THERMAL HOMOGENIZATION OF COOLANT IN DOWNCOMER OF NUCLEAR REACTOR VVER 440

Vladimír Kutiš[1], Jakub Jakubec[1], Gabriel Gálik[1], Juraj Paulech[1], Justín Murín[1], Juraj Hrabovský[1]

[1] Department af Applied Mechanics and Mechatronics
Institute of Automotive Mechatronics
Faculty of Electrical Engineering and Information Technology
Slovak University of Technology in Bratislava
Ilovičova 3, 81219 Bratislava
E-mail: vladimir.kutis@stuba.sk

Received 07 May 2015; accepted 15 May 2015

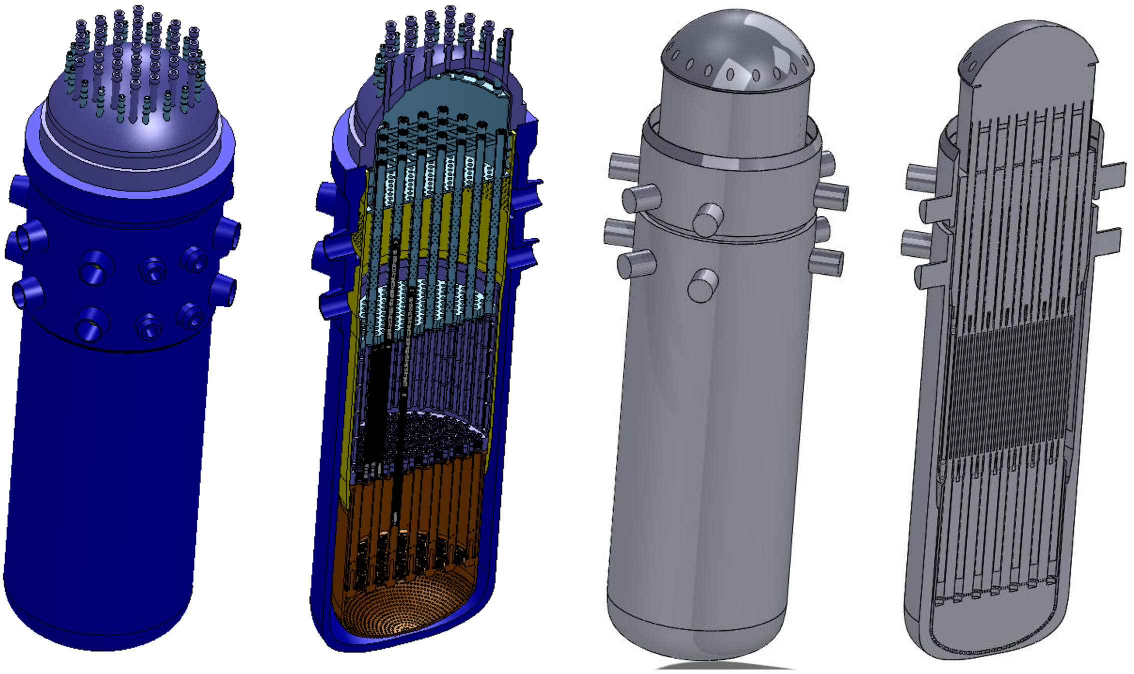
1. Introduction

During the phase of considering safety aspects of the nuclear power plant, one of the fundamental criterions rests in determination of the thermal-hydraulic conditions in the active zone of the nuclear reactor. Thermal-hydraulic performance of active zone is strongly affected not only by distribution of power in active zone but also by coolant input parameters like temperature and mass flow at inlet nozzles of fuel assemblies. These parameters depend on homogenization of coolant in downcomer and in bottom of shaft core barrel. In reactor VVER440, there are 6 inlet nozzles for cold coolant water at the reactor pressure vessel. In nominal operation of reactor the difference between temperatures of coolant at these inlet nozzles should occur and it could be caused by several reasons. The most common reasons are: different length and segmentation of steam piping, shut down of the one of steam generators (SG), unequal area of heat transfer surfaces of SG [1].

The paper deals with CFD modelling of coolant mixing and homogenizing of temperature in the mixing part of reactor vessel in respect to the temperature difference at inlet nozzles of the reactor vessel. The inlet nozzle of fuel assembly is set up as output region of the simulation model [2]. In CFD model, the flow in protective tubes for control rods is also considered. The influence of different turbulent models, single and double precision of numerical computation and number of finite volumes is investigated.

2. Geometry model of nuclear reactor VVER 440

To performed CFD simulation of coolant homogenization in downcomer of nuclear reactor, 3D CAD geometry of reactor VVER 440 was created. Nevertheless that in this paper only coolant in downcomer and bottom of shaft core barrel is investigated, whole reactor with all internal components was modelled. The reason is that at the end of our VEGA project, we should be able to simulated flow of coolant in the whole reactor. All geometry components of nuclear reactor were considered without simplifications and final model of whole reactor with all internal components is shown in Fig. 1 a) and b). This geometry model can be used not only in the process of creation of coolant - which represents the negative volume of reactor, but this model can also be used in structural (or thermal) analysis of pressure reactor vessel. Next step is simplification of all internal components and creation of negative volume, which as was mentioned above, represents volume of coolant - Fig. 1 c) and d).



a) *b)* *c)* *d)*

Fig.1: *Geometry model of nuclear reactor (a) and b)) and model of coolant in reactor (c)and d)).*

3. Discretization of coolant volume

After creation of volume of coolant, the process of discretization has to be performed. Specialized mesh tool ANSYS ICEM CFD [3] was used to create structured hexahedral and tetragonal mesh in downcomer and bottom of shaft core barrel.

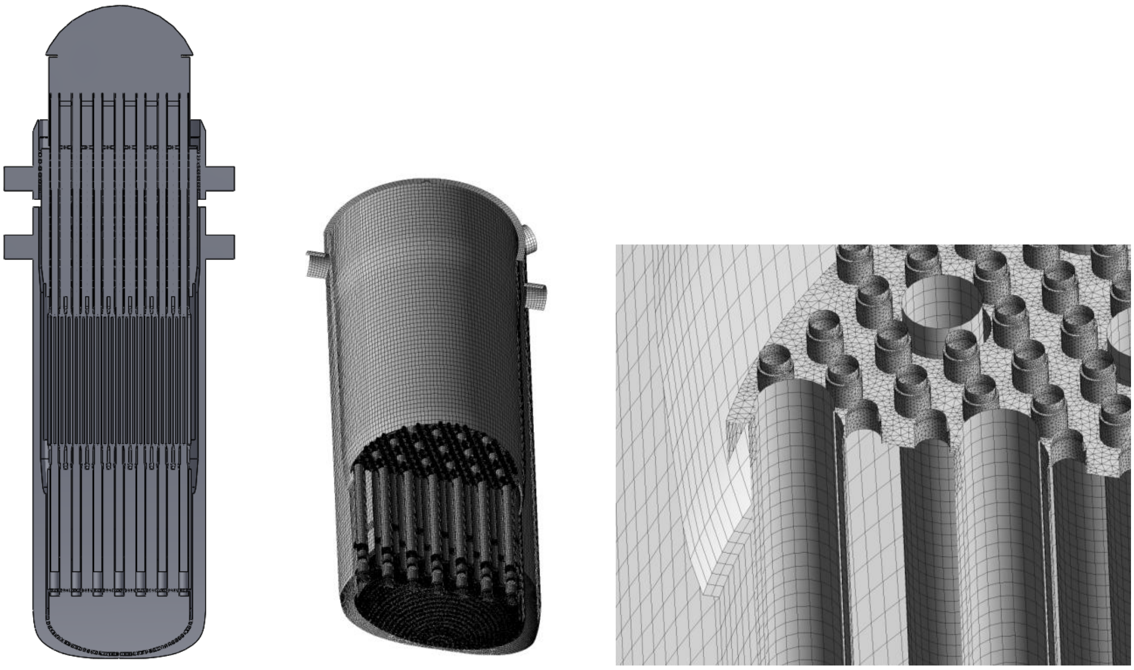


Fig.2: *Discretization of coolant in downcomer and bottom of shaft core barrel.*

In Fig. 2 we can see the details of the discretized model. Because one of our goal is to investigate the influence of mesh density on obtained results, we created two discredized model:

- Mesh 1 - contains approximately 24 millions of elements
- Mesh 2 - contains approximately 31.5 millions of elements

4. CFD simulations and obtained results

The simulation was calculated as steady-state flow of coolant in ANSYS CFX [3] software. The temperatures at six inlet nozzles of the reactor pressure vessel were set according to Table 1 and Fig. 3. Velocity of coolant was set to value 9.2 m/s for all six inlets and this value was derived from total mass flow of coolant through the nuclear reactor pressure vessel. Reference pressure, which represents the pressure at Output, was set to 12.25 MPa. Material properties of coolant (water) were set according to material model in material library IAPWS IF97.

Tab. 1. *Temperature at Inlet nozzles of reactor pressure vessel.*

Simulation	Inlet nozzles					
	In 1	In 2	In 3	In 4	In 5	In 6
Temperature [°C]	268	267	267	267	267	267

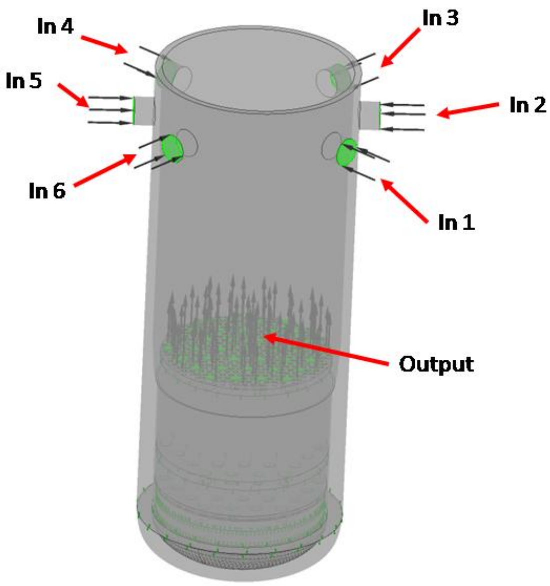


Fig.3: *Boundary conditions for steady-state analysis, six inlets and one output.*

Investigated turbulent models were BSL, k-omega and SST [4]. In SST turbulent model we considered computation with single and double precision. The influence of mesh density (Mesh 1 and Mesh 2) was also investigated using SST turbulent model.

Fig. 4 and Fig. 5 show the distribution of coolant temperature in downcomer and at the output of investigated region, which also represents the inlet nozzles into the fuel assemblies and inlet nozzles into control rods. As we can see from both figures, there is a slight difference in the temperature distribution in downcomer and also at the output region. As we can also see form Fig. 4, hotter stream of coolant from inlet nozzle In 1 is pushed into space between inlet nozzles In 1 and In 6 and the result is, that the temperature distribution at the output is more unsymmetrical - specially for SST turbulent model - see Fig. 5.

Also average output temperatures at the inlet nozzles into fuel assemblies PK1 and PK2 and at the inlet nozzles into control rods (see Fig. 5) are slightly different - Tab. 2.

Fig. 6 shows the distribution of coolant velocity at the bottom of shaft core barrel in plane, which is defined by inlet nozzle In 1. There is visible difference in region, where vertex is developed and this difference is caused by different formulations of individual turbulent models.

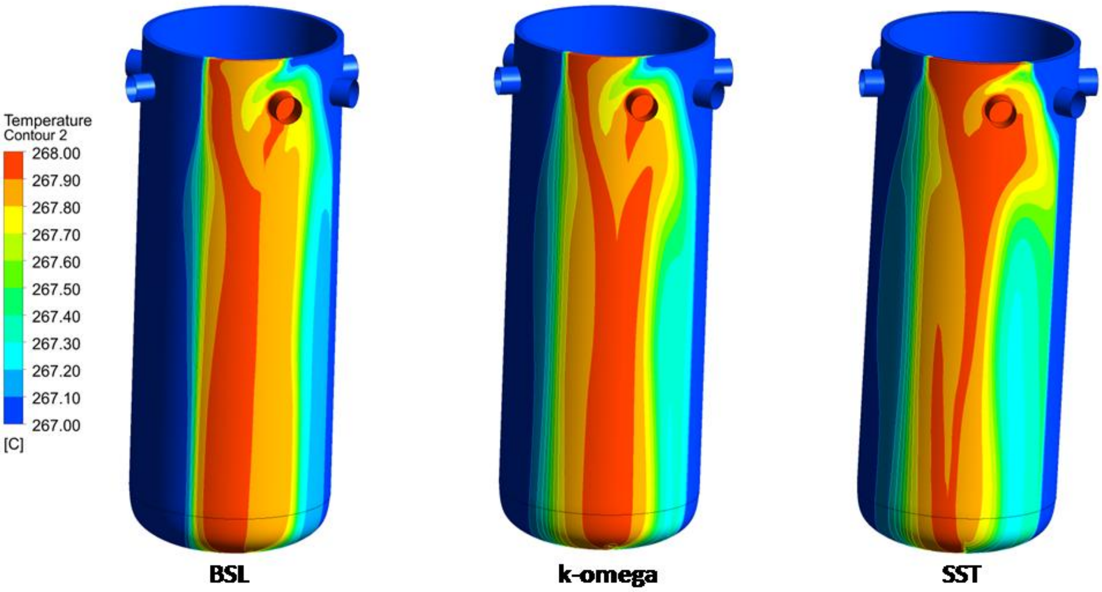


Fig.4: *Distribution of coolant temperature in downcomer for all three investigated turbulent models.*

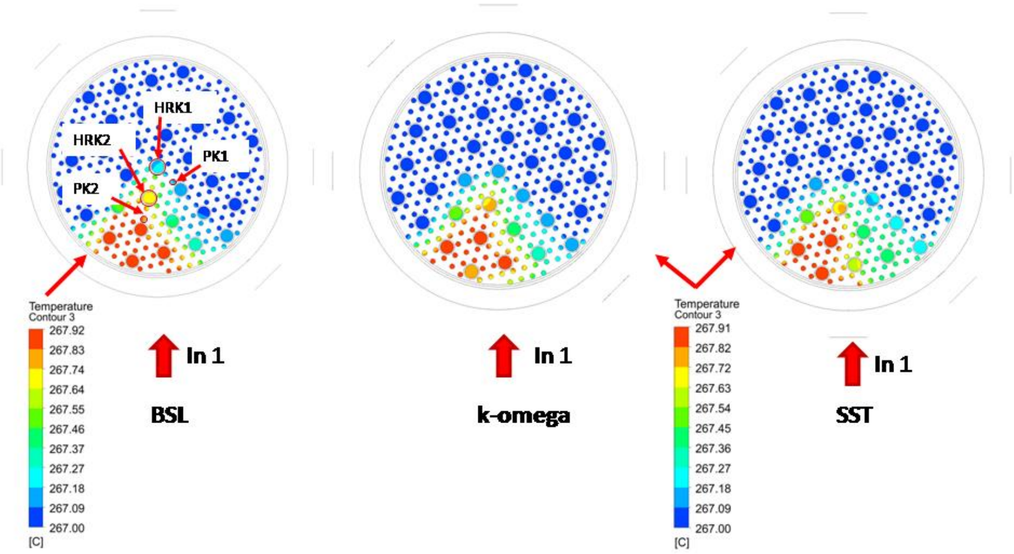


Fig.5: *Distribution of coolant temperature at the output (inlet nozzles of the fuel assemblies) for all three investigated turbulent models.*

Tab. 2. *The influence of different turbulent models on output temperature.*

Turbulent model	Average temperature at location [°C]			
	PK1	PK2	HRK1	HRK2
BSL	267,176	267,804	267,191	267,720
k-omega	267,131	267,865	267,136	267,733
SST	267,134	267,871	267,066	267,722

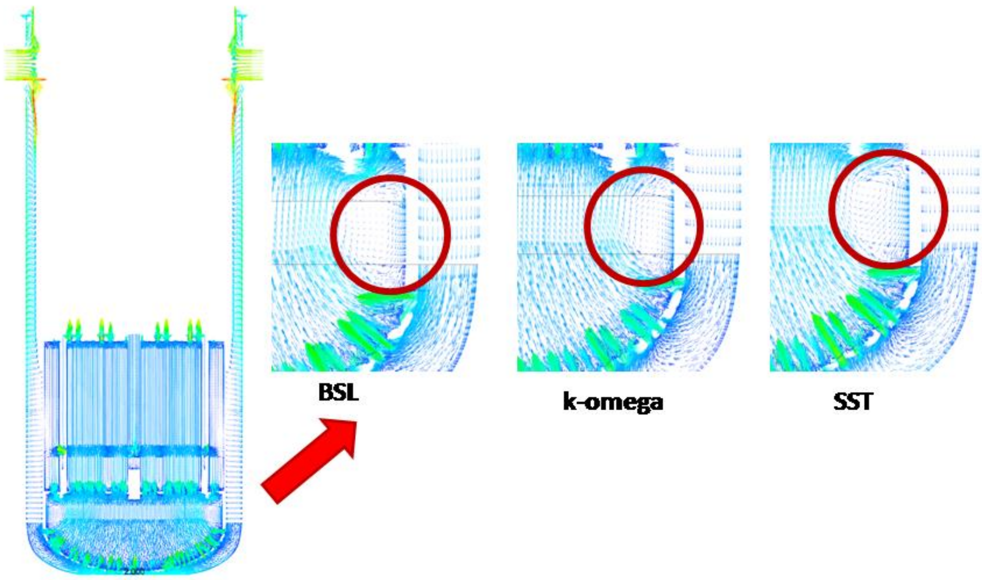


Fig.6: *Distribution of coolant velocity at the bottom of shaft core barrel.*

5. Conclusions

The presented paper dealt with modelling of thermal-hydraulic conditions in the nuclear reactor VVER-440 pressure vessel. Area of interest was the downcomer and the bottom part of reactor vessel where the coolant from six inlets is mixing together. The goal was to investigate the influence of different turbulent models, single and double precision of numerical computation and number of finite volumes on simulation of coolant temperature homogenization at the input nozzles of the fuel assemblies. Inlet temperature and the generated power in the fuel assemblies influence the outer temperature which is one of the limiting parameters in operation of the nuclear reactor. This is the reason why it is necessary to determine temperature distribution in the individual fuel assembly input nozzles.

Acknowledgement

This work was financially supported by grant of Science and Technology Assistance Agency no. APVV-0246-12 and Scientific Grant Agency of the Ministry of Education of Slovak Republic and the Slovak Academy of Sciences No. VEGA No. 1/0228/14 and VEGA No. 1/0453/15.

Authors are also grateful to the HPC Center at the Slovak University of Technology in Bratislava, which is a part of the Slovak Infrastructure of High Performance Computing (SIVVP project, ITMS code 26230120002, funded by the European Regional Development Funds), for the computational time and resources made available.

References:

[1] J. Zdražil, L. Benc, J. Zigmund: Vplyv prevádzkových parametrov niektorých zariadení JE s VVER-440 na stacionárne tepelne-hydraulické charakteristiky aktívnej zóny. In: Fyzikální a tepelné poměry v aktivních zónach jaderných reaktoru typu VVER-440 z hlediska bezpečnosti provozu. Ústrední informační středisko pro jaderní program, Praha. (1980)

[2] V. Kutiš, E. Mojto, J. Paulech: Modelovanie premiešavania chladiva pred vstupom do palivových kaziet JR VVER-440. In: Elosys 2011, Trenčín, (2011).

[3] ANSYS CFX. Help manual, (2015).

[4] D.C. Wilcox: Turbulence Modeling for CFD, DCW Industries, Inc, (1993).