Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 27-28, 2011

Benchmark Simulation of Turbulent Flow through a Tube Bundle

G.H. Lee [a*], S.W. Woo [a]
[a]*Korea Institute of Nuclear Safety, Daejon, Republic of Korea*
[*]*Corresponding author: ghlee@kins.re.kr*

1. Introduction

Because flow and heat transfer in tube bundles have many important industrial applications, such as CANDU calandria and lower plenum of the VHTR (Very High Temperature Reactor), extensive studies have been made both experimentally and numerically. Accurately estimating the local subcooling of the moderator inside CANDU calandria under either normal operational condition or transient conditions is one of the major concerns in the CANDU safety analysis. To have confidence in the design of the VHTR it is necessary to simulate correctly the physical phenomena of importance in the lower plenum.

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 commercial flow solver, FLUENT [3], and compare the simulation results with experimental ones to assess which turbulence models give the most reliable results.

2. Numerical Method and Results

Five different types of the Reynolds-averaged Navier-Stokes (RANS)-based turbulence models, that is, standard k-ε, RNG (ReNormalization Group) k-ε, standard k-ω, SST (Shear-Stress Transport) k-ω and Reynolds Stress Model (RSM), are used to assess the prediction capability of the flow through both staggered and in-line tube bundle. The SIMPLE algorithm is used for pressure-velocity coupling. More detailed descriptions of the numerical models can be found in the reference [3].

2.1 Staggered tube bundle

The experimental data of Simonin and Barcouda [1] for turbulent flow in a staggered tube bundle are used for benchmark simulation. Test rig consists of seven horizontal staggered rows of tubes with a diameter of 21.7 mm. The staggered tubes are uniformly spaced in both streamwise and spanwise directions with a distance of 45 mm. Mean velocity profiles are obtained using LDA (Laser Doppler Anemometry) at five locations as shown in Fig. 1.

Periodic boundary conditions are used for open boundaries in both streamwise and spanwise directions, as shown in Fig. 1. The mass flow of 40.75 kg/sec is imposed to provide a flow through the computation domain. No-slip condition is applied on the solid wall. Enhanced wall treatment is used to model the near-wall region.

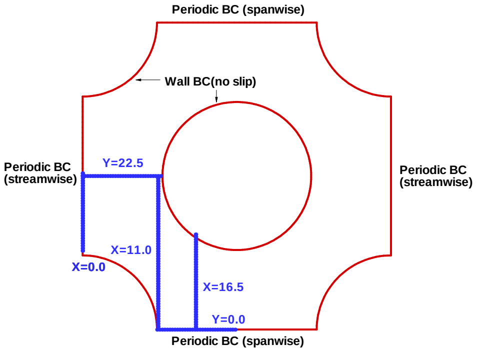
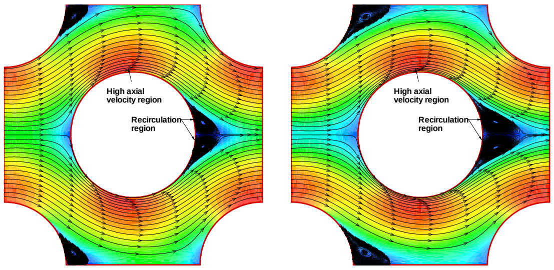
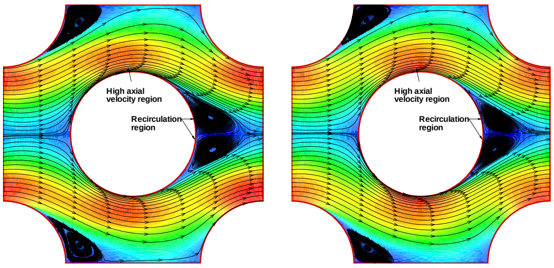


Fig. 1. Schematic diagram of staggered tube bundle test rig

The flow is assumed to be unsteady, incompressible and turbulent. The 2nd order QUICK scheme for the convective terms is used. The default under-relaxation factors are used. The residual tolerances for convergence are set to 1.0×10^{-6}. The 2nd order implicit scheme is used for the unsteady calculation. The time step size of 1.0×10^{-7} sec is used to capture the large scale motions without unphysical oscillations.



(a) k-ε (b) RNG k-ε

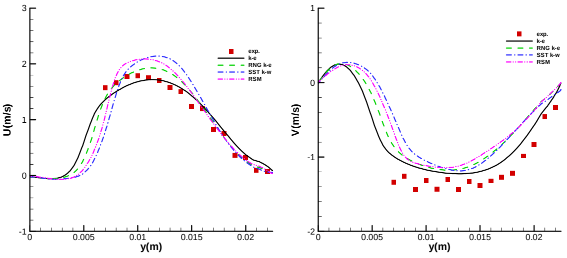


(c) SST k-ω (d) RSM

Fig. 2. Axial mean velocity contour and streamtraces

Fig. 2 shows the computed axial mean velocity contours and streamtraces. The overall features of turbulent flows predicted by turbulence models are similar in principle. The axial mean velocity reaches a maximum near the top (side) of the tube. The smallest recirculation region behind the tube is predicted by the standard k-ε model.

Fig. 3 shows the comparison of the axial and spanwise mean velocity profile at x=11mm. The standard k-ε model gives the best agreement with the experimental data.



(a) axial (b) spanwise

Fig. 3. Comparison of mean velocity profile at x=11mm

2.2 In-line tube bundle

The experimental data of Hadaller et al. [2] for turbulent flow in an in-line tube bundle are used for benchmark simulation.

As shown in Fig. 4, the in-line tube bundle consisted 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.

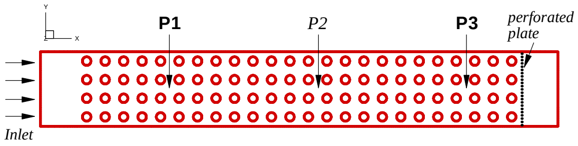


Fig. 4. Schematic diagram of in-line tube bundle test rig

Uniform velocity with the magnitude of 0.054m/s, which corresponds to Reynolds number 2,746, is imposed at inlet boundary. Turbulence intensity at inlet is 4.87% and the turbulence length scale is set to be 11.45mm. The Reynolds stresses at the inlet are derived from the assumption of an isotropic turbulence by using the precalculated turbulence kinetic energy or the turbulence intensity. At the outlet boundary, a zero normal gradient for all flow variables except the pressure is applied. No-slip condition is applied on the solid wall. In FLUENT [3], either wall function or enhanced wall treatment can be used to model the near-wall region. In this case, the prediction with wall function is shown because wall function gives the better prediction results than enhanced wall treatment.

The flow is assumed to be steady, incompressible and turbulent. The 1st order accurate upwind differencing for the convection terms of each governing equation is used because this differencing scheme gives the better prediction results and convergence than the 2nd order accurate upwind differencing in this case. The convergence criterion is set to the scaled residuals of 10^{-5} for all relevant variables.

The comparisons of the experimental and calculated pressure drops ($\Delta p = P1 - P3$) are summarized in Table I. Difference between the measurement and the current prediction is below about 5.3%. Two equation turbulence models except the RNG k-ε model give the better prediction than RSM with linear pressure-strain model.

Table I: Comparison of the magnitude of pressure drop

	Exp.[2]	k-ε	RNG k-ε	k-ω	RSM
Δp [Pa]	28.2	28.9	29.6	28.8	29.6
Error [%]	-	-2.49	-5.29	-2.29	-5.13

3. Conclusions

In this study, benchmark simulation of turbulent flow through both staggered and in-line tube bundle using the commercial flow solver, FLUENT, was conducted and the simulation results were compared to experimental ones to assess which turbulence models give the most reliable results. The major conclusion could be summarized as follows:

1) The overall features of turbulent flows predicted by turbulence models were similar in principle.
2) Although the RSM had greater potential to give accurate predictions for turbulent flows in tube bundle, this model did not yield results that were clearly superior to the two-equation turbulence models in both the mean velocity and the pressure drop.

Acknowledgement

The authors gratefully thank Dr. Johnson in the Idaho National Laboratory for valuable comments.

REFERENCES

[1] O. Simonin, M. Barcouda, Measurements of Fully Developed Turbulent Flow across Tube Bundle, Proc. Third Int. Symp. on Applications of Laser Anemometry to Fluid Mechanics, 1986, Lisbon, Portugal.

[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] FLUENT user's guide, Version 6.2, FLUENT Inc., 2006.