CFD simulation of Coolant mixing in VVER-1000 RPV

Dang Phuong Nam[1], Dong Van Thanh[1], Ta Van Chuong[2]

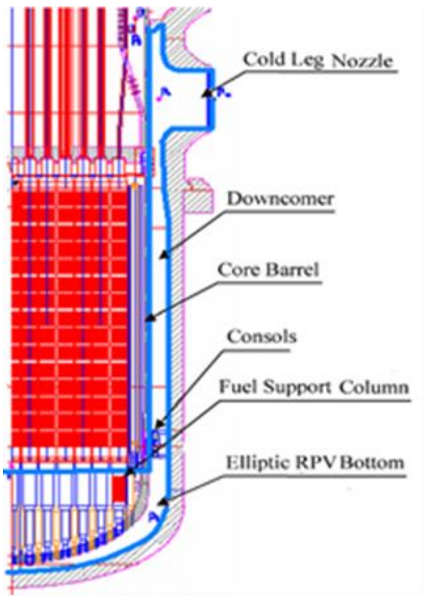
[1]Viet Nam National University, Ha Noi

[2]Ha Noi University of Science and Technology

The VVER-1000 is a common type of the nuclear reactors in the world. Coolant mixing inside the nuclear reactor vessel plays an important role in nuclear safety analysis. Different Computational Fluid Dynamics (CFD) codes can be used to investigate in details the fluid flow and heat transfer inside the reactor pressure vessel. In this study, the results of CFD analysis on coolant mixing calculated with ANSYS CFX 14.5 are presented. The calculated results are validated with measured ones that are presented in Exercise 1 of VVER-1000 Coolant Transient Benchmarks (V1000CT) by OECD/NEA [1, 2, 3].

Introduction and Geometry model of the reactor

The VVER-1000 is a four loop pressurized reactor with hexagonal fuel assembly design and horizontal steam generators. In the reactor pressure vessel (RPV) the coolant enters into the vessel through the cold legs, than flows downward in the downcomer and enters the lower plenum by passing the perforated elliptical bottom plate. After this part the coolant flows through the core bottom plate and enter the core. To perform CFD simulation the geometry model of the reactor was created. The ANSYS ICEM 14.5 was used to generate the geometrical details such as the inlets, the downcomer, the consols and the lower plenum (the elliptical perforated plate and the support plate). The model ends at the core inlet, so the fuel assemblies are not included into the investigated domain. The geometric details of RPV have strong influence on the flow field and on the mixing, so it was important to build the geometry of consols. The basis geometry of the RPV is presented in Fig. 1. The blue line on Fig 1(a) shows the investigated domain.



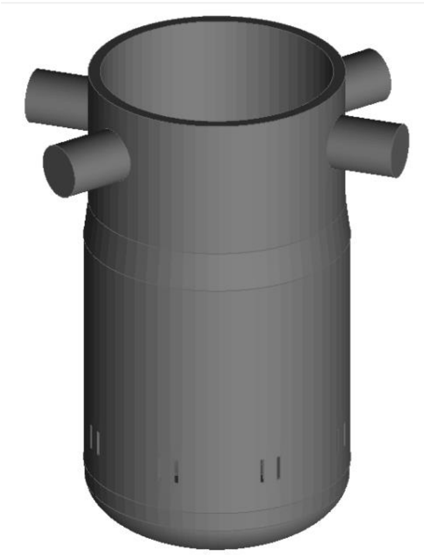


Figure 1. Cross section [1] and 3D model of the VVER-1000 RPV

Grid generation

The grids were created by ANSYS ICEM 14.5. The investigated volume has been discretized with tetrahedral mesh with added prisms layer in the near wall region. Five layers of prisms were generated in order to better predict the flow in the region adjacent to boundaries of the inlet nozzles, the consols and the vessel wall. These mesh types are generally the most flexible when dealing with complex geometries. Three different mesh resolutions were applied: coarse, medium and fine mesh. The resulting grids count about 3 million tetrahedral with 2.6 million prismatic elements, 8 million tetrahedral with 5.9 million prismatic elements, 8.7 million tetrahedral with 4.7 million prismatic elements for the coarse, medium and fine mesh respectively. The coarse mesh is shown in Fig 2.

Kontakt: kiss.bela@reak.bme.hu

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

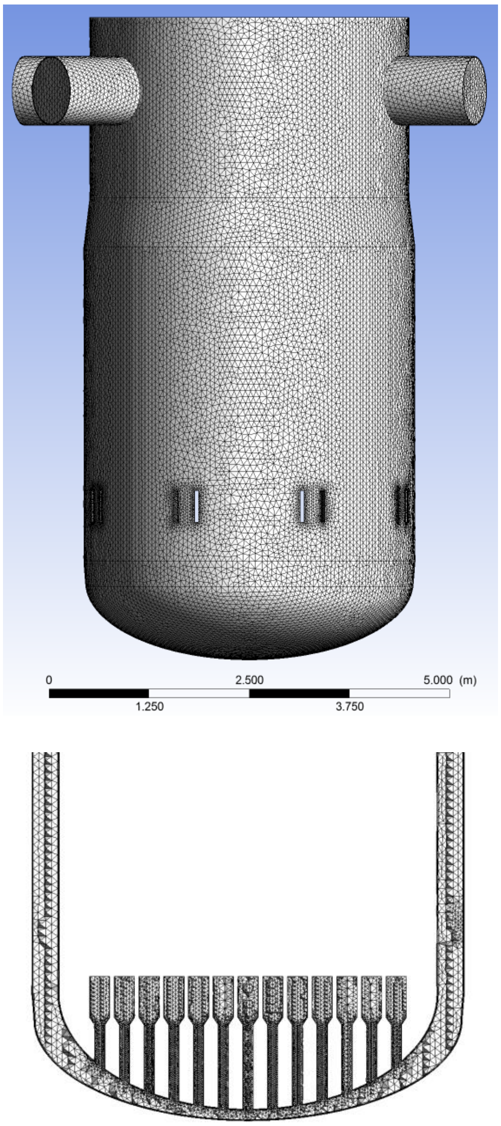


Figure 2. Surface mesh and vertical cut of the volume mesh of the lower part

Boundary conditions

The applied boundary conditions (mass flow rate, temperature) at the inlet nozzles are given in Table 1. These values were set accordingly to the boundary conditions of the final state of the experiment [1, 2, 3]. The walls were modeled using adiabatic conditions. Outlet boundary conditions were set at zero relative pressure. The reference pressure was set to 157 bar.

Table 1: Inlet boundary conditions

Loop	*Mass flow (kg/s)*	*Temperature (°C)*
1	4566	282.2
2	4676	269.9
3	4669	269
4	4819	269.2

Three turbulence models such as SST, k-ε and SSG Reynolds Stress with High resolution scheme were applied to find impact of different turbulence models on results [4]. The convergence criteria was set 10^{-5} for RSM residual.

Results and discussion

Several CFX results with three different turbulence models for each mesh resolutions (coarse, medium and fine mesh) were obtained. The computational time was about several days for a run.

The numbering of the fuel assemblies and the location of the loops are shown in Fig. 3. The average temperature was determined for each fuel assemblies and was compared with measured values.

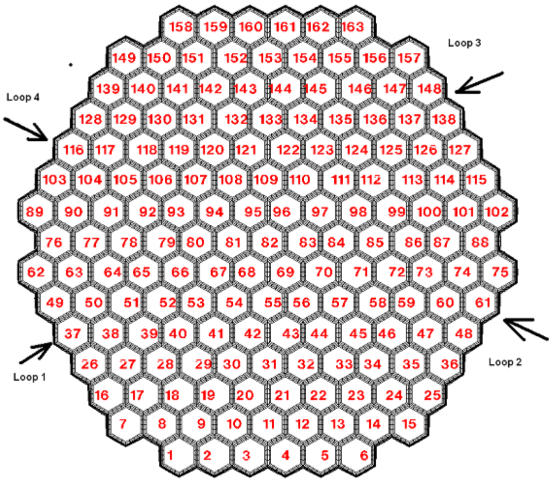


Figure 3. The numbering of the fuel assemblies

SST (Shear Stress Transport) turbulence model

The SST is a widely used and robust two-equation eddy-viscosity turbulence model used in CFD. The SST model was designed to give a highly accurate prediction of the onset and the amount of flow separation under adverse pressure gradients by the inclusion of transport effects into the formulation of the eddy-viscosity. This results in a major improvement in terms of flow separation predictions. The SST model is recommended for high accuracy boundary layer simulations [4].

Fig. 4 shows the calculated temperatures at the inlet core with three different mesh resolutions in comparison with experimental data. The results obtained by SST turbulence model with three different mesh resolutions have good agreement with the experimental data. The temperature difference between calculated and measured values is in the range up to 1-2 K in case of all meshes, which is not very significant. However, the best agreement with experimental data is calculated with the fine mesh.

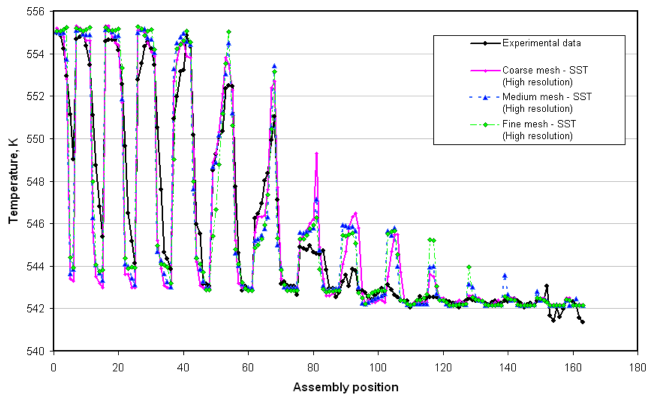


Figure 4. Comparison of calculated results (SST) and experimental data

k-ε turbulence model

The k-ε model focuses on the mechanisms what affect the turbulent kinetic energy. The k-ε model is known to have weaknesses, but the simple structure compared with more advanced models makes its usage attractive. The k-ε model is very popular for industrial applications due to its good convergence rate and relatively low memory requirements. It does not very accurately compute flow fields that exhibit adverse pressure gradients, strong curvature to the flow, or jet flow. It does perform well for external flow problems around complex geometries [4].

Fig. 5 shows the comparison of measured and calculated temperatures at the core inlet with three different meshes. The deviations between calculated and measured temperatures are somewhat larger than in case of SST turbulence model. The agreement between measured and calculated data is worse than in case of SST. The temperature deviations are up to 6 K with some exceptions at some assembly positions.

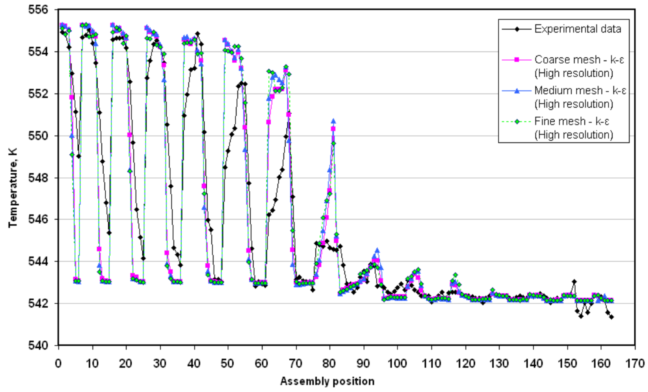


Figure 5. Comparison of calculated results (k-ε) and experimental data

SSG Reynolds Stress model

Two-equation turbulence models (k-ε, SST) offer good predictions of the characteristics and physics of most flows of industrial relevance. In flows where the turbulent transport or non-equilibrium effects are important, the eddy-viscosity assumption is no longer valid and results of eddy-viscosity models might be inaccurate. Reynolds Stress models naturally include the effects of streamline curvature, sudden changes in the strain rate, secondary flows or buoyancy compared to turbulence models using the eddy-viscosity approximation. It develops from partial differential equation for each of six Reynolds' stress term [4].

Fig. 6 shows the CFX SSG Reynolds calculated temperatures at the inlet core with three different mesh resolutions in comparison with experimental data. It can be seen that the core inlet temperature calculated by CFX SSG Reynolds model with the coarse mesh has significant difference from the experimental data. The temperature deviations are up to about 7 K. This is due to a high quality mesh required when using a Reynolds Stress model. However, calculated temperatures with the medium and fine mesh have reasonable agreement with experimental data. For improving the results, transient calculation might be taken. Another possibility for the improvement is refining the mesh or discretize the investigated domain with block-structured hexahedral elements.

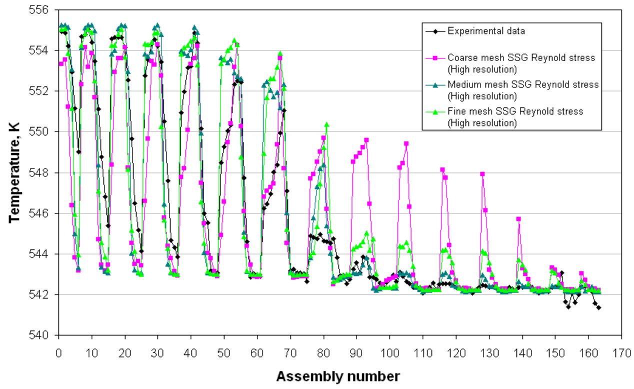


Figure 6. Comparison of calculated results (SSG Re) and experimental data

Upwind vs. High Resolution

The SST simulations have been repeated with first order accuracy to check the impact of different advection scheme. This investigation is based on the result of different benchmark participants. In the reference report almost all of the participants used the upwind (first order) scheme [1]. In case of Upwind the numerical diffusion is more significant, the High Resolution gives more accurate results [4]. Results for Upwind and High resolution scheme with fine mesh are compared against experimental data (Fig. 7).

As shown in the figure, the result of High Resolution scheme is closer to experimental data than the result of upwind scheme.

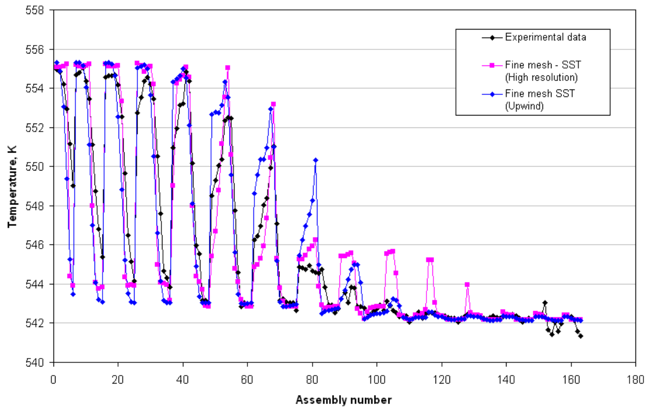


Figure 7. CFX-SST calculated inlet core temperatures with fine mesh for both Upwind scheme and High Resolution scheme in comparison with plant data

Comparison with benchmark results

The comparison of SST results in case of fine mesh with Trio_U results and experimental data are presented in. Fig.8. As can be seen, Trio_U results are closer to the measured values. This good agreement a little bit misleading because the Trio_U code was used to develop and validate the benchmark specifications and to provide support calculations. The difference between Trio_U and ANSYS CFX results are not significant.

Additionally, the results calculated by other benchmark participants are shown in Fig. 9. We can conclude, that our results agree well with the results calculated by other codes.

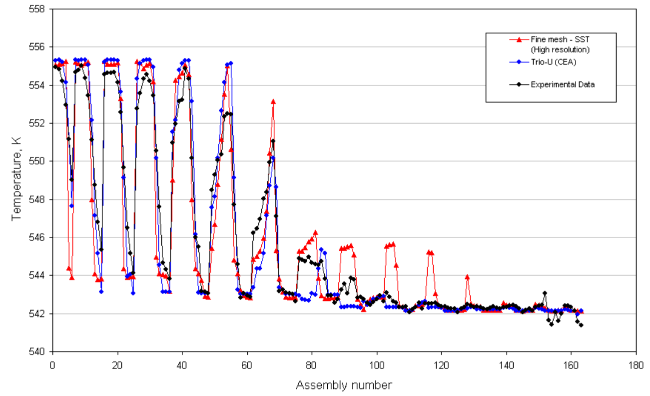


Figure 8. Comparison of calculated results (SST, Trio_U) and experimental data

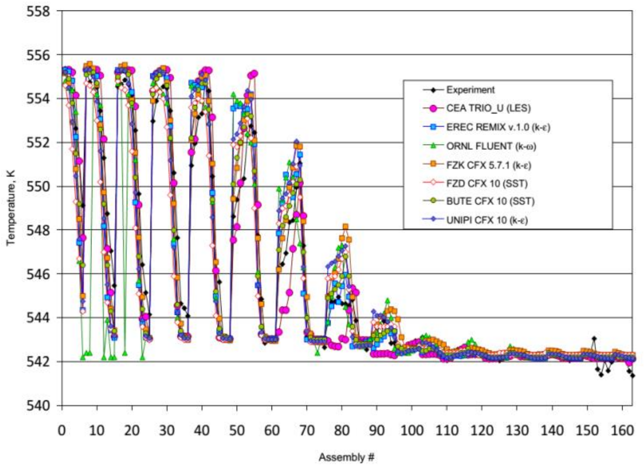


Figure 9. Core inlet temperature results calculated by other codes and experimental data [1]

Temperature and velocity distribution in the RPV

Fig. 10 and Fig. 11 show the calculated temperature distribution at core inlet and on the wall of reactor vessel. The coolant of cold leg 1. goes down in the downcomer in a sector flow. This sector defines the temperature field at the core inlet.

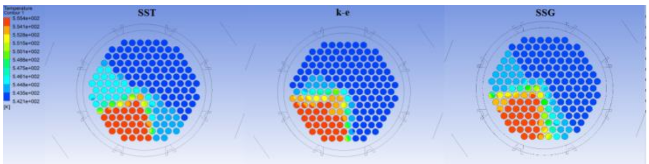


Figure 10. Temperature distribution at core inlet (fine mesh)

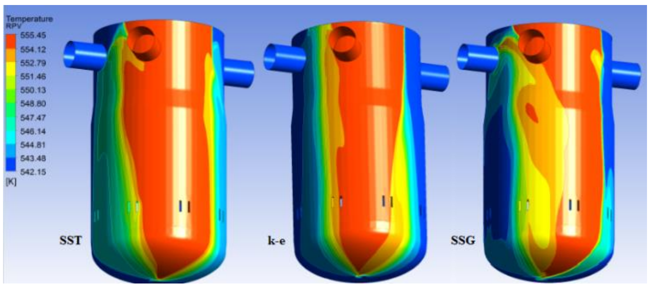


Figure 11. Temperature distribution on the vessel wall

The temperature distributions at the core inlet (Fig 10) show that temperature deviations at the border of the sector are larger than at the center. In case of SST and SSG Reynolds models the borders are wider, than in case of k-ε. As shown in Fig. 11 the flow turns in the downcomer slightly in counter- clockwise direction.

The flow field inside the RPV can be visualized with streamlines, which can be colored by temperature or velocity. In Fig 12(a) the temperature distribution can be seen in the downcomer. The sudden change in the flow direction results that under the inlet nozzles the fluid velocity is lower and some recirculation zones can be observed (Fig 12(b)). The consols have effect on the water flow because they form a barrier, the coolant has to pass round them (Fig 12(c)).

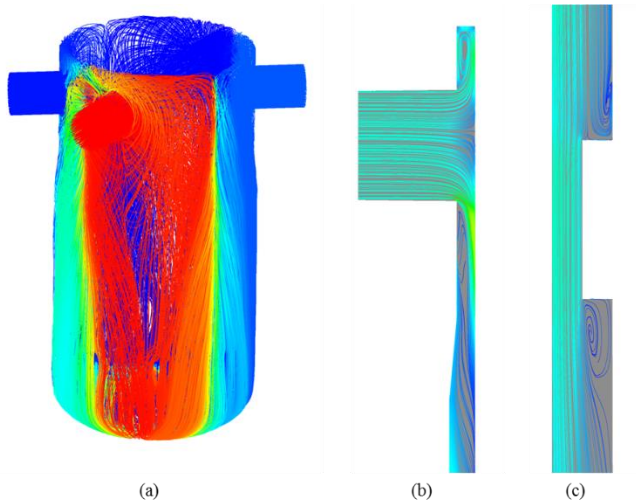


Figure 12. Temperature distribution on streamlines (a), Velocity distribution on streamlines in the vicinity of inlet nozzles (b) and around consols (c)

Conclusion

Three dimension CFD calculations were carried out to investigate the coolant flow inside a VVER-1000 RPV. The results show that it is possible to study the coolant mixing in pressure vessel with ANSYS CFX 14.5 code. Differences can be observed between the calculated and measured temperature distributions. It seems that the calculated temperature field at the core inlet is rotated compared to the measured distribution. It seems that the results estimated by ANSYS CFX similar the results from other codes [1, 2, 3]. The results could be improved with transient calculation or improvement of mesh resolution.

Acknowledgments

Authors would like to thank The Institute of Nuclear Techniques (INT) of Budapest University of Technology and Economics (BME) for the support, specially B. Kiss (BME) for his support and fruitful discussions. The work reported about in this paper was sponsored by Viet Nam Ministry of Education and Training.

REFERENCES

[1] *N.P. Kolev, I. Spasov: VVER-1000 Coolant Transient Benchmark phase 2 (V1000CT-2), 2010, NEA/NSC/DOC(2010)10.*

[2] *B.Ivanov, K.Ivanov, P., et al.: VVER-1000 Coolant Transient Benchmark PHASE 1 (V1000CT-1). 2002, NEA/NSC/DOC(2002)6*

[3] *N.Kolev, N.Petrov, S., et al.: VVER-1000 Coolant Transient Benchmark - Overview and Status of Phase 2., International Conference Nuclear Energy for New Europe, 2005.*

[4] *ANSYS CFX Help, 2012*

[5] *B.Khanbabaei, A.Ghasemizad, H.Farajollahi: CFD-Calculation of Fluid Flow in VVER-1000 Reactors, Journal of Applied Sciences, 2008, 8(5) pp.780-788*

[6] *M.Bottcher: Detailed CFX-5 study of the coolant mixing within the reactor pressure vessel of a VVER-1000 reactor during a non-symmetrical heat-up test, Nuclear Engineering and Design, 2007, vol 283 pp 445-452*

[7] *T.Hohne, G.Grunwald, U.Rohde: Coolant mixing in pressurized water reactors, Proceeding of 8th AER Symposium on VVER Reactor Physics and Reactor Safety, 1998.*

[8] *U.Bieder, G.Fauchet, S., et al.: Simulation of mixing effects in a VVER-1000 reactor, Nuclear Engineering and Design, 2007, vol.237 1718–1728*