorinsky model (DSM) devised by Germano et al. (1991). The main difference between these two models lies within the procedure in determining the eddy-viscosity coefficient. The SM utilizes a constant eddy-viscosity coefficient which is not suitable for complex flows (Olsson and Fuchs, 1996). To overcome this limitation, Germano et al. (1991) implemented a dynamic method in which the model coefficient is determined dynamically via the scale-similarity definition and the local-equilibrium hypothesis. The model coefficient thus obtained is a local value, varying in time and space over a fairly wide range with both negative and positive values, and vanishes near the solid boundary with the correct near-wall behavior (Piomelli, 1993). However, a negative coefficient and consequently a negative eddy-viscosity causes numerical instability, eventually leading to an excessive level of numerical noise or even divergence of the numerical solution. To avoid this occurrence, the model coefficient is simply clipped at zero. This method is somewhat different from the usual practice in which the total viscosity (laminar viscosity + eddy-viscosity) is equated to zero whenever negative. Recently, Taghinia (2014) developed an SGS model called RAST (Rahman–Agarwal–Siikonen–Taghinia) model with a variable eddy-viscosity coefficient that depends nonlinearly on both the rotational and irrotational strains, responding to flow separation and reattachment. Unlike the DSM, the RAST model utilizes a single filter, making it more robust in computations. This aspect makes this model capable of simulating complex flows with a sudden pressure-drop, high strain-rate and streamline curvature effect.

This paper aims at investigating the flow in the HPH valve using the LES with DSM and RAST model for the first time since there is no published study on this subject based on these approaches. The current study provides a suitable benchmark to assess the ability of these models in reproducing a real behavior of the fluid flow especially close to the gap area. The predictions are compared with the experimental data available in the literature (Håkansson et al., 2010; Innings and Trägårdh, 2007). The following section briefly explains the under-laying governing equations for LES and SGS models.

2. Mathematical formulation

2.1. Large eddy simulation (LES)

In an LES the largest eddies that contain the major fraction of energy are computed whereas the small eddies are modeled. This process is performed by applying a filter function $G(x;x')$ to a decomposed function f

$$f = \overline{f} + f_{sgs}, \quad \overline{f} = \int_{R^3} G(x;x') f(x')\, dx' \tag{1}$$

where the function f is decomposed to resolved and sub-grid scale values. The implied filter function $G(x;x')$ herein, operated on a filter width $\overline{\Delta}$ is a top-hat filter given by

$$G_i(x_i;x_i') = \begin{cases} \dfrac{1}{\overline{\Delta}^3}, & \text{if } |x_i - x_i'| \le \dfrac{\overline{\Delta}}{2}, \\ 0, & \text{otherwise.} \end{cases} \tag{2}$$

Applying the spatial filter to incompressible Navier–Stokes equations and using the commutation characteristics, the LES equations yield

$$\frac{\partial \overline{u_j}}{\partial x_i} = 0 \tag{3}$$

$$\frac{\partial \overline{u_i}}{\partial t} + \frac{\partial \overline{u_i}\,\overline{u_j}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial \overline{u}_i}{\partial x_j}\right) - \frac{\partial \tau_{ij}}{\partial x_j} \tag{4}$$

where the *overbar* notation denotes the application of top-hat filter, ρ signifies the fluid density and ν is the kinematic viscosity. The sub-grid scale (SGS) stress tensor is defined as

$$\tau_{ij} = \overline{u_i u_j} - \overline{u}_i \overline{u}_j \tag{5}$$

The sub-grid scale stresses are unknown and need to be modeled.

2.2. RAST sub-grid scale model

The RAST model with a single grid filter is recently developed for the large eddy simulation (Taghinia, 2014). In this sub-grid scale (SGS) model, the unknown SGS turbulent stresses resulting from the filtering operation in Eq. (5) need a closure. Following the Boussinesq approximation, the relationship between the anisotropic part of the SGS stress tensor and the large-scale (i.e., resolved) strain-rate tensor can be expressed as

$$\tau_{ij} - \frac{1}{3}\delta_{ij}\tau_{kk} = -2\nu_T \overline{S}_{ij}, \quad \overline{S}_{ij} = \frac{1}{2}\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right) \tag{6}$$

The isotropic part of stress tensor ($\frac{1}{3}\delta_{ij}\tau_{kk}$) is implicitly added to the pressure. The SGS eddy-viscosity ν_T is a scalar quantity and is determined as

$$\nu_T = C_\mu \overline{\Delta}^2 \overline{S} \tag{7}$$

where C_μ is a model coefficient, $\overline{S} = \sqrt{2\overline{S}_{ij}\overline{S}_{ij}}$ is the invariant of resolved strain-rate tensor, and $\overline{\Delta}$ is the grid-filter length (or width) computed from the cell-volume

$$\overline{\Delta} = (\Delta_1 \Delta_2 \Delta_3)^{1/3} \tag{8}$$

where Δ_1, Δ_2 and Δ_3 are the grid sizes in x, y and z directions, respectively. The eddy-viscosity coefficient C_μ appearing in Eq. (7) is an indisputably flow-dependent quantity which can be readily computed as a scalar function of the invariants formed on the resolved strain-rate $\overline{S}_{ij}$ tensor and the resolved vorticity tensor given by

$$\overline{W}_{ij} = \frac{1}{2}\left(\frac{\partial \overline{u}_i}{\partial x_j} - \frac{\partial \overline{u}_j}{\partial x_i}\right) \tag{9}$$

The invariant of resolved vorticity tensor is defined by $\overline{W} = \sqrt{2\overline{W}_{ij}\overline{W}_{ij}}$.

The SGS turbulent kinetic energy k_{sgs} transport model accounts for the history and non-local effects, having the potential to benefit the modeling of complex flows with non-equilibrium turbulence. The SGS kinetic energy is defined as

$$k_{sgs} = \tfrac{1}{2}\tau_{kk} = \tfrac{1}{2}(\overline{u_k u_k} - \overline{u}_k \overline{u}_k) \tag{10}$$

which can be obtained by contracting the sub-grid scale stress in Eq. (5). However, with the RAST model k_{sgs} is computed algebraically as

$$k_{sgs} = C_\mu^{\frac{2}{3}} \left(\overline{\Delta}\,\overline{S}\right)^2 \tag{11}$$