THE INFLUENCE OF ORIFICE DIAMETER ON COOLANT MASS FLOW THROUGH FUEL ASSEMBLY OF NUCLEAR REACTOR VVER 440

Vladimír Kutiš[1], Jakub Jakubec[1], Juraj Paulech[1], Gabriel Gálik[1], Tibor Sedlár[1]

[1] *Department of Applied Mechanics and Mechatronics, Institute of Automotive Mechatronics*
Faculty of Electrical Engineering and Information Technology
Slovak University of Technology in Bratislava
Ilkovičova 3, 812 19 Bratislava
E-mail: vladimir.kutis@stuba.sk

Received 02 May 2016; accepted 16 May 2016

1. Introduction

During the phase of considering safety aspects of the nuclear power plant, one of the fundamental criterions rests in determination of the thermohydraulic conditions in the active zone of the nuclear reactor [1]. In reactor VVER440, there are 6 inlet nozzles for cold coolant water at the reactor pressure vessel (RPV). The paper deals with the influence of nuclear reactor components design on coolant mass flow and velocity distribution during nominal project conditions and with the influence of orifice diameter change on coolant mass flow distribution. The nominal project conditions are defined as isothermal and equal mass flow conditions of coolant in all 6 RPV inlet nozzles. The inlet nozzles of fuel assemblies and control rods are set up as output region of the simulation model.

2. Geometry model of reactor and discretization of coolant in downcomer

To performed CFD simulation [2, 3, 4], the geometry of reactor had to be created. Because only downcomer of the nuclear reactor VVER 440 is investigated by CFD, only three geometry parts were created, namely reactor pressure vessel (RPV), reactor shaft core barrel and bottom of reactor shaft core barrel - see Fig. 1.

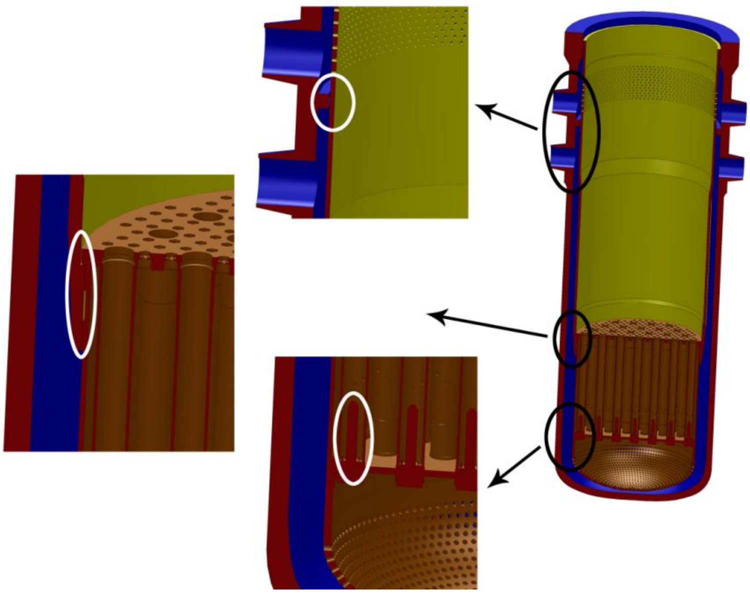

Fig.1: *Geometry model of RPV, reactor shaft core barrel and bottom of reactor shaft core barrel with details.*

All components were created with all design details. For CFD simulations, some of the design details can be simplified - e.g. in reactor shaft core barrel there was simplified outer shape, where surveillance channels were removed. Fig. 2 shows negative volume of geometry model, which represents volume of coolant in downcomer of reactor.

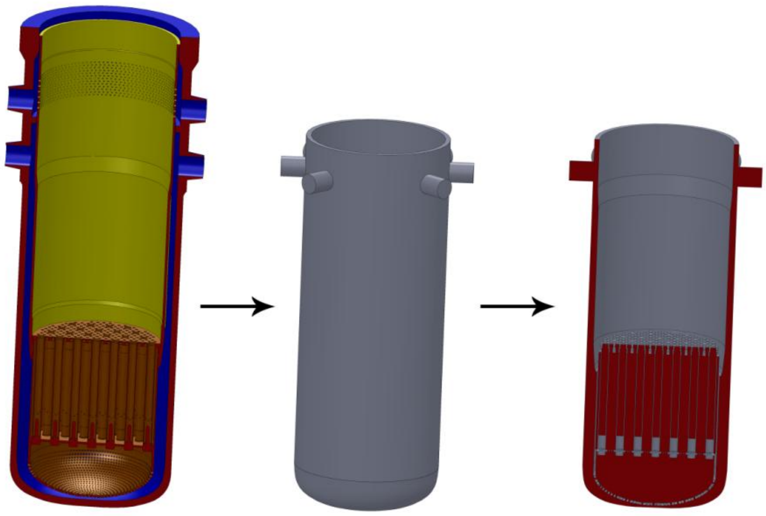

Fig.2: *Negative volume of reactor - volume of coolant in downcomer.*

Specialized mesh tool ANSYS ICEM CFD was used to create structured hexahedral and unstructured tetrahedral mesh - Fig. 3.

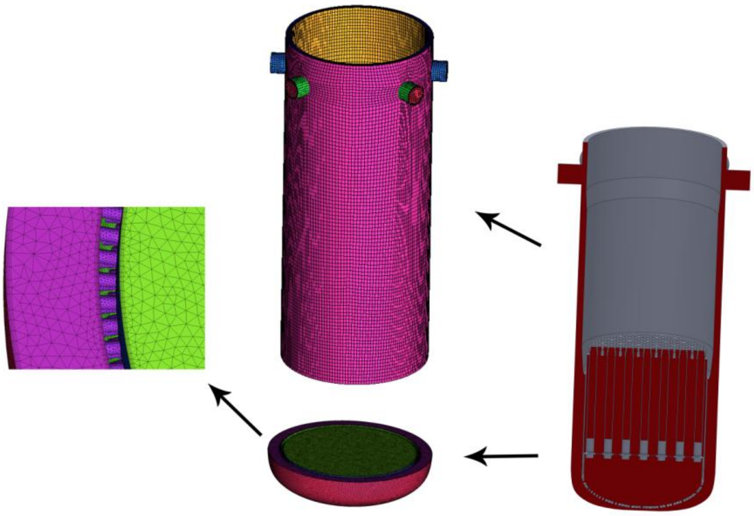

Fig.3: *Discretized volume of coolant in downcomer.*

In common, hexahedral mesh in comparison with tetrahedral mesh provides numerically more accurate results and the number of elements is significantly smaller. Total number of elements was approximately 24 million of elements and 26 million of nodes. Fig. 4 shows discretized one outlet of investigated region - orifice.

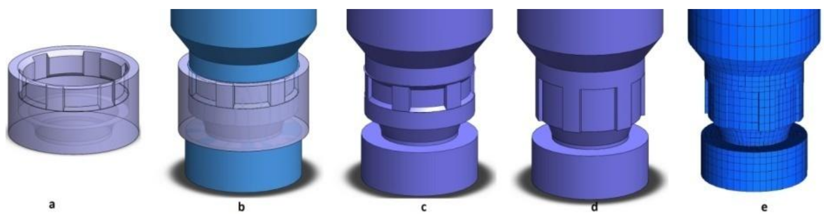

Fig.4: *Discretized volume of coolant in orifice.*

3. Boundary conditions, CFD simulations and results

In presented CFD analyses of coolant flow in downcomer of nuclear reactor VVER 440, nominal project conditions are considered. They can be described as follows:

- coolant mass flow through nuclear reactor: 33 383 t/hour
- coolant mass flow through each of 6 inlet nozzles are equal with value: 1545.5 kg/s
- coolant temperature in all 6 inlet nozzles: 268.0 C
- coolant bypass of core: 6.3%
- coolant output pressure: 12.25 MPa

All boundary conditions of CFD model were set up according the nominal project conditions. The simulation was performed by CFD code ANSYS CFX. The goal of the CFD analyses is to determine the distribution of coolant velocity in downcomer, the distribution of coolant mass flow at the individual inlet fuel assembly (FA) nozzles and to investigate the influence of orifice diameter change on mass flow distribution through individual orifices, when boundary conditions are set up according nominal project conditions.

Distribution of coolant velocity in downcomer of nuclear reactor VVER 440 in two different vertical planes is shown in Fig. 5.

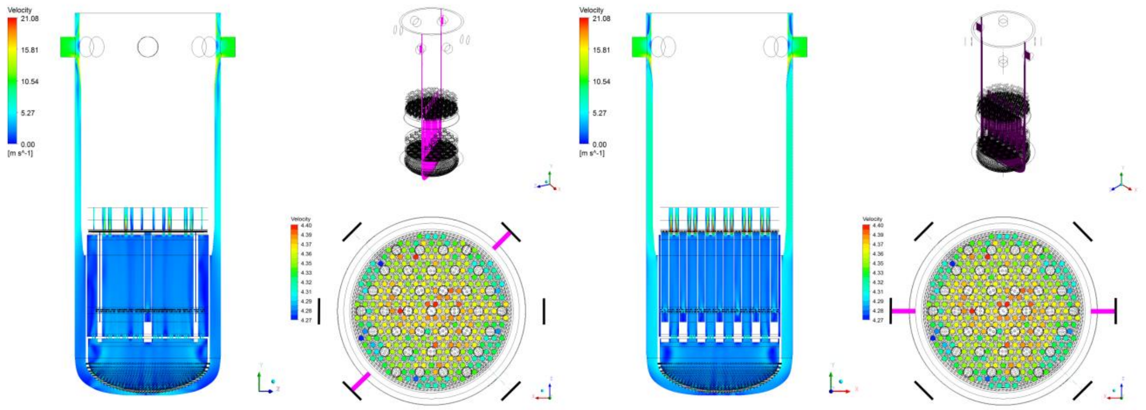

Fig.5: *Distribution of coolant velocity in downcomer in two different vertical planes.*

As we can see from this figure, the coolant velocity can reach value up to 21 m/s. This high velocity of coolant is located near the outlet part of investigated region, where the orifice is located. Coolant flows through the 312 orifices to the individual fuel assemblies and 37 protecting tubes. Nominal diameter of orifice is 50 mm but in real conditions this diameter can be reduced by sediments, which can even increases the velocity in this region.

Fig. 6 shows the distribution of coolant mass flow at the outlet of investigated region, which also represents inlet into the fuel assemblies nozzles. There are shown only 312 orifices of fuel assemblies', but in CFD calculation the flow through protecting tubes for control rods were considered. Fig. 6a shows the distribution of coolant mass flow in individual orifice or fuel assembly inlet nozzles. In this picture we can see that mass flow of coolant is in range from 23.60 kg/s to 24.31 kg/s. This coolant mass flow difference has value 0.71 kg/s and it represents 2.92% from maximal coolant mass flow value. Fig. 6b shows average coolant mass flow in individual orifices when 1/6 symmetry of active zone is considered. The difference between actual coolant mass flow in individual orifice and average value of correspondent 6 orifices is shown in Fig. 6c. In Fig. 6d, there is shown maximal difference of correspondent orifices expressed in percentage. As we can see from these last two figures, the difference between actual and average value of correspondent 6 orifices is in range -0.21 kg/s to 0.22 kg/s and their maximal percentage difference is in range from 0.36 % to 1.71 %. But most of coolant mass flow percentage difference in considering 1/6 symmetry is below 1 %.

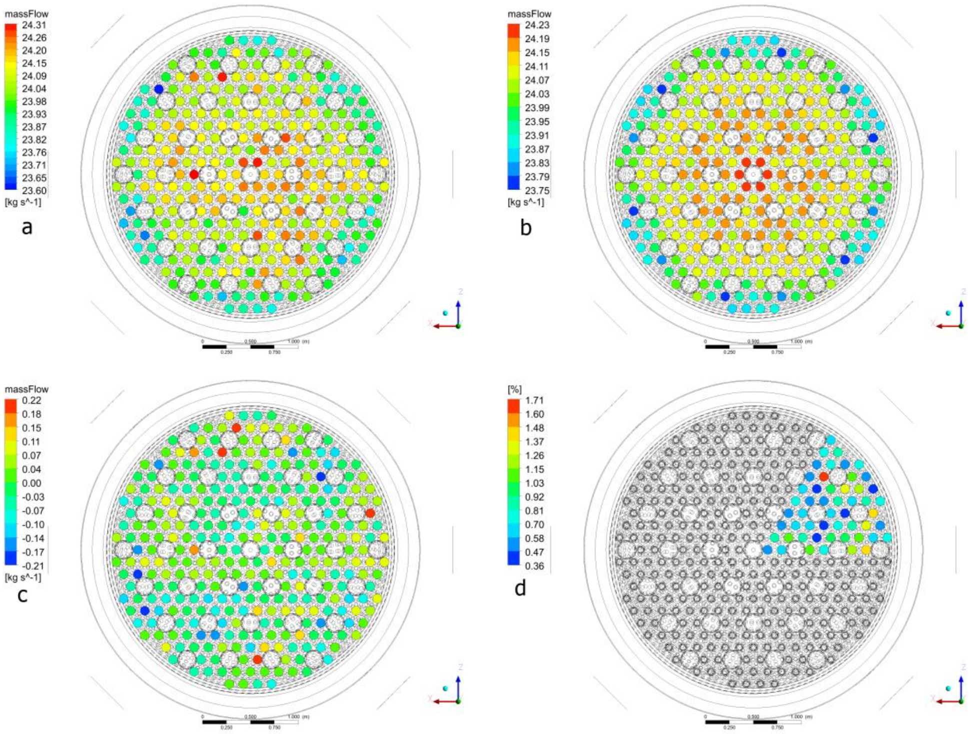

Fig.6: *Distribution of coolant mass flow at the outlet of investigated region.*

The influence of orifice diameter change on mass flow distribution through individual orifices was investigated in three different orifices, which also represents inlet into three different FA (Fig. 7 left):

- FA 174 - FA in the central area next to HRK (safety rod)
- FA 154 - FA in the central area between HRK
- FA 167 - FA in peripheral area next to HRK

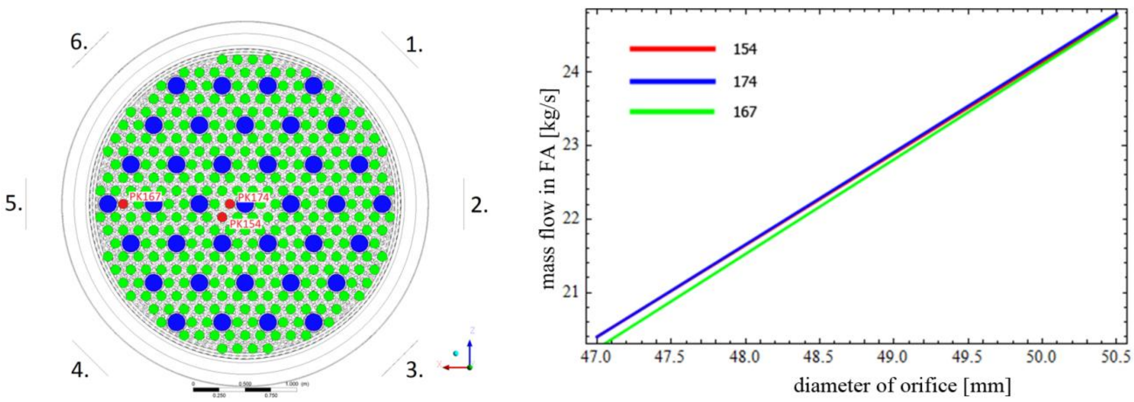

Fig.7: *Dependence of mass flow on diameter of selected orifices.*

Orifice diameter change was considered in range from 47 mm to 50.5 mm with step 1 mm. Each diameter change represents new CFD analysis with the same boundary conditions as was presented above.

Based on performed CFD analyses, it can be stated that:

- relationship between the coolant mass flow through investigated orifice and the diameter of the orifice is practically linear
- coolant mass flow is only minimally affected by locations of investigated orifice

Linear relationship between the coolant mass flow and the orifice diameter for all three investigated orifices is shown in Fig.7 right.

4. Conclusion

The presented paper dealt with modeling of thermohydraulic conditions in downcomer of nuclear reactor VVER 440. Boundary conditions were considered as nominal project conditions. Area of interest was the fuel assemblies' inlet region where distribution of mass flow and velocity of coolant were investigated. The influence of orifice diameter change on mass flow distribution through individual orifices was also investigated. This CFD analysis can be considered as introductory CFD analysis of real conditions in fuel assembly inlet nozzles. In our next research we will focused on real conditions in reactor pressure vessel inlet nozzles, but also on influence of generated heat in individual fuel assemblies on coolant mass flow distribution in individual orifices.

Acknowledgement

This work was supported by the Slovak Research and Development Agency under the contract No. APVV-0246-12 and APVV-14-0613, by Grant Agency VEGA, grant No. 1/0228/14 and 1/0453/15. Authors are also grateful to the HPC Centre 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] Todreas, N. E., Kazimi, M. S.: Nuclear Systems Volume I: Thermal Hydraulic Fundamentals. CRC Press; 2nd edition, ISBN 1439808872, (2011).

[2] Versteeg, H., Malalasekera, W.: An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Prentice Hall; 2nd edition, ISBN 0131274988 (2007).

[3] Hirsch, Ch.: Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics, Butterworth-Heinemann; 2nd edition, ISBN 9780750665940 (2007).

[4] Wilcox, D. C.: Turbulence Modeling for CFD. D C W Industries, 3rd edition, ISBN 1928729088 (2006).