Nuclear Engineering and Design 240 (2010) 2116–2122

Contents lists available at ScienceDirect

Nuclear Engineering and Design

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

Large-Eddy Simulation study of turbulent mixing in a T-junction

A.K. Kuczaj*, E.M.J. Komen, M.S. Loginov

Computational Fluid Dynamics, Safety & Performance, Nuclear Research and Consultancy Group (NRG), P.O. Box 25, 1755 ZG Petten, The Netherlands

ARTICLE INFO

Article history:
Received 27 January 2009
Received in revised form 2 July 2009
Accepted 8 July 2009

ABSTRACT

A potential cause of thermal fatigue failures in energy cooling systems is identified with cyclic stresses imposed on a piping system. These are generated due to temperature changes in regions where cold and hot flows are intensively mixed together. A typical situation for such mixing appears in turbulent flow through a T-junction, which is investigated here using Large-Eddy Simulations (LES). In general, LES is well capable in capturing the mixing phenomena and accompanied turbulent flow fluctuations in a T-junction. An assessment of the accuracy of LES predictions is made for the applied Vreman subgrid-scale model through a direct comparison with the available experimental results. In particular, an estimation of the minimal mesh-resolution requirements for LES is examined on the basis of the complementary RANS simulations. This estimation is based on the characteristics turbulent scales (e.g., Taylor micro-scale) that can be computed from LES or RANS simulations.

1. Introduction

Development and validation of modelling approaches for turbulent mixing is an important issue implicitly connected with the nuclear reactor safety. Turbulent mixing in reactor cooling systems can potentially lead to appearance of the thermal fatigue phenomena (Chapuliot et al., 2005). Detailed information about the amplitudes and frequencies of the flow temperature fluctuations in the pipelines is highly desired in order to prevent possible damages. Usual strategy applied to thermal fatigue predictions involves Computational Fluid Dynamics simulations to determine the temperature fluctuations, which serve as an input for the structural mechanics analyses. Turbulent mixing of hot and cold fluid streams in a T-junction is a challenging test case for validation of applied numerical methods. At high Reynolds number, accurate flow predictions require considerably large computational effort due to the amount of various flow-scales that need to be numerically resolved.

In the available literature, a number of numerical experiments in T-junctions can be found (see Walker et al., 2009, and references therein). Simulations with five different physical conditions and two configurations of T-junctions were presented by Hu and Kazimi (2006). This benchmark study of high cycle temperature fluctuations showed the applicability of LES in prediction of turbulent flow features in a T-junction. Similar conclusions were drawn by Coste et al. (2006), where an influence of secondary flow on temperature averages and fluctuations is demonstrated. The secondary flow was imposed by an additional elbow attached to one of the T-junction legs. Simulations with the standard and dynamic Smagorinsky models were compared by Merzari and Ninokata (2007). An advantage of the dynamic model over the Smagorinsky method was shown as the dynamic procedure better resolves the small-scale turbulence flow features. The utility of both models application for predictions of the mean flow features was demonstrated there. Optimal operating conditions for a flow in a mixing-T were studied by Hosseini et al. (2008). Various jet rising mechanisms as a result of the T-junction geometry were examined there. An extensive literature survey that covers joint US–Japanese and European research programmes on this subject was prepared by Walker et al. (2009). Although thermal fatigue topic obtained escalating attention in the recent years, detailed validation of applied numerical methods is still needed in order to determine their accuracy and range of application.

In the previous work (Kuczaj and Komen, 2010), we investigated accuracy of the flow predictions coming from two LES models (Vreman, 2004; Smagorinsky, 1963) applied for modelling of turbulent mixing in a T-junction. An engineering estimation of the required computational mesh resolution based on the 'a posteriori' computed Taylor micro-scale length was provided. In this paper, we concentrate on two issues. First we try to answer the question whether 'a priori' estimated turbulence length scales from RANS simulations can be useful for predictions of the required LES mesh resolution. Other estimation approaches for the required grid resolution will be also briefly discussed. Afterwards, flow predictions obtained with the applied LES model on various mesh resolutions will be analyzed.

Comparison with the experimental data (Andersson et al., 2006) provides a conclusive opportunity for assessing suitability of the

* Corresponding author.
E-mail address: arek@kuczaj.pl (A.K. Kuczaj).
URL: http://www.kuczaj.pl (A.K. Kuczaj).

0029-5493/$ – see front matter © 2010 Published by Elsevier B.V.
doi:10.1016/j.nucengdes.2009.11.027

obtained results in prediction of turbulent mixing. We pay special attention to the resolution that needs to be used for accurate flow predictions. We will show that large differences occur in predictions of turbulence quantities (mean and fluctuations) that are obtained on the basis of the same LES model but with under-resolved mesh density.

The organization of this paper is as follows. In Section 2 we introduce the experimental and computational setup. Minimal LES mesh-resolution requirements are examined with complementary RANS simulations in Section 3. Section 4 is devoted to verification of the applicability of applied modelling methods and their numerical validation through a direct comparison of results with the experimental data. Concluding remarks are collected in Section 5.

2. Experimental and computational setup

The experiment conducted by Andersson et al. (2006) was particularly designed to investigate thermal mixing in a T-junction with prescribed flow conditions. It consists of two perpendicularly connected pipes with diameters of $2r_h = 0.1$ m (hot flow: the z-direction, vertical) and $2r_c = 0.14$ m (cold flow: the x-direction, horizontal), which form the considered T-junction presented in Fig. 1. In the experiment, the approximate cold and hot flow rates are 6 and 12 l/s. They give inlet bulk velocity values of 0.76–0.78 m/s and the corresponding Reynolds numbers are $(0.95–0.96) \times 10^5$. The temperature difference between the hot and the cold fluid was set to 15° with $T_h = 303$ K and $T_c = 288$ K. Velocity profiles in the inlet pipes and downstream of the T-junction were measured with the Laser Doppler Velocimetry (LDV). The mixing process has also been studied with the Laser Induced Fluorescence (LIF) in which the mixing of a passive scalar was measured by means of concentration. For further experimental details we refer to Andersson et al. (2006). Additionally, the flow measurements three diameters upstream from the center of the T-junction for both hot and cold legs allow setting the inflow conditions for the numerical simulations that are described next.

In our numerical model, we solve the filtered incompressible Navier–Stokes equations, which have the following form (Fureby et al., 1997; Weller et al., 1998):

$$\frac{\partial \bar{\mathbf{u}}}{\partial t} + \nabla \cdot (\bar{\mathbf{u}} \otimes \bar{\mathbf{u}}) + \frac{1}{\rho}\nabla \bar{p} = -\nabla \cdot (\mathbf{B} - 2\nu \mathbf{D}), \quad (1)$$

where $\bar{\mathbf{u}}$ is the filtered velocity with its components (u, v, w) correspondingly in x-, y-, z-direction, $\bar{p}$ is the filtered pressure, and ρ is the fluid density. The symmetric deformation tensor $\mathbf{D}$ is defined as:

$$\mathbf{D} = \frac{1}{2}(\nabla \bar{\mathbf{u}} + \nabla \bar{\mathbf{u}}^T), \quad (2)$$

and the subgrid-scale (SGS) stress tensor is:

$$\mathbf{B} = \overline{\mathbf{u} \otimes \mathbf{u}} - \bar{\mathbf{u}} \otimes \bar{\mathbf{u}}. \quad (3)$$

The standard Smagorinsky model implies parameterization of the SGS stress tensor:

$$\mathbf{B} = \mathbf{B}_D + \frac{2}{3}k\mathbf{I}, \quad (4)$$

where $k = (1/2)tr(\mathbf{B})$ and B_D is the deviatoric part of $\mathbf{B}$, which is usually modelled using an eddy-viscosity assumption:

$$\mathbf{B}_D = -2\nu_t \mathbf{D}_D, \quad (5)$$

with respect to the deviatoric part of the deformation tensor $\mathbf{D}_D$. This way, we arrive to a situation in which k and ν_t must be specified in order to mathematically close the equations. For our simulation purposes, the recently developed LES subgrid model by Vreman (2004) was implemented. This eddy-viscosity model is based on the computation of the turbulence viscosity in the following form:

$$\nu_t = \frac{5}{2} C_s^2 \Delta^2 \sqrt{\frac{inv_2(\boldsymbol{\alpha}^T \cdot \boldsymbol{\alpha})}{tr(\boldsymbol{\alpha}^T \cdot \boldsymbol{\alpha})}}, \quad (6)$$

where $\boldsymbol{\alpha} = \nabla \bar{\mathbf{u}}$, $K = 2\sqrt{2}\nu_t|\mathbf{D}_D|$ and Δ is the filter-width directly linked with the mesh resolution in our simulations. The second invariant of the velocity gradients product is denoted by $inv_2(\boldsymbol{\alpha}^T \cdot \boldsymbol{\alpha})$. This formulation is consistent with Eqs. (5)–(8) in the original paper (Vreman, 2004).

The Navier–Stokes equations (1) are solved along with a passive scalar equation in the following form:

$$\frac{\partial \bar{T}}{\partial t} + \bar{\mathbf{u}} \cdot \nabla \bar{T} = \left(\frac{\Theta}{\rho C_p} + \frac{\nu_t}{Pr_t} \right) \nabla^2 \bar{T}, \quad (7)$$

where $\bar{T}$ is the filtered scalar, $\Theta = 0.6$ kg m/(s^3 K) is the heat capacity, $C_p = 4.18 \times 10^3$ m^2/(s^2 K) is the specific heat and $Pr_t = 0.85$ is the turbulent Prandtl number. The other parameters used in the simulations are density $\rho = 10^3$ kg/m^3 and laminar viscosity $\nu = 1.005 \times 10^{-6}$ m^2/s. The system of Eqs. (1)–(7) along with the incompressibility constraint is solved with a second-order discretization scheme both in space and time. The code is based on standard numerical libraries (Weller et al., 1998), which were developed for Large-Eddy Simulations with an additional passive scalar equation and a new SGS model. In detail, a second-order backward-differencing based on the current and two previous time-step values is used. Pressure is eliminated by solving the Poisson equations with mixed SIMPLE/PISO velocity–pressure coupling algorithm. For keeping stability of numerical solutions, the simulation time-step is dictated by the Courant condition with the Courant number equal 0.9.

In the numerical model, the measured velocity profiles as the inflow conditions were used (see Fig. 2). Geometrical length of the T-junction is 25 diameters (3.5 m) in x-direction, with a center of the mixing zone three diameters (0.42 m) from the cold inflow. In the z-direction the hot leg has length of 3.1 diameters (0.31 m) measured from the hot inflow to the center of the mixing zone. The last measurement location is situated two diameters upstream from the outflow. For the prescribed flow conditions, the main generation of turbulence is caused by the mixing of the two fluid streams. This was verified in our test simulations and it was also found in Westin et al. (2008). Hence, no perturbation of the inflow velocity conditions was applied.

In the previous work (Kuczaj and Komen, 2010) simulations were performed with a sequence of fully hexahedral meshes (see

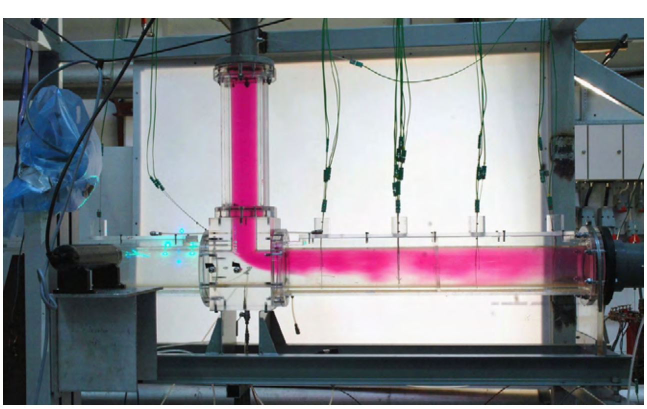

Fig. 1. Experimental setup of the T-junction experiment at Älvkarleby Laboratory, Vattenfall Research and Development (Andersson et al., 2006).