Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting and
8th International Conference on Nanochannels, Microchannels, and Minichannels
FEDSM-ICNMM2010
August 1-5, 2010, Montreal, Canada

FEDSM-ICNMM2010-30109

COMPARATIVE STUDIES OF TURBULENCE MODELS APPLICATION IN THERMAL HYDRAULIC ANALYSIS OF NUCLEAR REACTOR CORE

ZENG Heyi
College of Nuclear Science and Technology,
Harbin Engineering University
Harbin, Heilongjiang, China
Nuclear Safety and Simulation Technology Key
Laboratory of National Defense Disciplines,
Harbin Engineering University
Harbin, Heilongjiang, China

GUO Yun
College of Nuclear Science and Technology,
Harbin Engineering University
Harbin, Heilongjiang, China
Nuclear Safety and Simulation Technology Key
Laboratory of National Defense Disciplines,
Harbin Engineering University
Harbin, Heilongjiang, China

ABSTRACT

Rod bundles are essential elements of pressurized water nuclear reactors. They consist of tightly packed arrays of rods, which contain the nuclear fuel and are surrounded by flowing liquid coolant. Flow phenomena in the subchannels bounded by adjacent rods are quite complex and exhibit patterns not present in pipe flows. Development of nuclear reactors and of fuel assemblies requires fluid dynamics analysis activities. The detailed prediction of velocity and temperature distributions inside a rod bundle is one of the main objectives of the current research in reactor thermal hydraulics. Computational fluid dynamics (CFD) simulation is of great interest for the design and safety analysis of nuclear reactors since it has recently achieved considerable advancements. In the present studies, numerical simulation were performed on developed turbulent flow through core subchannels with configurations of triangle and square lattice, and impact of different turbulence models built-in software package FLUENT upon simulation results of velocity distribution and hydraulic characteristics in channels with complicated geometry were compared and analyzed. Results show that simulation result greatly depends on turbulence models. Due to the complicated geometric construction, the complicated three-dimensional turbulent flow shows highly anisotropic characteristics. Turbulence models assuming isotropic turbulent viscosity failed to predict secondary flow phenomena during turbulent flow in fuel assembly channel. By solving Reynolds stresses transport equations, more elaborate Reynolds stress model (RSM) can catch secondary flow accurately. The present studies have provided valuable references and guidelines for further investigation on convective heat transfer simulation in complicated geometry and thermalhydrulic analysis of nuclear reactor core.

1. INTRODUCTION

In the design of a nuclear reactor, it is very important to predict the detailed flow and temperature distributions in reactor core. This is because a safe and reliable operation of a reactor system relies on an accurate thermal-hydraulic design. Rod bundles are essential elements of nuclear reactors. They consist of tightly packed arrays of rods, which contain the nuclear fuel and are surrounded by axially flowing liquid coolant. Liquid coolants can be water, gas or liquid metals, generally the flow are turbulent. As the flow and temperature field can not be obtained with exact analytical method, numerical modeling plays a vital role. Recently, with the increasing power of computers and computing techniques, computational fluid dynamics (CFD) calculation become an appropriate numerical tool which is in principle able to capture all the flow and heat transport details.

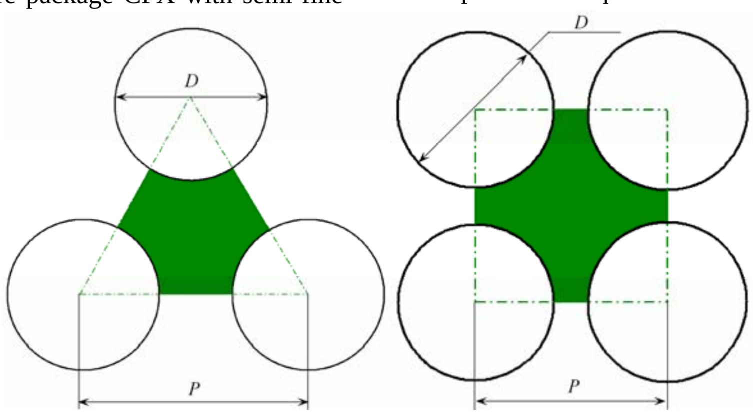
In recent years Direct Numerical Simulation (DNS) for low Reynolds number and Large Eddy Simulations (LES) were performed to obtain information about the complicated secondary flow phenomena and mixing process in tightly packed fuel pin subassemblies with triangular array arrangement, and simulation results show good agreement with experiment data [1, 2, 3, 6, 10]. However, high Reynolds numbers and complex geometry do not permit the adoption of fine techniques such as DNS or LES [4-6]. Practical calculations at present time must therefore be based on Reynolds Averaged Navier-Stokes Equations (RANS), with the aid of turbulence modeling. Turbulence models used in previous researches are mainly limited to the first order closure models, which assume isotropic eddy diffusivity in modleling the Reynolds stress tensor, and therefore anisotropic effect are not accounted for. However in the complex geometric channel flow such as subchannels in a core, strongly anisotropic behavior exists and turbulence driven secondary flow occur. During the last few years, more and more efforts have been made to assess the applicability of existing CFD codes. Baglietto and Ninokata [7] conducted evaluation of various turbulence models for calculation of detailed coolant velocity distribution in tight triangular lattice fuel bundle, and their results shown that all linear models, due to simple eddy-viscosity assumption, have shown to produce consistent predictions, but are not able to reproduce the wall shear stress behavior and over predict the diversity of the velocity fields between the channel center and the narrow gap region. Meanwhile, performances of various turbulence models built-in most widely used commercial codes including STAR CD, CFX and FLUENT to simulating the flow and heat transport in reactors having been evaluated by more and more researchers [8, 9]. Two types of turbulence models, i.e. the ε type turbulence models and the ω type turbulence models were comparative studied, and the ε type Reynolds stress model of Speziale (SSG) built-in software package CFX with semi-fine mesh structure was recommended for the application of simulating supercritical water convective flow in subchannels.

In the present study, turbulence models built-in commercial code FLUENT were comparative studied, and their influences on the simulations of turbulent flow in subchannels with complicated configurations of both triangular and square arranged bare rod bundle with a pitch to diameter ratio (P/D) of 1.5. Efforts were emphasized on the applicability of turbulence models to reproduce the turbulence-driven secondary flow phenomena in channels with complicated geometric configurations.

2. PHYSICAL MODELS AND NUMERICAL METHODS

Steady state developed turbulent water flowing with constant properties through subchannels with configurations both in equilateral triangle and square lattice configurations, as shown in Fig.1, were numerical investigated in the present studies, and the shaded areas in Fig.1 denote their computation domains, both configured with triangular lattice and square lattice, respectively.

The computations were carried out using a specified mass flow rate and a periodic boundary condition in the streamwise direction, by which the velocity field on the outlet plane was inserted back on the inlet plane. The use of periodic boundary condition permitted the attainment of fully developed conditions in a relatively short flow domain, thus reducing the computational requirements.



a. triangular lattice arrangement   b. square lattice arrangement

Fig. 1 Schematic of physical problem and computational domains

A pressure based steady solver is used to deal with the mass, momentum, energy and turbulence equations. The second order QUICK (Quadratic Upwind Interpolation of Convective Kinematics) scheme is used for discretization of momentum equations, and the second order upwind scheme is used to discretize the other equations for reducing numerical viscosity. The SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm is adopted for the treatment of the pressure-velocity coupling problem. The periodic boundary condition was applied on the inflow and outflow planes, whereas the no slip condition was applied to the rod walls and symmetrical condition to the interfaces of subchannels. Unstructured meshes with hexahedral elements were used, having 6 boundary elements increasing at a growth factor of 1.2. The values of y^+ for the first nodes close to the walls were kept to be lower than 1.0 for all runs. To permit application of the periodic boundary condition, the meshes at the inlet and outlet were matched by linking both faces. The convergence criteria are set in such a way that the residual for the conservation equations is less than 10^{-6}.

3. RESULTS AND DISCUSSION

Based on the numerical methods as aforementioned, in the following sections simulation results for developed turbulent flow in subchannels both in triangular and square lattice arrangements were shown, the influences of turbulence models on the results were analyzed and the applicability of turbulence models built-in commercial software package FLUENT in thermal hydraulic analysis of nuclear reactor core with complex geometric configurations were comparative studied.

3.1 TRANGULAR LATTICE SUBCHANNEL

Fig. 2 shows one typical cross-sectional vector plot obtained from various models during developed turbulent flow in triangular subchannel. Compared with those obtained by large eddy simulation and direct numerical simulation [1～3], RSM model can accurately reproduce these complicated secondary flow phenomena, while other one- or two-equation models introducing isotropic eddy diffusive concept and assuming Boussinesq hypothesis failed to correctly simulate those turbulence driven secondary flow phenomena occurred in triangular lattice subchannels. It must be noted that only the results from RSM, SST k-w and RNG k-e models were shown in Fig. 2 for clarity.

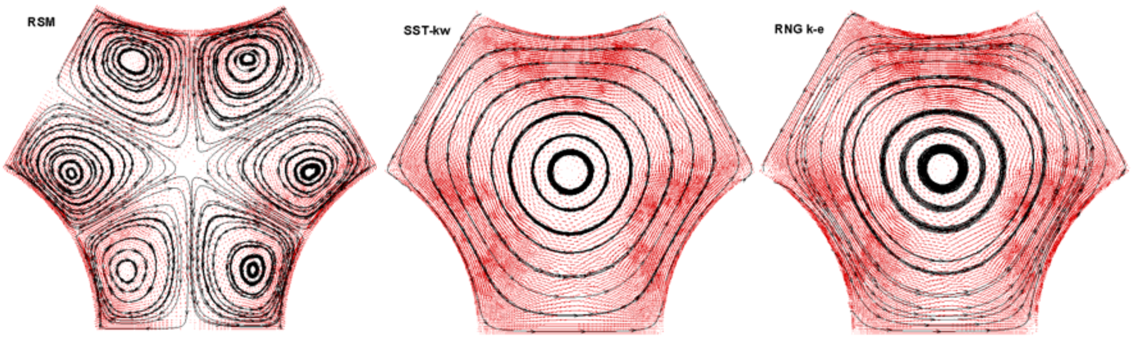


Fig.2 Comparison of cross sectional secondary flow for triangular lattice based on different models

The main velocity distribution during developed turbulent flow in triangular channel obtained from different turbulence models were compared in Fig. 3. It can be seen that main stream velocity distributions obtained from different turbulence models are same alike in whole. But going through Fig. 3 with more carefulness, one can see the influences of turbulence models on the simulation results, particularly in narrow gap regions where the main stream velocity gradient obtained from RSM changes more acute in the narrow gap regions than those from other one- or two-equation turbulence models, such as SST k-w and RNG k-e turbulence model, as shown in Fig. 3.

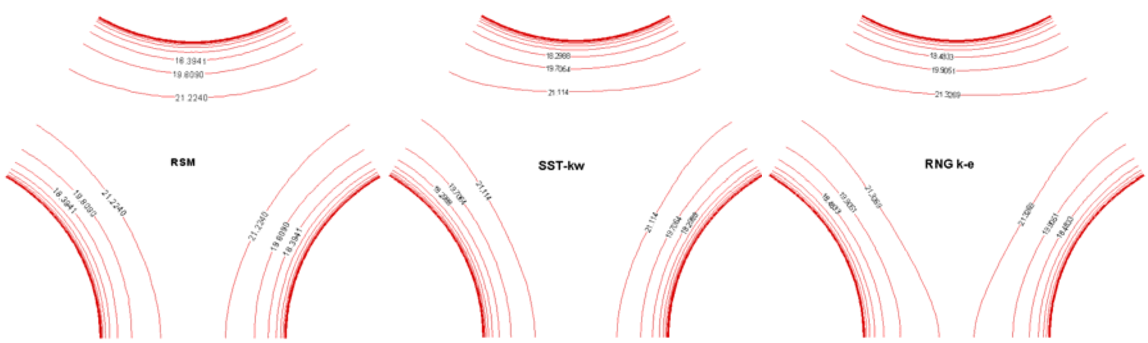


Fig. 3 Comparison of the main velocity distributions for triangular lattice based on different models

The influences of turbulence models upon the simulation results can also be found quantitatively as shown in table 1. The simulated friction factors for developed turbulent flow in triangular subchannel from different turbulence models were listed in table 1, where the factors were normalized by result obtained from RSM model for distinct comparison. One can see from table 1 that the simulated friction coefficient depends on turbulence model. In present case, friction factor obtained from SST k-w model are most close to those obtained from RSM model, while RNG k-e turbulence model gives the lowest value among the these turbulence models, and the discrepancy can reach up to 40%.

Table 1 Comparison of calculated friction coefficient for channel in triangular lattice

Turbulence models	RSM	S-A	SST k-w	RNG k-e	Realizable k-e
Normalized values	1	0.743	1.032	0.592	0.796

3.2 SQUARE LATTICE SUBCHANNEL

The obtained secondary flow for developed turbulent flow in square arrayed subchannel was shown in Fig. 4. One can see again that the turbulence models introducing isotropic eddy diffusive concept can not reproduce complex flow phenomena during developed turbulent flow in square lattice subchannels. According to these isotropic models, only one large eddy (secondary flow cell), while the simulation results obtained from RSM model clearly show that there are eight secondary flow cells exist in whole cross section of one subchannel in square lattice configuration. The simulated secondary flow phenomena obtained from RSM model agreed quite well with those obtained by Ikeno and Kajishima using large eddy simulation [10]. Again results obtained from SST k-w turbulence model were shown in Fig. 4 to be compared with those from RSM model for clarity.

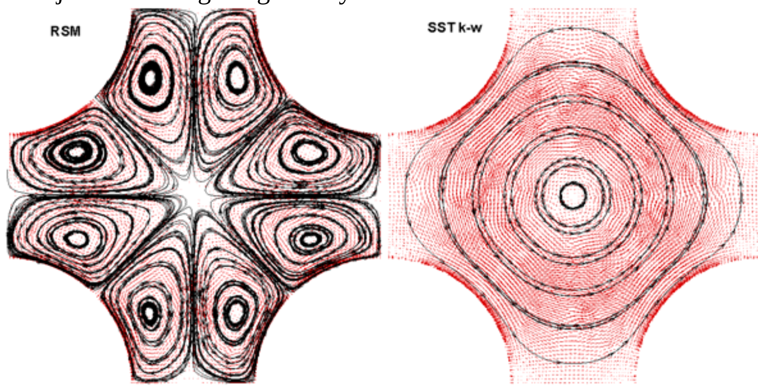


Fig. 4 Comparison of cross sectional secondary flows for square lattice based on different models

The main velocity distribution during developed turbulent flow in square arrayed subchannel obtained from different turbulence models were shown in Fig. 5. One can see that the main stream velocity distributions obtained from different turbulence models are same alike in general. However the influences of turbulence models on the simulation results, particularly in narrow gap regions can be found with more carefulness, as the case in triangular lattice subchannel.

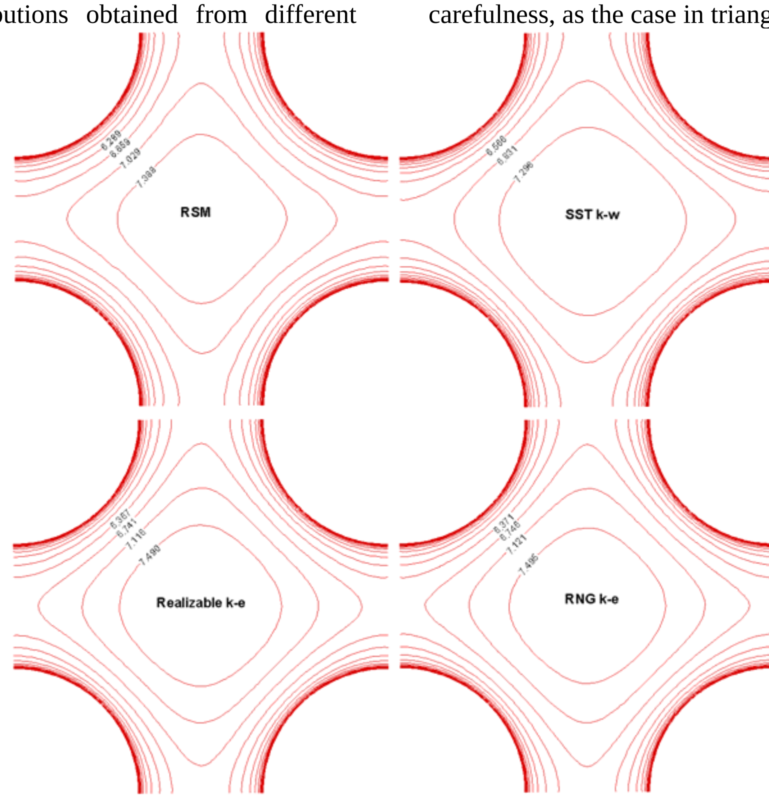


Fig. 5 Comparison of the main velocity distributions for square lattice based on different models

Table 2 Comparison of calculated friction coefficient for channel in square lattice

Turbulence models	RSM	S-A	SST k-w	Standard k-e	RNG k-e	Realizable k-e
Normalized values	1	1.0164	0.9093	0.9659	0.9525	0.9433

The quantitative influences of turbulence models upon the simulation results were shown in table 2, where the simulated friction factors normalized by result obtained from RSM model for developed turbulent flow in square lattice subchannel from different turbulence models were listed. As the case of developed turbulent flow in triangular lattice subchannel, the simulated friction coefficient also depends on turbulence model. In present case smaller than 10% of discrepancy exist, and results obtained from S-A turbulence model are the most close to those obtained from RSM model.

4. CONCLUDING REMARKS

Computational fluid dynamics (CFD) simulation is of great interest for the design and safety analysis of nuclear reactors since it has recently achieved considerable advancements. In the present studies numerical simulations were performed on the developed turbulent flow in subchannels both in triangular and square lattice configurations and the applicability of turbulence models built-in commercial CFD software package FLUENT to the thermal hydraulic analysis of nuclear reactor core. Results show that simulation results greatly depends on turbulence models. Due to the complicated geometric construction in nuclear reactor core, complicated three-dimensional turbulent flow shows highly anisotropic characteristics. One equation Spalart-Allmaras turbulence model and two-equation, such as k-epsilon, k-omega turbulence models assuming isotropic turbulent viscosity failed to predict secondary flow phenomena during developed turbulent flow in subchannels both in triangular and square lattice configurations. Unlike one- or two-equation turbulence models, solving Reynolds stresses transport equations, Reynolds stress model (RSM) accounts for the effect of flow anisotropy and can catch those complicated secondary flow phenomena accurately. As can be seen from Fig. 2 and Fig. 4 that secondary flow exist in developed turbulent flow in subchannels both in triangular and square lattice configurations. Results obtained from RSM model show that one secondary flow cell exist in each one-eighth subchannel of the square lattice and in each one-sixth subchannel of the triangular lattice, while results obtained from one- or two equation turbulence models are not able to reproduce these complicated phenomena occurred in subchannels with complicated geometric configurations. The present results from RSM model agreed quite well with those from direct numerical simulation and large eddy simulation available in public literatures, and the Reynolds stress model built-in commercial software package FLUENT is highly recommended to the application of thermal hydraulic analysis of nuclear reactor core with complicated geometric configurations.

ACKNOWLEDGMENTS

Project (HEUFT07092) Supported by the Fundamental Research Foundation of Harbin Engineering University and Project (10805011) Supported by National Natural Science Foundation of China are gratefully acknowledged.

REFERENCES

[1] Mayer G., Házi G., 2006, Direct numerical and large eddy simulation of longitudinal flow along triangular array of rods using the lattice Boltzmann method. Mathematics and Computers in Simulation, 72, 173-178.

[2] Mayer G., Páles J., Házi G.., 2007, Large eddy simulation of subchannels using the lattice Boltzmann method. Annals of Nuclear Energy, 34, 140-149.

[3] Ninokata H., Merzari E., Khakim A., 2009, analysis of Reynolds number turbulent flow phenomena in nuclear fuel pin subassemblies of tight lattice configuration. Nuclear Engineering and Design, 239, 855-866.

[4] Simoneau J-P, Champigny J., Gelineau O., Applications of large eddy simulations in nuclear field. Nuclear Engineering and Design (2008), doi:10.1016/j.nucengdes.2008.08.018.

[5] Jeong J. H., Han B. S., 2008, Coolant flow field in a real geometry of PWR downcomer and lower plenum. Annals of Nuclear Energy, 35, 610-619.

[6] Baglietto E., Ninokata H., Misawa T., 2006, CFD and DNS methodologies development for fuel bundle simulations. Nuclear Engineering and Design, 236, 1503-1510.

[7] Baglietto E., Ninokata H., 2005, A turbulence model study for simulating flow inside tight lattice rod bundles. Nuclear Engineering and Design, 235, 773-784.

[8] Gu H. Y., Cheng X., Yang Y. H., CFD analysis of thermal-hydraulic behavior of supercritical water in sub-channels. Nuclear Engineering and Design (2008), doi:10.1016/j.nucengdes.2008.08.22.

[9] Cheng X., Kuang B., Yang Y. H., 2007, Numerical analysis of heat transfer in supercritical water cooled flow channels. Nuclear Engineering and Design, 237, 240-252.

[10] Ikeno T., Kajishima T., Analysis of dynamical flow structure in a square arrayed rod bundle. Nuclear Engineering and Design (2008), doi:10.1016/j.nucengdes.2008.07.012.