Transactions of the Korean Nuclear Society Autumn Meeting
Gyeong-ju, Korea, October 25-26, 2012

A Study on the Feasibility of CFD Analysis in VHTR Gas Mixing Analysis

Hyeon-Kyeong Choi*, Jin-Su Kim, Jong-Woon Park
Dongguk University, 707, Sekjang-Dong, Gyeong Ju, South Korea
** Corresponding author: chk89418@naver.com*

1. Introduction

Computational fluid dynamic (CFD) is generally adopted for design process of a very high temperature reactor (VHTR). For this, benchmarking a CFD code is necessary for in reliability of the analysis. A study by Ridluan and Tokuhiro [1] shows that all the typical turbulence models do not sufficiently reproduce flow phenomena. Thus, a need is raised to confirm these limitations of a CFD before application to Korean VHTR problems. In this work, two typical validation cases such as a backward step and a tube bundle array are analyzed by using FLUENT [2] and they are compared with existing analysis and experimental data for reattachment lengths and local velocity profiles.

2. Methods and Results

2.1. Backwards step

Backward step flow is generally used to validate turbulence models in CFD. Two-dimensional geometry of Kim et al [3] where the expansion ratio (outlet height/inlet height) is 1.5 and Re number is 44,000 based on the inlet centerline velocity and the step height [4] is adopted for the present analysis

Figure 1 shows the overall picture of the geometry. At the inlet, a fully-developed turbulent velocity profile was applied (16.5m/s) at 1*h* upstream of the step and at the outlet a static pressure condition is applied. Quadrilateral dominant mesh is used and the total number of cells is 39,650. The mesh of recirculation zone is twice as dense as the far stream and the mesh around walls is three times as dense as the far stream.

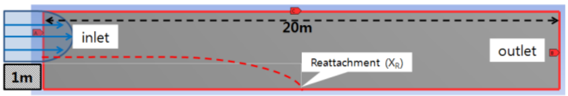

Fig.1 Geometry and Boundary condition of Backward step flow

Following turbulence models are used to compare with existing LBM results by Teixeira [5].

- Standard k-ε
- Standard k-ε with pressure-gradient extended law-of-the-wall (PGE-LW)
- RNG k-ε with PGE-LW

The velocity streamlines thus obtained is shown in Fig.2. The locations of the reattachment points (X_R/h) obtained are 5.1, 6.3 and 10.2 for standard k-ε, standard k-ε PGE-LW and RNG k-ε PGE-LW, respectively. These results somewhat differ from the experimental findings of Zhou et al [6] who reported X_R/h= 7.0 ± 0.5. Furthermore, existing results from lattice Boltzmann method (LBM) [5] differ from the data except the RNG k-ε PGE-LW model.

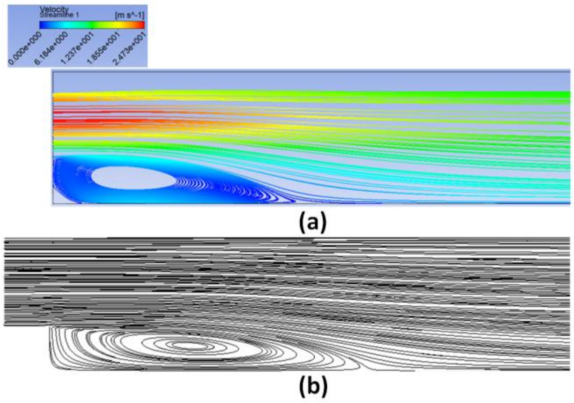

Fig.2 (a) Present standard k-ε PGE-LW (b) RNG k-ε PGE-LW by LBM [5]

Table I summarizes the computed and experimental values. Standard k-ε PGE-LW model gives reattachment length closest the experiment, which under-predicts the experimental value by 10.0%. Standard k-ε model gives slightly lower valves but applying the PGE-LW model increased the reattachment length by about 23.5% to the direction of the experimental value. For the RNG k-ε PGE-LW case, reattachment length is too long. Although the present results differ from existing experimental data, the reasonable accuracy of standard k-ε PGE-LW is confirmed by comparing with existing analyses [5, 6].

Table I: Reattachment length

Model	Standard k-ε	Standard k-ε PGE-LW	RNG k-ε PGE-LW
LBM [5]	6.1	6.3	7.2
Present	5.1	6.3	10.1
Zhou's analysis [6]	6.1	6.4	6.7
Zhou's Exp. [6]	7.0 ± 0.5		

2.2. Staggered tube bundle

The original tube array geometry and the analysis cell used in this analysis is shown in Fig. 3 (tube diameter = 21.7mm, pitch = 45mm [7]). The

Reynolds number is 18,000 based on the diameter and the properties of liquid water. Quadrilateral dominant mesh is used as well and the total number of cells is 42,952 and steady calculation is performed. The standard k-ε turbulence model is applied to compare present results with existing analysis [1] and the experiment [7]. Experimental mean velocity upstream of the array is 1.06m/s and the maximum speed around rods in domain is 3.06m/s [7].

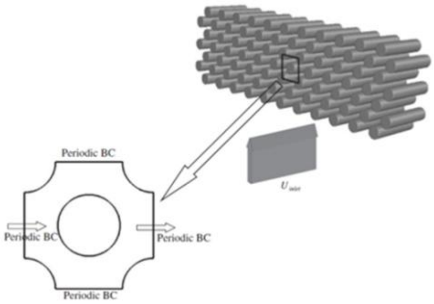

Fig.3 Geometries of the actual and simplified staggered tube bundle array by Ridluan and Tokuhiro

Figure 4 shows present steady velocity contours compared with existing unsteady analysis of Ridluan and Tokuhiro using Reynolds stress model (RSM) [1]. Present standard k-ε steady calculation is very close to the unsteady analysis [1]. Also, the two calculations are close to experimental data [7] for streamwise and spanwise velocities at x=11mm as shown in Fig.5 (only streamwise velocity shown). However, the present velocity profiles at y=5 mm slightly differ from experimental and Ridluan and Tokuhiro's result. However, overall results are close to experimental data. Therefore, it can be stated that the steady standard k-ε calculation is reasonable for tube bundle engineering analysis.

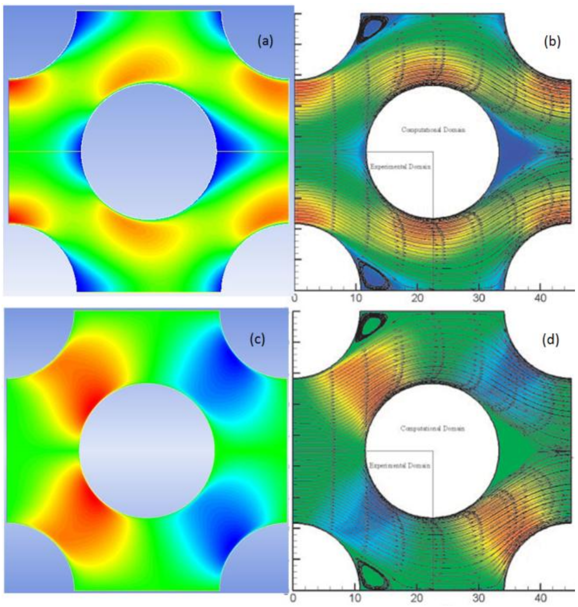

Fig.4 (a) present streamwise velocity, (b) streamwise velocity [1] (c) present spanwise velocity (d) spanwise velocity [1]

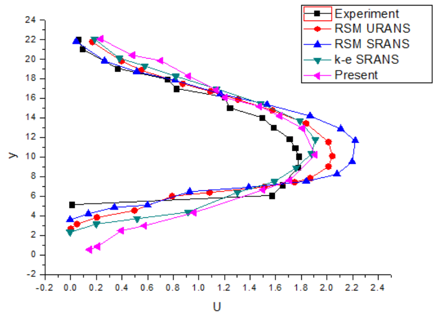

Fig.5 the streamwise velocity at x=11mm

3. Conclusions

In order to confirm limitations of a CFD application to Korean VHTR problems, two CFD validation cases such as a backward step and a tube bundle array are analyzed by using FLUENT and they are compared with existing analysis and experimental data. For backward-facing step flow of Re= 44,000, it is found that standard k-ε PGE-LW model is more appropriate than other turbulence models. After comparing the present steady analysis of flow through staggered tube bundle for Re= 18,000 with existing unsteady calculation and experimental data, it can be concluded that standard k-ε steady calculation is reasonably appropriate for cost-effective VHTR lower plenum analysis.

4. References

[1] A. Ridluan, A. Tokuhiro, Benchmark Simulation of Turbulent Flow through a Staggered Tube Bundle to Support CFD as a Reactor Design Tool. Part II URANS CFD Simulation, Journal of Nuclear Science and Technology, Vol. 45, No. 12, p. 1305, 2008.

[2] J.Y. Kim, ANSYS CFD FLUENT, ANSYS Inc., Canonsburg, 2011.

[3] J. Kim, S.J. Kline, J.P. Johnson, Investigation of a Reattachment Turbulent Shear Layer: Flow over a Backward-Facing Step, Journal of Fluids Engineering, Vol. 102, p. 302, 1980.

[4] C.K. Chen, T.S. Yen, Y.T. Yang, Lattice Boltzmann Method Simulation of a Cylinder in the Backward-facing Step Flow with the Field Synergy Principle, International Journal of Thermal Sciences, Vol. 45, p. 982, 2006.

[5] C.M. Teixeira, Incorporating Turbulence Models into the Lattice-Boltzmann Method, International Journal of Modern Physics C, International Journal of Modern Physics C, Vol. 9, p. 1159, 1998.

[6] Y. Zhou, G. Vahala, S. Thangam, "Development of a Recursion RNG-Based Turbulence Model", NASA Langley Research Center, Hampton, VA 1993.

[7] O. Simonin, M. Barcouda, Measurements and Prediction of Turbulent Entering a Staggered Tube Bundle, 4th Int. Symp. Applications of Laser Anemonetry to Fluid Mechanics, Vol. 5, p. 23, 1988.