Nuclear Engineering and Design 278 (2014) 472–480

Contents lists available at ScienceDirect

Nuclear Engineering and Design

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

Towards the efficient turbulence closure for mixing phenomena in the core outlet of a nuclear reactor

M.S. Loginov[a], A. Shams[a,*], N. Amini[b], Y.A. Hassan[b]

[a] Nuclear Research and Consultancy Group (NRG), 1755 ZG, Petten, The Netherlands
[b] Department of Nuclear Engineering, Texas A&M University, 3133 TAMU, College Station, TX 77843-3133, United States

HIGHLIGHTS

- Quantitative assessment of various turbulence modeling approaches is performed.
- The assessment is performed in terms of accuracy and computational costs.
- The accuracy is evaluated by comparing the numerical results with the experimental data.
- The computational costs are assessed from the processor time spent for simulation.

ARTICLE INFO

Article history:
Received 9 February 2014
Received in revised form 30 July 2014
Accepted 1 August 2014

ABSTRACT

Proper mixing in the core outlet region of a nuclear reactor is a crucial issue for its safety assessment. Because of the high temperatures involved, it is especially important for the further development of high temperature reactors. Computational fluid dynamic (CFD) simulations are very useful for the assessment of such mixing phenomena, although they have to be accurate and cost efficient. Advanced approaches for turbulent modeling, such as large eddy simulation (LES), are usually more accurate and also more computationally expensive. On the other hand, the gain in accuracy with respect to the traditional models, such as Reynolds averaged Navier–Stokes (RANS) and its unsteady counterpart (URANS), is not quantified.

The present study provides a quantitative assessment of different approaches for turbulence modeling (RANS, URANS and LES) in terms of accuracy and computational costs. The accuracy is evaluated by direct comparison of the numerical results to the experimental data and the costs are assessed from the processor time spent for simulation. The results show that the average cost of unsteady simulations is higher than RANS almost by two orders of magnitude. Although the increase in the accuracy is not very big, the RANS simulations seem to be more efficient for the considered flow.

1. Introduction

Concepts of very high temperature reactors (VHTRs) and super critical water reactors (SCWRs) are considered in the frame of development of the generation IV nuclear reactors. Prismatic core VHTRs are currently one of the main designs under the investigation for generation IV program (US DOE, 2002). Consequently, various research projects have focused on the analysis and prediction of different thermal–hydraulic phenomena occurring within the core, upper and lower plenum of the core, and the hot duct of the prismatic core VHTRs. Recently, a lot of attention has been paid to the mixing phenomena in the lower plenum of a VHTR, where coolant flow impacts the bottom surface of the lower plenum and interacts with the support rods. The major concern is the distribution of the temperature coming from mixing of the high temperature helium coolant with the lower plenum flow. Similar phenomena occur in the upper plenum of a SCWR.

Research studies performed on the prismatic core VHTRs confirmed the need for benchmark experiments as well as CFD simulations predicting and characterizing the flow behavior and mixing phenomena within the lower plenum of the core for reactors of this type (McEligot and McCreery, 2004; McCreery and Condie, 2006). The estimates for flow exhausting into the lower plenum of a prismatic VHTR predict Reynolds numbers from 5000 under partial loading conditions to 50,000 at full-power operation.

An experimental study performed by Amini and Hassan (2009) on a conceptual model of the lower plenum of a VHTR demonstrated the complex nature of the lower plenum flows, coherent structures

* Corresponding author. Tel.: +31 224568152.
E-mail address: shams@nrg.eu (A. Shams).

http://dx.doi.org/10.1016/j.nucengdes.2014.08.002
0029-5493/

within the flow field, and the interaction of the coolant flow entering the lower plenum in form of impinging jets with the support rods present in the lower plenum and the bottom surface of the lower plenum. In this experimental study, to model the Reynolds numbers of the helium jets injecting to the lower plenum under different operating conditions of the reactor and depending on the location of the coolant channel within the core, a range of Reynolds number (4470–13,400) were used for the impinging jets.

A Reynolds averaged Navier–Stokes (RANS) simulation with $k-\epsilon$ model for jets in a channel with rod bundles is performed for the experimental configuration used in the investigation of Amini and Hassan (2009) for a case of dual impinging jets by Salpeter and Hassan (2008). It was not possible to obtain a grid-independent solution. A quantitative assessment of several RANS models for the heat transfer is summarized by Zuckerman and Lior (2005) with indication of cost for every model. The authors recommend v^2-f or $k-\omega$ SST model since they have the best accuracy and low to moderate costs. Another qualitative assessment of the suitability of RANS (Ridluan and Tokuhiro, 2008a) and URANS (Ridluan and Tokuhiro, 2008b) models for tube bundle flow suggested that unsteadiness is important in this flow. The accuracy of several RANS models was evaluated against experimental data using mean flow and Reynolds stresses, and was summarized per flow region.

Although many numerical studies have been performed on the accuracy of different approaches for turbulence modeling, most of them do not report the cost of the simulation. Obviously, LES is more precise, but also more expensive than RANS. It is good to know how big the gain in accuracy is, and if it is worth the additional costs. In other words, an estimate of overall efficiency is required, which takes into account both the accuracy and the costs. The objective of the current paper is to estimate the efficiency for turbulence modeling with LES, URANS and RANS approaches. A detailed discussion of the physical aspects of the flow is not the aim of the current paper; therefore, the capabilities of turbulence models for prediction of different flow features are not considered.

The present numerical study is performed for an experimentally investigated case, which is described in the next section. It is followed by details on the simulation method, assessment of accuracy and efficiency. The obtained results are discussed, and a recommendation on the most efficient model is given in the conclusion.

2. Experimental data

The experimental study of the mixing flow in the lower plenum of a conceptual VHTR was performed by Amini and Hassan (2009). The experimental test section consisted of a rectangular channel containing a rod bundle of vertical rods with two inlet jets impinging into it (Fig. 1). The channel had dimensions of 1016 mm × 76.2 mm × 76.2 mm with one outlet representing the single outlet in the lower plenum geometry (i.e. hot duct). The rod bundle consisted of 29 tubes fixed to the upper and lower surfaces of the channel. These tubes were set as “in-line” in z direction with 19 mm pitch and as “staggered” in x direction with displacement of 25.4 mm. The jet inlet pipes with the inner diameter of d = 10.16 mm were fixed at 10 mm from the top wall of the channel. It should be mentioned that the rod bundle tubes are the same as the tubes used for the inlet jets.

The flow rate of the jets was controlled independently. In the case considered, the Reynolds number $Re = U_b d/\nu$ was set to Re_1 = 11, 160 for the first jet, and Re_2 = 6250 for the second one. Here U_b is bulk velocity, and ν is kinematic viscosity of the working fluid. Water at room conditions was used in the experiment, and the bulk velocities were 1.03 and 0.57 m/s for the first and second jets, respectively.

Matched Index of Refraction (MIR) technique along with a 2D dynamic particle image velocimetry (DPIV) technique were applied in the experimental investigation to obtain the velocity fields in areas within the rod bundle (Amini and Hassan, 2009). Application of MIR technique provides optical access to the interior regions of the rod bundle which are normally blocked by the presence of the rods. Three vertical measurement planes are shown in Fig. 1. The instantaneous velocity components u_x and u_y were measured from 5000 snapshots taken during 5 s. After that, the mean velocities U_x and U_y, and the components of Reynolds stress tensor R_{xx}, R_{yy}, R_{xy} were calculated from these fields as:

$$U_x = \langle u_x \rangle, \qquad U_y = \langle u_y \rangle,$$
$$R_{xx} = \langle {u'_x}^2 \rangle = \langle u_x^2 \rangle - U_x^2, \qquad R_{yy} = \langle {u'_y}^2 \rangle = \langle u_y^2 \rangle - U_y^2, \qquad (1)$$
$$R_{xy} = \langle u'_x u'_y \rangle = \langle u_x u_y \rangle - U_x U_y$$

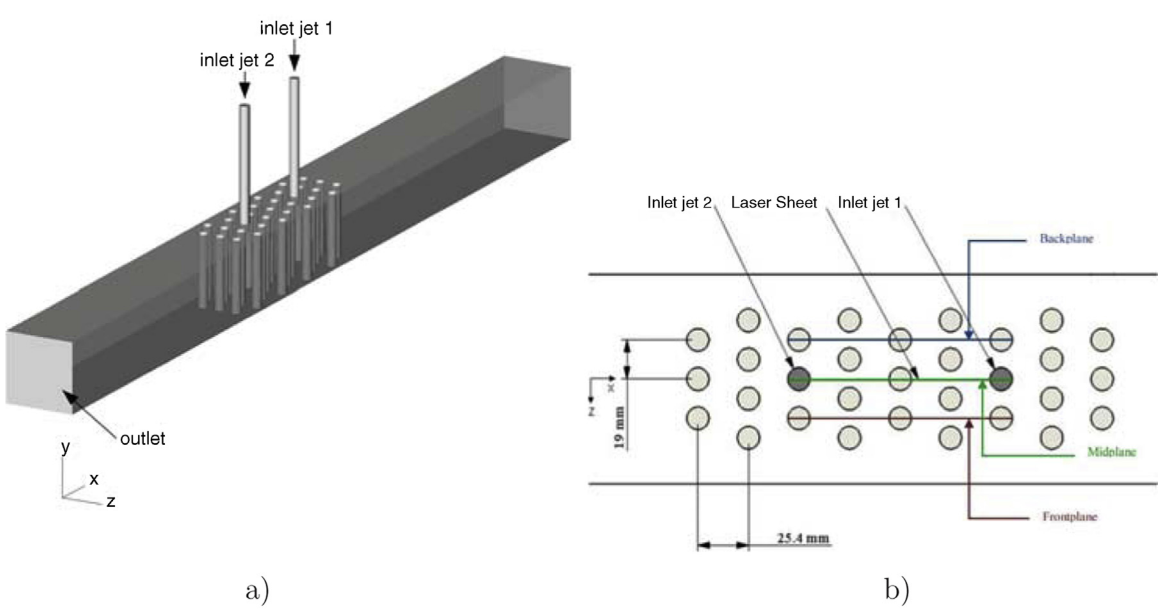

Fig. 1. Sketch of experimental setup (a) and bundle arrangement details (b).