Analysis of the fluid flow characteristics in subchannels of VVER-1000 reactor's fuel assemblies by CFD method

Dinh Van Thin[1], Phan Le Hoang Sang[2], Luong Van Tho[3]

[1]Department of Nuclear Power Plants, Electric Power University, Hanoi, Vietnam.

[2]Department of Nuclear Physics, University of Science, HCMC, Vietnam.

[3]Department of Physics, Danang University, Vietnam.

Computational Fluid Dynamics (CFD) is a widely used method around the world for complex flow and heat industrial problems. In this paper, the fluid flow parameters were investigated in subchannels of VVER-1000 reactor's fuel assemblies by ANSYS CFX program. Different mesh resolutions and turbulence models were tested to deal with the water flow problems such as velocity distribution and pressure change as well as the hydraulic resistances of the spacer grids. The obtained results are good agreement with the measured values and the published reports from other authors.

Introduction

VVER-1000 is a pressurized water reactor with an open type of hexagonal core geometry that contains 163 fuel assemblies. A fuel assembly consists of 312 fuel rods in triangular arrangements. There are fourteen spacer grids in the fuel assembly [1].

The nuclear power plants generate heat energy, this heat energy is produced from nuclear fission in fuel rods and is transfered to a liquid coolant that flows in the space between the rods. The rod bundle is constructed from parallel fuel rods.

A series of spacer grids spaced axially along the rod bundle provides structural support for the fuel rods. In addition to positioning the fuel rods, the spacer grids affect the hydrodynamics of the pressurized water and heat transfer from the rods. The spacer grids locally reduce the flow area through the rod bundle. This causes flow acceleration and deceleration in regions upstream and downstream of the spacer grid, respectively. The growth of the thermal and hydrodynamic boundary layer along the surfaces of the rods is disrupted by a spacer grid. In addition, the spacer grid increases the turbulence intensity in the flow downstream of the spacer grid. For proper design and operation, an accurate knowledge of the fluid flows and heat transfer properties is required because of high sensitivity of reactor behavior to some operating parameters, such as the eddy regions and coolant mixing temperature. In particular, for reasons of more safety importance in nuclear reactors, specification of the fluid distribution helps us to prevent and predict the possible impacts that relate to fluid characteristic change.

Hydrodynamic safety analysis is an important tool for justifying the safety of nuclear power plants. Typically, this type of analysis is performed by means of system computer codes with one dimensional approximation for modelling real plant systems.

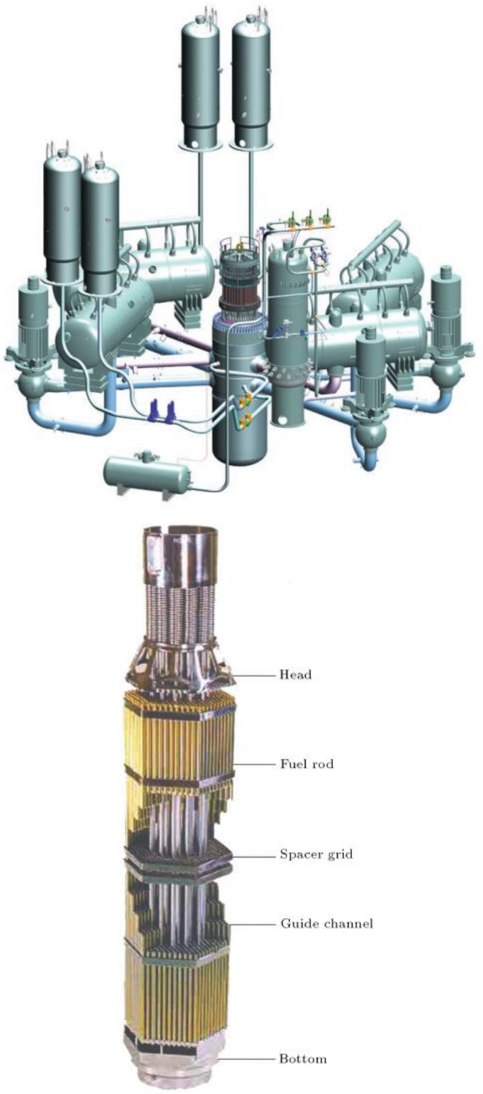

Figure 1. Layout of VVER-1000 reactor and fuel assembly

Kontakt: thin.ninbinh@gmail.com

Beérkezett: 2015. április 15.
Közlésre elfogadva: 2016. január 13.

However, in the nuclear area there are issues for which traditional treatment using one dimensional system codes is considered inadequate for modelling local flow and heat transfer phenomena. There is therefore increasing interest in the application of three dimensional computational fluid dynamics (CFD) codes as a supplement to or in combination with system codes. There are a number of commercial CFD codes as well as special codes for nuclear safety applications available.

When the fluid flows into the rod bundle, they can be classified as either laminar, transition and turbulent depend on Reynolds number [2,3,8].

$$\mathrm{Re} = \frac{\rho v L}{\mu} \qquad (1)$$

Where Re is a Reynolds number; v is velocity [m/s]; L is a characteristic linear dimension [m]; ρ is density [kg/m³]; μ is a dynamic viscosity [Pa s].

The walls are main source of vorticity and turbulence, they affect the velocity profile, pressure drop, separation, recirculation, shear effects and heat transfer.

Subchannel simulations

Numerical model of the subchannel

A central subchannel in a VVER-1000 fuel assembly that is surrounded by fuel rods only was simulated. The 5 mm long model of this subchannel was built with 9.1 mm fuel rod diameter and 12.75 mm pitch [1]. The hydraulic diameter of the subchannel is 10.639 mm. The fuel rods are arranged in triangular arrangement.

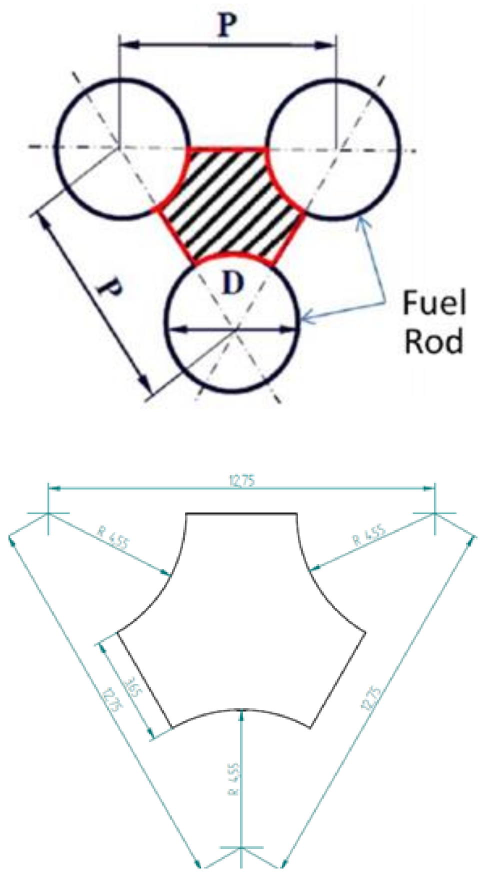

Figure 2. The modelled subchannel and its dimensions

ANSYS ICEM CFD code was used to build the geometrical model and to create different numerical meshes of the subchannel [4]. The geometry was resolved with six different meshes (Figure 3) in order to investigate the mesh resolution to the parameters. So-called extruded mesh was used to discretize the model in space. The core region was meshed with prism elements, hexahedral cells were applied in the near wall region. The cross sectional resolution of the meshes was different (Figure 3), in axial direction the same number of layer was applied. The main characteristic of the meshes can be found in Table 1.

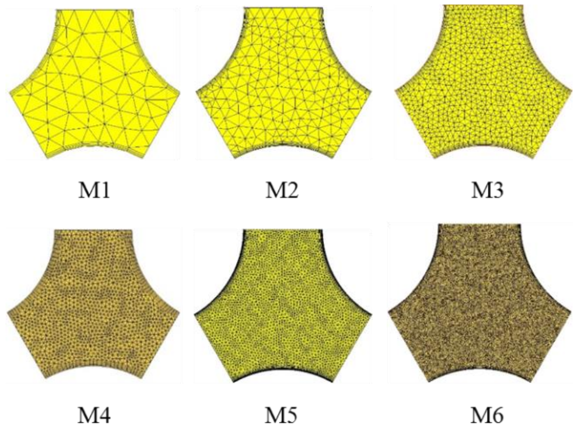

Figure 3. The different mesh resolutions of the subchannel.

Table 1. Characteristics of the different meshes

Mesh	*M1*	*M2*	*M3*	*M4*	*M5*	*M6*
Number of nodes	798	2472	6432	19195	24264	1327746
Number of elements	1400	4296	11858	36530	46162	2441419
y^+	40	20	22	140.9	6	2.8

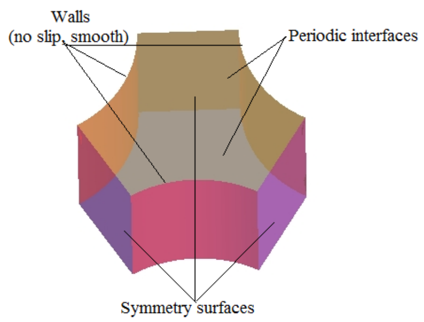

Figure 4. Boundary conditions of the subchannel calculations

For the flow calculations, the solid surfaces of the fuel rod have no slip and smooth boundary condition, and symmetry condition was imposed on the symmetry planes. The lower and upper surfaces were set as periodic interfaces, which helps to simulate the flow in fully developed turbulent flow condition. The selected base Reynolds number for the mesh sensitivity study was 150000. The reference pressure was set to 157 bar. Isotherm simulations were performed, the reference temperature was set to 303 °C. Based on the reference documents the BSL Reynolds Stress turbulence model was chosen for the mesh sensitivity analyses [5, 7]. The y^+ values in Table 1 were calculated with this Reynolds number.

Results of mesh sensitivity analyses

The velocity streamline distributions were affected directly from the different mesh resolutions as show in the Figure 6. The streamlines of M1 and M2 were not symmetric. In contrast, M3, M4, M5 and M6 streamlines were clear symmetry and closer to real physics behavior. Beside, mesh M5 and M6 have too many elements, so we did not use these meshes. We have decided using M3 and M4 for the continuous studies.

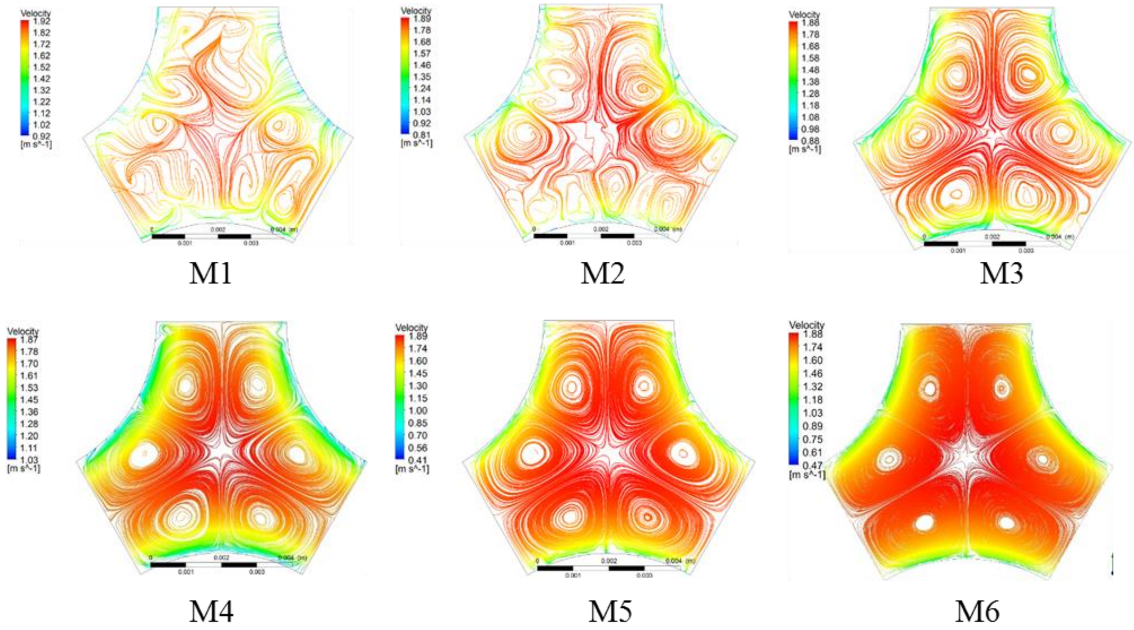

Figure 5. 2D streamlines on the midplane in case of different mesh resolutions

The results of the calculations were compared with each other along "L" line using the presented coordinate system (Figure 6). We experienced that the axial velocity distribution is the same from M3 to M6, the mesh independency has been reached already with M3.

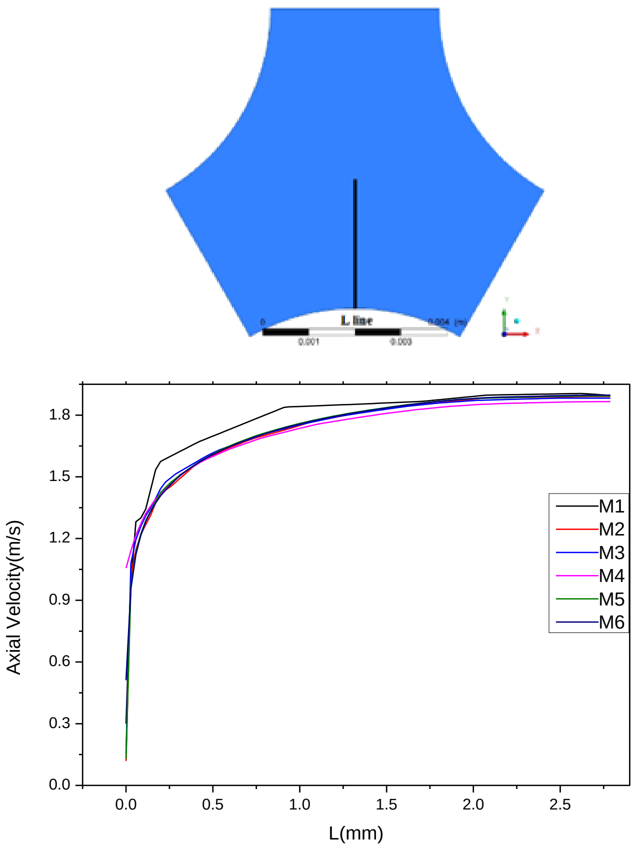

Figure 6. Axial velocity with the different meshing types

Results of turbulence model study

Different turbulence models were tested on M4 to see their effect on the axial velocity distribution and secondary flows. The k-ε, SST, SSG Reynolds Stress and BSL Reynolds Stress turbulence models have been selected for the study. The k-ε model gives good results for many industrial applications but it has limited ability to predict secondary flow characteristics, reattachment or separation, poor performance in geometries with high curvature, flows with sudden changes in the mean strain rate and strong swirling flow [3, 4].

SST model has relative performances of k-ε and k-ω models depend on the region of flow. SST model accounts for transport of turbulent shear stress as prevents over-prediction of eddy viscosity and improves prediction of onset and degree of separation from smooth surfaces [3, 4].

The SSG Reynolds stress and BSL Reynolds stress models belong to the second-order closure models in which transport equations for the individual Reynolds stresses are solved. The SSG Reynolds Stress model is k-ε, the BSL Reynolds Stress is k-ω based turbulence model.

We experienced that the k-ε and SST models did not simulate accurately the secondary flows. The Reynolds Stress models calculates better the real physics phenomena. The axial velocity distribution almost the same in case of all turbulence models (Figure 8).

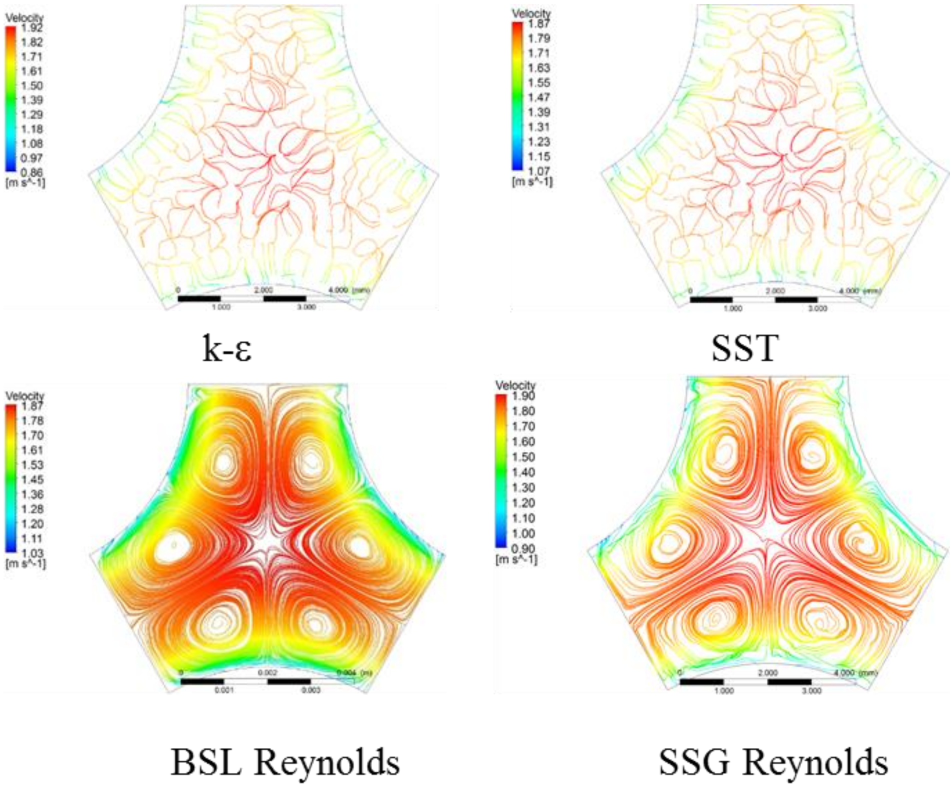

Figure 7. 2D streamlines at on the midplane with the different turbulence models

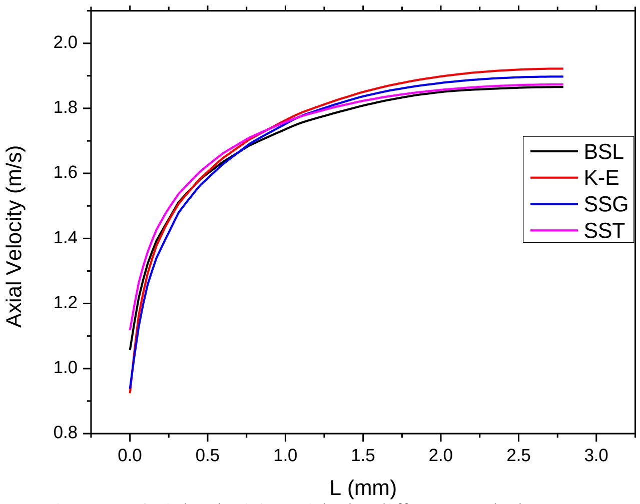

Figure 8. Axial velocities with the different turbulence types

Spacer grid simulations

Numerical model of the spacer grid

To analyse the effect of spacer grid to the coolant flow and to determine the resistance coefficient a new model has been developed. It was enough to investigate a spacer grid part to calculate the appropriate values. The investigated section and the main dimensions can be seen in Figure 9. The total height of the investigated domain is 480 mm, the height of the spacer grid is 30 mm.

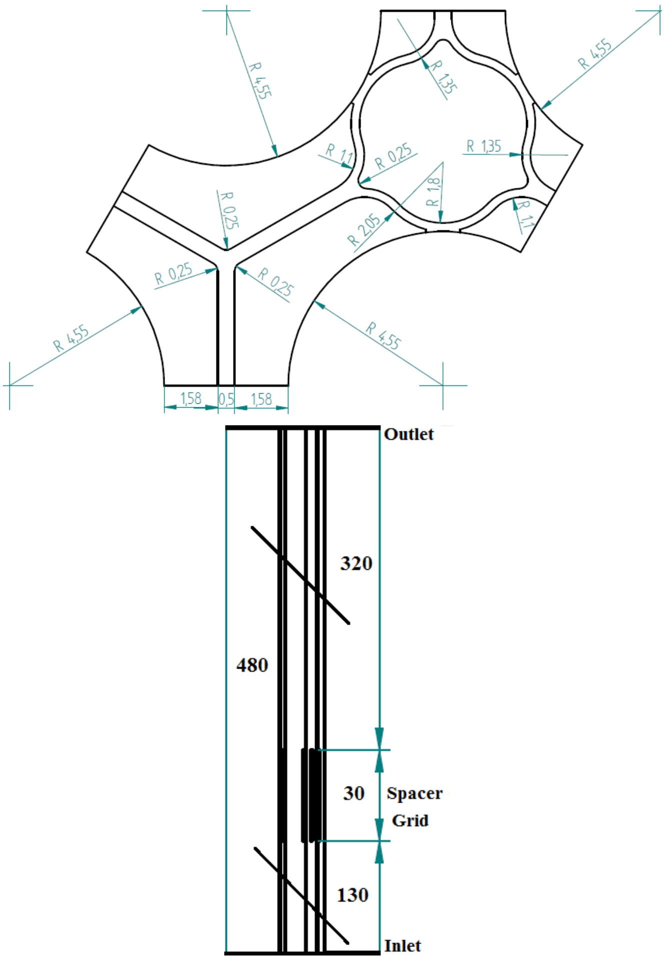

Figure 9. The investigated part of the rod bundle with one spacer grid

The core region was meshed with unstructured prism elements with the same density as the mesh density of the model M3 and M4 from subchannel calculations. The resolution in the near wall regions was increased because of the shape of the geometry. The numerical meshes of the spacer grid model are called G3 and G4, respectively.

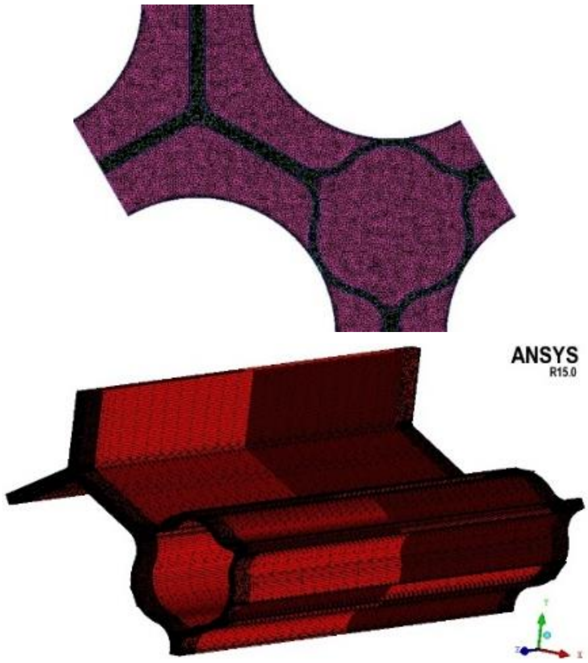

Figure 10. Numerical mesh of the spacer grid (G4)

Table 2. Parameters of G3 and G4 meshes

Mesh	*G3*	*G4*
Number of nodes	1164437	3159557
Number of elements	2851415	8332337

The Reynolds number was 150000 in the first calculation. Mass flow inlet and 0 Pa relative pressure outlet boundary conditions were applied. The side surfaces were connected with translational periodic interface. The calculation was repeated with different Reynolds numbers to determine the resistance coefficient of the spacer grid.

Results of spacer grid calculations

The velocity distribution in different heights is presented in Figure 11. At the inlet (Z=0 mm) the velocity distribution is homogeneous. In Z= 150 mm height the velocity of the coolant increases because of the spacer grid. After the spacer grid (Z=190 mm) the effect of the spacer grid still can be observed, the velocity distribution is still inhomogeneous. At the outlet (Z=480 mm) the effect of the spacer grid is negligible. There are measured values for the resistance coefficient of the spacer grid, which can be used to validate the CFD model [9]. The pressure drop and resistance coefficients were calculated on G3 and G4 meshes with k-ε, SST and BSL Reynolds Stress turbulence models. It can be observed in Figure 8, that the calculated axial velocity is almost the same in case of these turbulence model. We wanted to test the capability of these turbulence models with the two mesh resolutions for this problem. The resistance coefficients were determined in case of different Reynolds numbers.

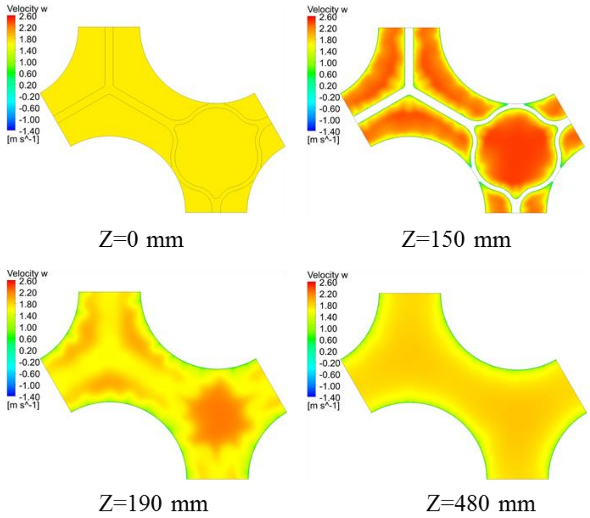

Figure 11. Velocity contour at different z-axis levels (G4)

The calculated and measured resistance coefficients are collected in Table 3 and presented in Figure 12. The results show that the resistance coefficient calculated by k-ε model has the worst agreement with the measured data. In case of SST the agreement is better at smaller Reynolds numbers. The results of BSL Reynolds Stress and k-ε models have the same error in case of higher Reynolds numbers. The highest deviation can be observed in case of G3 mesh with BSL Reynolds Stress model. Maybe the axial resolution of this mesh is not enough to calculate well the pressure drop and the resistance coefficient. The simulations should be repeated with more finer axial mesh resolution. In case of G4, the errors are under 5% - except only two points, which can be accepted. Unfortunately there is not any information about the error of the measurement in the reference document [9], so we could not take it into account.

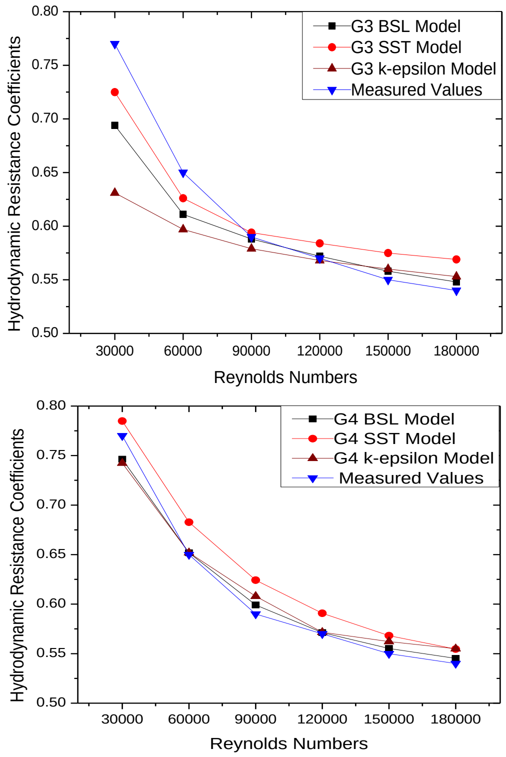

Figure 12. Hydrodynamic resistance coefficient of spacer grid with different Reynolds numbers using G3 and G4.

Table 3. Hydrodynamic resistance coefficient of the spacer grid

Re	30000	60000	90000	120000	150000	180000
ξ measured value	0.770	0.650	0.590	0.570	0.550	0.540
ξ calculated value (G3 BSL)	0.694	0.611	0.588	0.572	0.558	0.548
Error (%)	9.870	6.000	0.339	0.351	1.455	1.482
ξ calculated value (G3 SST)	0.725	0.626	0.594	0.584	0.575	0.569
Error (%)	5.844	3.692	0.678	2.456	4.546	5.370
ξ calculated value (G3 k-ε)	0.631	0.597	0.579	0.568	0.560	0.553
Error (%)	18.052	8.154	1.864	0.351	1.818	2.407
ξ calculated value (G4 BSL)	0.746	0.652	0.599	0.571	0.555	0.545
Error (%)	3.117	0.308	1.525	0.175	0.909	0.926
ξ calculated value (G4 SST)	0.785	0.683	0.624	0.591	0.568	0.555
Error (%)	1.935	5.033	5.801	3.655	3.305	2.712
ξ calculated value (G4 k-ε)	0.742	0.652	0.608	0.572	0.562	0.555
Error (%)	3.586	0.298	3.048	0.286	2.211	2.712

Conclusion

The water flow characteristics in the subchannel and rod bundle sections of VVER-1000 reactors were investigated using the code ANSYS CFX 14.5. Subchannel models were studied with some different mesh density. This sensitivity study showed that the suitable mesh resolution is very important to correctly predict the turbulence quantities in a subchannel. Besides, the suitable mesh was also used to study the turbulence models as k-ε, SST, SSG Reynolds Stress and BSL Reynolds Stress models. Based on these studies, the BSL Reynolds stress model was selected for the further investigations. The second model includes a spacer grid part as well in order to investigate its effect. The spacer grids locally reduce the flow area through the rod bundle. This causes flow acceleration and deceleration in regions upstream and downstream of the spacer grid, respectively. In addition, the spacer grid increases the turbulence intensity in the flow just downstream of the spacer grid. An accurate knowledge of the fluid flows and heat transfer properties is required because of high sensitivity of reactor behavior to some operating parameters, such as the eddy regions and coolant mixing. The fluid distribution helps us to prevent and predict the possible impacts that relate to fluid characteristic change.

In the future, we plan to investigate a full-length fuel bundle model to make the detail data of fluid characteristics and apply their results to a safety analysis and operation of the nuclear power plants in Viet Nam.

ACKOWLEDGMENTS

This paper is the result of training collaboration program between Vietnam and Hungary for culvituring the human resources in nuclear power field. We would like to thank the Ministry of Education and Training (MOET) of Vietnam and the Institute of Nuclear Techniques (NTI) of Budapest University of Technology and Economics (BME), Hungary for offering and sponsoring this training program.

REFERENCES

[1] *IAEA- WWER-1000 Reactor Simulator, Material for Training Courses and Workshops, Second Edition, 2005*

[2] *Jiyuan Tu, Guan Heng Yeoh and Chaoqun Liu: Computational Fluid Dynamics, A Practical Approach, Elsevier, 2008*

[3] *ANSYS Inc, Turbulence modeling.*

[4] *ANSYS, Inc. ANSYS ICEM CFD Help Manual, 2012*

[5] *S. Tóth and A. Aszódi: CFD analysis of flow field in a triangular rod bundle, Elsevier, 2008*

[6] *H. Ganjiani and B. Firoozabadi: Three-Dimensional Simulation of Turbulent Flow in 3-Sub Channels of a VVER-1000 Reactor, Sharif University of Technology, 2010*

[7] *Sándor TÓTH and Dr. Attila ASZÓDI: Detailed analysis of coolant flow in VVER-440 fuel rod bundle, 16th Symposium of AER on VVER Reactor Physics and Reactor Safety, Slovakia, 2006*

[8] *J.H. Ferziger and M. Peric: Computational Methods for Fluid Dynamics, Third edition, Springer, 2002*

[9] *M. Bukov, A. Shishov, D. Posysaev, O. Kudryavtsev: Investigation of hydrodynamic characteristics of fuel rod bundle (TVS-2M) STAR-CD and ANSYS CFX codes, AER Working Group Meeting, 2007*