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

Performance Comparison of the Commercial CFD Software for the Prediction of Turbulent Flow through Tube Bundles

Gong Hee Lee [a*], Young Seok Bang [a], Sweng Woong Woo [a]
[a] *Korea Institute of Nuclear Safety, Daejon, Republic of Korea*
[*] *Corresponding author: ghlee@kins.re.kr*

1. Introduction

Because turbulent flow through tube bundles can be found in many important industrial applications, such as PWR reactor, steam generator, CANDU calandria and lower plenum of the VHTR, extensive studies have been made both experimentally and numerically. Although recently licensing applications supported by commercial CFD software are increasing, there is no commercial CFD software which obtains a licensing from the regulatory body until now. Therefore, it is necessary to perform the systematic assessment for the prediction performance of the commercial CFD software. The main objective of the present study is to numerically simulate turbulent flow through both staggered [1] and in-line tube bundle [2] using the two popular commercial CFD software, ANSYS CFX [3] and FLUENT [4] and to compare the simulation results with the experimental data for the assessment of these software's prediction performance.

2. Numerical Method and Results

2.1 Staggered tube bundle

The experimental data of Paul et al. [1] for turbulent flow through the staggered tube bundles are used for the comparison. As shown in Fig. 1, the staggered tube bundles consist of 6 rows of tubes of outer diameter of 25.4 mm. The longitudinal distance between two tubes is 53.34 mm and the transverse distance is 96.52 mm. The longitudinal distance between inlet and first row of tube is 1.116m.

The flow is assumed to be steady, incompressible and turbulent. Uniform velocity with the magnitude of 0.34 m/s which correspond to Reynolds number 9,300 is imposed at inlet boundary. Turbulence intensity at inlet is set to be 4%. At the outlet boundary, static pressure is specified. No-slip condition is applied on the solid wall.

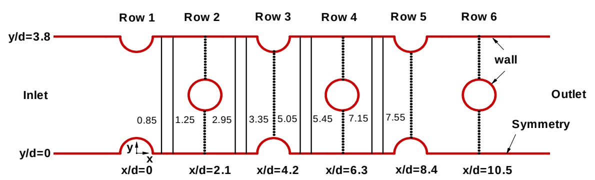


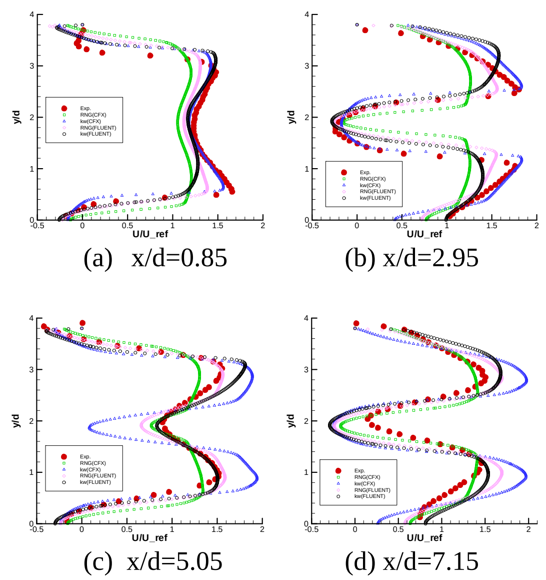
Fig. 1. Schematic diagram of staggered tube bundle test rig

Table I: Summary of the numerical modeling

Items	ANSYS CFX	FLUENT
Turbulence model	RNG k-ε & k-ω	
Wall treatment	Scalable wall function	Enhanced wall treatment
Convergence criteria	$\leq 10^{-6}$	
Convection term	2^{nd} order	

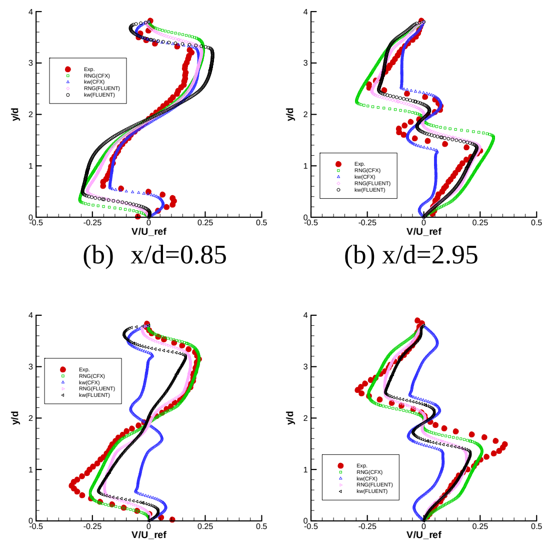
Because of the symmetric geometry (y-direction), the computational domain considers only half of the experimental domain with symmetry boundary condition. Two different turbulence models, that is, RNG (ReNormalization Group) k-ε model and standard k-ω model are used. Table I shows the summary of the numerical modeling. A total number of cells with multi-block structured hexahedral shape are 643,200.

Fig. 2 and 3 show the comparison of the streamwise and transverse mean velocity profile at the selected axial locations. For the streamwise velocity a combination of FLUENT and RNG k-ε model gives the superior prediction performance to a combination of ANSYS CFX and RNG k-ε model except at x/d=7.15. In case of k-ω model, ANSYS CFX predicts well in the developing flow region, whereas FLUENT does well in the developed flow region.



(a) x/d=0.85 (b) x/d=2.95
(c) x/d=5.05 (d) x/d=7.15

Fig. 2. Comparison of streamwise mean velocity profile at the selected axial locations



(b) x/d=0.85 (b) x/d=2.95

(c) x/d=5.05 (d) x/d=7.15

Fig. 3. Comparison of transverse mean velocity profile at the selected axial locations

For the transverse velocity a combination of ANSYS CFX and k-ω model predicts flow differently in comparison with that of the experimental data from x/d=2.95 to x/d=7.15.

2.2 In-line tube bundle

The experimental data of Hadaller et al. [2] for turbulent flow through an in-line tube bundle are used for the comparison. As shown in Fig. 4, the in-line tube bundles consist of 4 columns wide by 24 rows long tubes enclosed in a rectangular box (0.286m width by 0.2m height). A diameter and pitch of tube is 71.4mm and 33mm respectively. The first pressure tap is located five pitch lengths into the tube bank. The next two pressure taps are spaced at eight pitch lengths each further into the channel.

The flow is assumed to be steady, incompressible and turbulent. Uniform velocities with the magnitude of 0.054 m/s which correspond to Reynolds number 2,746 are imposed at inlet boundary. Turbulence intensity at inlet is set to be 4.87%. At the outlet boundary, static pressure is specified. No-slip condition is applied on the solid wall. Because of the symmetric geometry (z-direction), the computational domain considers only half of the experimental domain with symmetry boundary condition. Two different turbulence models, that is, k-ε model and RNG k-ε model are used. The numerical modeling is same as Table I except the turbulence model. A total number of cells with multi-block structured hexahedral shape are 2,334,000.

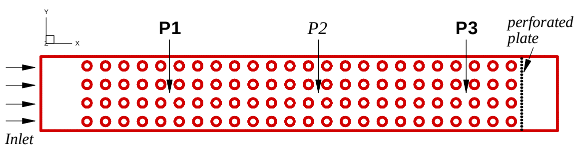


Fig. 4. Schematic diagram of in-line tube bundle test rig

The comparisons of the experimental and calculated pressure drops (Δp=P1-P3) are summarized in Table II. Difference between the measurement and the prediction is above about 21.7%. These differences increase significantly in comparison with previous study [5] with 1st order upwind scheme for convection term and less dense grid. A combination of FLUENT and k-ε model shows the best prediction performance.

Table II: Comparison of the magnitude of pressure drop

Item	Exp.[2]	ANSYS-CFX		FLUENT	
		k-ε	RNG k-ε	k-ε	RNG k-ε
Δp [Pa]	28.2	17.6	16.2	22.1	19.8
Error [%]	-	*37.6*	*42.7*	*21.7*	*29.9*

Note) Error [%] = (Exp.-Comp.)/Exp.×100

3. Conclusions

In this study, numerical analysis of turbulent flow through both staggered and in-line tube bundle using the two popular commercial CFD software, ANSYS CFX and FLUENT, was conducted and the simulation results were compared to experimental ones to assess the prediction performance of those software. The major conclusion could be summarized as follows:

1) Simulation results showed the large difference with the measurement especially for the in-line tube bundle.
2) FLUENT showed the overall superior prediction performance to ANSYS CFX for the converged solution.

Acknowledgement

This study was conducted under the financial support of the National Research Foundation of Korea [project title: Development of Safety Evaluation Capability on New Design Features]. The authors also gratefully thank Mr. Kang Dong-Gu for valuable comments.

REFERENCES

[1] S. S. Paul, M. F. Tachie, and S. J. Ormiston, "Experimental Study of Turbulent Cross-Flow in a Staggered Tube Bundle using Particle Image Velocimetry," International Journal of Heat and Fluid Flow, vol. 28, pp. 441-453, 2007.

[2] G. I. Hadaller, R. A. Fortman, J. Szymanski, W. I. Midvidy, and D. J. Train, Frictional Pressure Drop for Staggered and In-line Tube Banks with Large Pitch to Diameter Ratio, 17th Annual Canadian Nuclear Society Conference, June 9-12, 1996, Fredericton, New Brunswick, Canada.

[3] ANSY CFX-Solver Modeling Guide, Version 13.0, ANSYS Inc., 2010.

[4] FLUENT User's Guide, Version 6.2, FLUENT Inc., 2006.

[5] G.H. Lee and S.W. Woo, "Benchmark Simulation of Turbulent Flow through a Tube Bundle," Trans. of KNS Autumn Meeting, October 27-28, 2011, Gyeongju, Korea.