Journal of MECHANICAL ENGINEERING
– Strojnícky časopis

VOLUME 66, NO 2, 2016

Print ISSN 0039-2472
On-line ISSN 2450-5471

pp. 55 – 62
DOI:10.1515/scjme-2016-0018

CFD ANALYSIS OF DOWNCOMER OF NUCLEAR REACTOR VVER 440

KUTIŠ Vladimír[1], JAKUBEC Jakub[1], PAULECH Juraj[1], GÁLIK Gálik[1], SEDLÁR Tibor[1]

[1] Slovak University of Technology in Bratislava, Faculty of Electrical Engineering and Information Technology, Institute of Automotive Mechatronics, Department of Applied Mechanics and Mechatronics, Ilkovičova 3, 812 19 Bratislava, Slovakia

Abstract: The paper is focused on CFD analyses of the coolant flow in the nuclear reactor VVER 440. The goal of the analyses is to investigate the influence of the orifice diameter on the mass flow through individual fuel assemblies in the reactor core. The diameter of orifice can be changed during the operation of a nuclear power plant. Considered boundary conditions in the investigated region of the coolant are based on nominal coolant flow conditions in the nuclear reactor VVER 440.

KEYWORDS: CFD simulation, ANSYS CFX, thermal-hydraulics, nuclear reactor, VVER 440

1 Introduction

In the nuclear reactor safety, thermo-hydraulics is a very important subject [1]. Thermo-hydraulics as a multiphysical domain has an influence not only on thermal conditions of a nuclear fuel [2], but also influences on the distribution of the neutron flux in the reactor core, the thermal and pressure loading of the reactor pressure vessel (RPV) and sets up the critical value of the heat flux. Many years thermo-hydraulics of nuclear reactors has been investigated only by specialized system codes, like RELAP and ATHLET. In the last decade, computational fluid dynamics - CFD [3] emerged as a very useful alternative tool to analyze thermo-hydraulics, where a real 3D geometry can be considered. Modern computer simulation techniques, like CFD [3, 4] or FEM [5], can be very useful in a detail study of such processes, because after verification and validation processes [6] of the CFD model, you can relatively easily change boundary and initial conditions, or other input parameters of the model like geometry parameters.

In our research, we focused on the modelling and simulation of thermo-hydraulic processes in the downcomer of the nuclear reactor VVER 440, where the distribution of the velocity field in the coolant is investigated. The paper also deals with the influence of a nuclear reactor components design on the coolant mass flow distribution during nominal project conditions and with the influence of the diameter change of throttling orifices, which are installed in the support plate of the core barrel, on the coolant mass flow distribution through individual fuel assemblies. The nominal project conditions are defined as isothermal and equal mass flow conditions of the coolant in all 6 RPV inlet nozzles. The inlet nozzles of the fuel assemblies (FA) and control rods (HRK) are set up as a output region of the simulation model. All CFD analyses were performed by software ANSYS CFX [7], which computes basic thermo-hydraulic differential equations by Finite Volume Method [3].

2 Geometry model of reactor

To perform the thermo-hydraulic analysis of the reactor VVER440 downcomer, it is necessary to create a 3D geometry model of the coolant in the investigated region. Creating of the coolant geometry model is divided into three steps.

In the first step, the geometry models of the RPV, the reactor shaft core barrel and the core barrel - bottom part with all details are created and assembled together. This first geometry model represents the basic geometry model, that can be used not only in geometry creation for CFD analysis, but also for structural analysis of individual components of the nuclear reactor. Figure 1 shows fully detailed 3D CAD model of the assembled geometry model with three details.

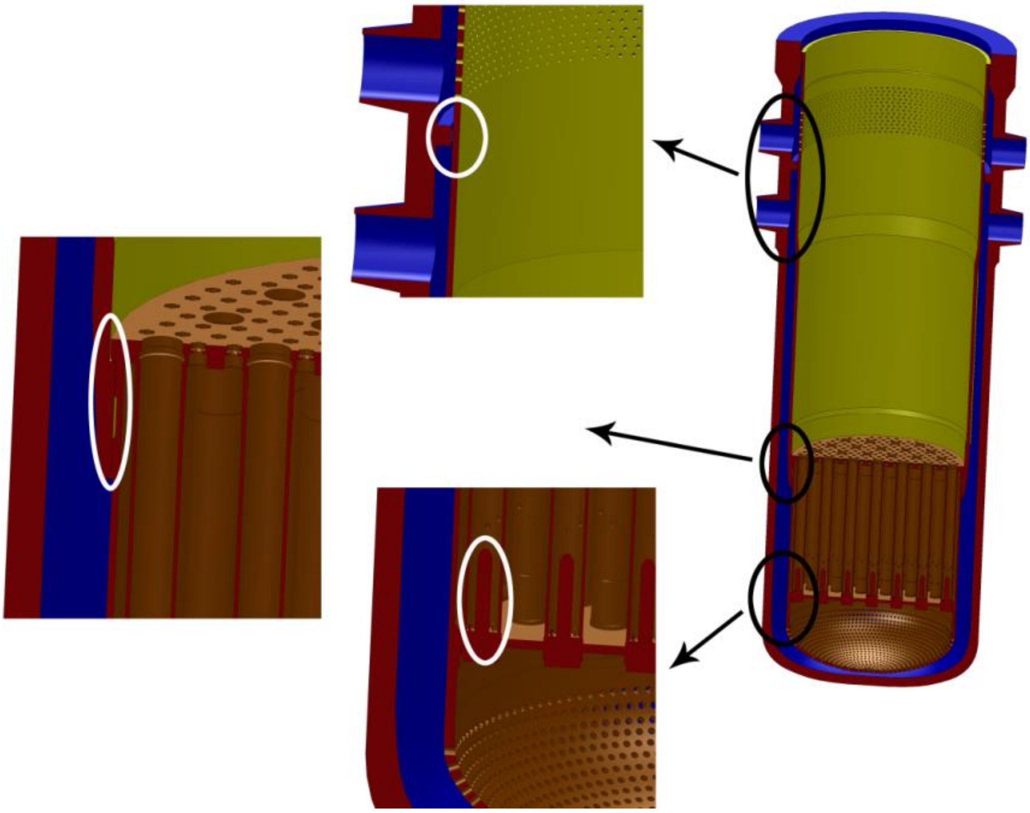

Fig. 1 Geometry model of the RPV, the reactor shaft core barrel and the core barrel - bottom part with details

In second step, the detailed geometry model of the nuclear reactor is simplified. The simplification of the geometry model is necessary, because our discretized models are limited by hardware, that are used to CFD computations. The simplifications are performed on input and also on output parts of the reactor and they are shown in Figure 1 on all three details with white colour. All three simplifications do not change the character of coolant flow but drastically simplified the discretization process of the computational region.

In third step, a negative volume of the reactor, which represents the volume of the coolant, is created. Final geometry model of the coolant in the reactor between inlet nozzles of the RPV and inlet nozzles of the FA is shown in Figure 2.

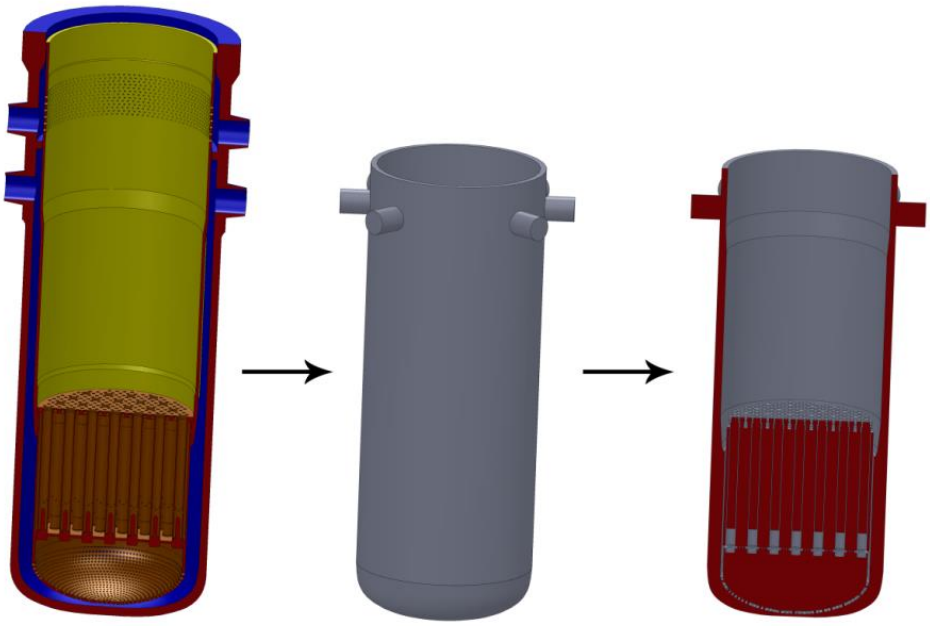

Fig. 2 Negative volume of the reactor - a volume of the coolant in the downcomer

3 Discretization of coolant in downcomer

To solve Reynolds Averaged Navier-Stokes equations (RANS) by Finite Volume Method (FVM), a division of the geometry of the coolant into small cells is necessary. The cells can directly represents finite volumes – if FVM uses Cell Centered formulation, or finite volume is created from several parts of adjacent cells – if FVM uses Vertex Centered formulation.

The process of discretization was performed in mesh tool ANSYS ICEM CFD where two different strategies were used - namely a blocking strategy for a structured mesh and an octree area meshing with Delaunay volume meshing for a tetrahedral mesh generation - see Figure 3.

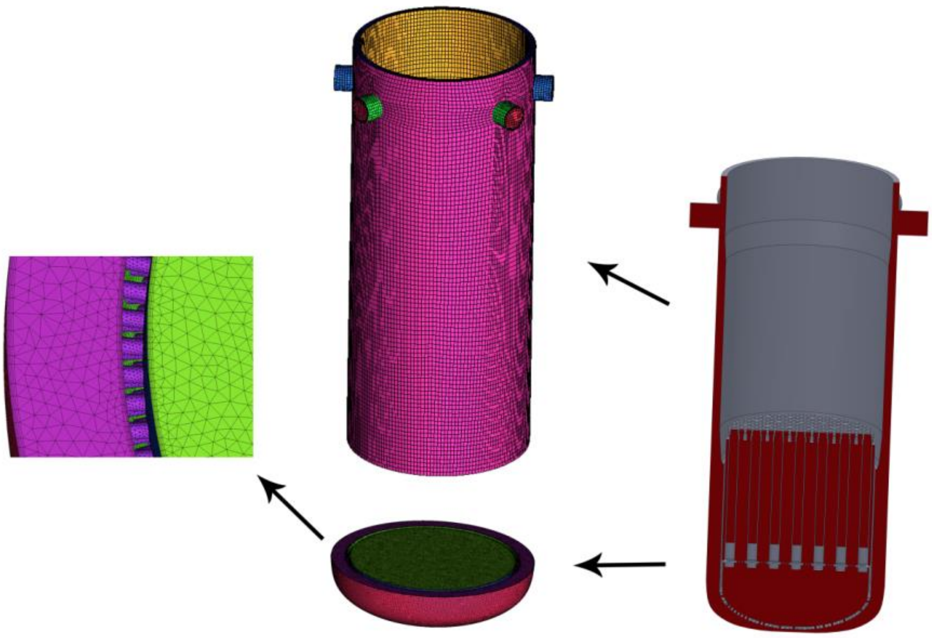

Fig. 3 Discretized volume of the coolant in the downcomer

In common, a hexahedral mesh in comparison with a tetrahedral mesh provides numerically more accurate results and the number of elements is significantly smaller. In the discretized model of the coolant, there was approximately 24 million of elements and 26 million of nodes.

Special attention was paid to the outlet region, where throttling orifices are installed in the support plate of the core barrel. Single orifice with the volume of the coolant, the simplification of the coolant and its discretization is shown in Figure 4. The volume of the coolant which is in the orifice was placed in all 312 positions in the top region of the investigated volume of the coolant.

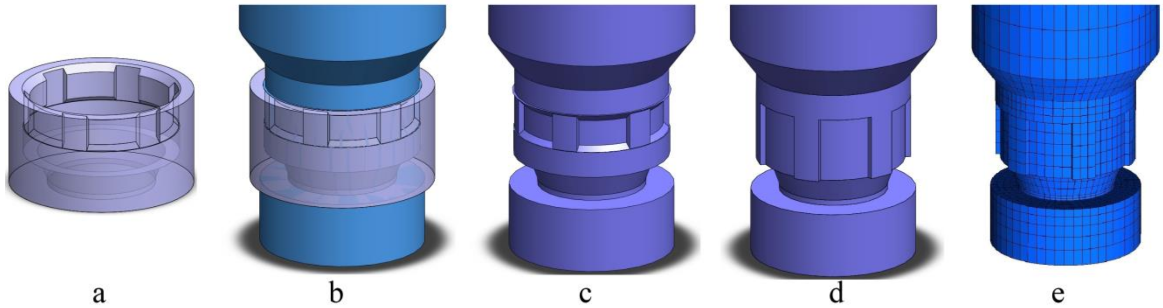

Fig. 4 Discretized volume of the coolant in the orifice

4 Boundary conditions and CFD simulations

Analysed state of the coolant flow in the downcomer of the nuclear reactor VVER 440, which is called the nominal project conditions, can be described as follows:

- coolant mass flow through the nuclear reactor: 33 383 t/hour
- coolant mass flow through each of 6 inlet nozzles are equal with the value: 1545.5 kg/s (Figure 5 - In 1 - 6)
- coolant temperature in all 6 inlet nozzles: 268.0 °C (Figure 5 - In 1 - 6)
- coolant bypass of the core: 6.3% (in whole volume of the coolant)
- coolant output pressure: 12.25 MPa (Figure 5 - Output)

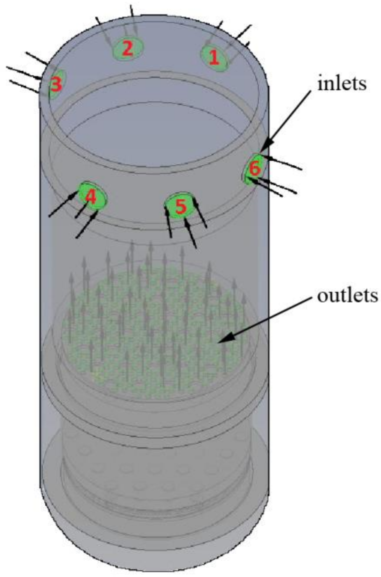

Fig. 5 Boundary conditions in the investigated region

All boundary conditions of the CFD model were set up according the nominal project conditions. The simulation was performed by CFD code ANSYS CFX with following set up:

- type of the analysis: steady-state
- turbulent model: SST
- material model of the coolant: water from the library IAPWS-IF97
- advection scheme: high resolution

The goal of the CFD analyses is to determine the distribution of the coolant velocity in the downcomer, the distribution of the coolant mass flow at the individual inlet fuel assembly nozzles and to investigate the influence of the orifice diameter change on the mass flow distribution through individual orifices, when boundary conditions are set up according the nominal project conditions.

5 The results

The distribution of the coolant velocity in the downcomer of the nuclear reactor VVER 440 in two different vertical planes is shown in Figure 6.

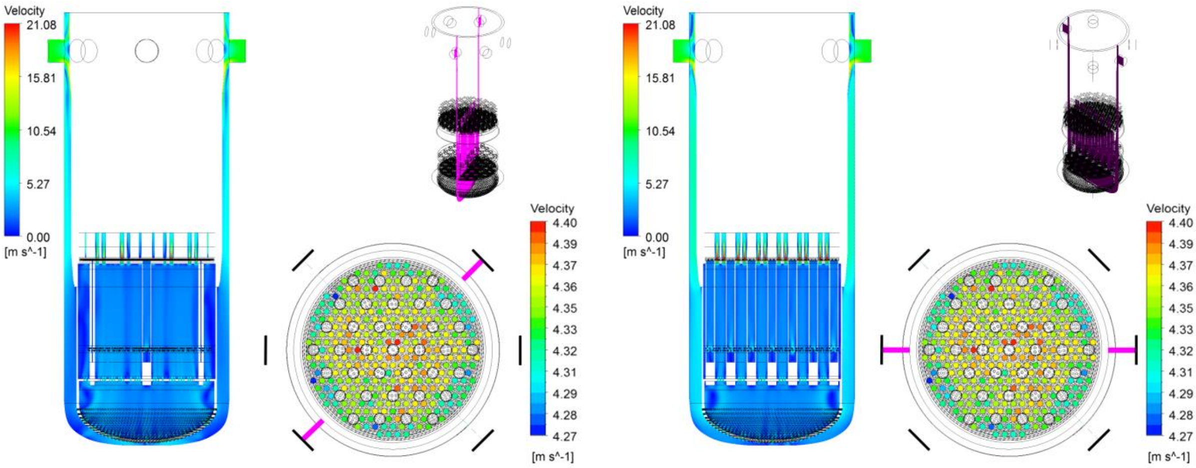

Fig. 6 Distribution of coolant velocity in downcomer in two different vertical planes

From this figure we can observe the difference in maximum values of the velocity in both investigated vertical planes in the downcomer region - between the inlet nozzles and the elliptic bottom, which is caused by geometry locations of the inlet nozzles in the RPV. As we can see from this figure, the coolant velocity can reach a value up to 21 m/s. The highest velocity of the coolant is located near the outlet part of the investigated region, where the orifice is located - see Figure 7.

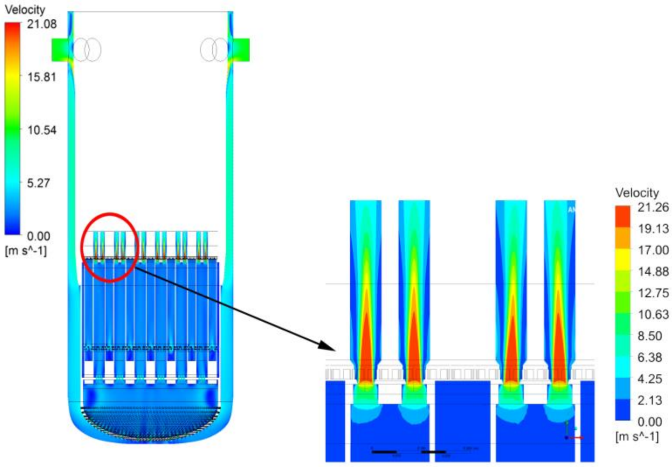

Fig. 7 Discretized volume of the coolant in the orifice

Coolant flows through the 312 orifices to the individual fuel assemblies and 37 protecting tubes to the individual control rods. The nominal diameter (the smallest diameter of the throttling orifice - see Figure 4) of the orifice is 50 mm but in real conditions this diameter can be reduced by sediments, which can even increases the velocity in this region.

Figure 8 shows a large vortex structure in the coolant, which is formed over the elliptic bottom with perforations and below horizontal lower perforated plate of core barrel - bottom part.

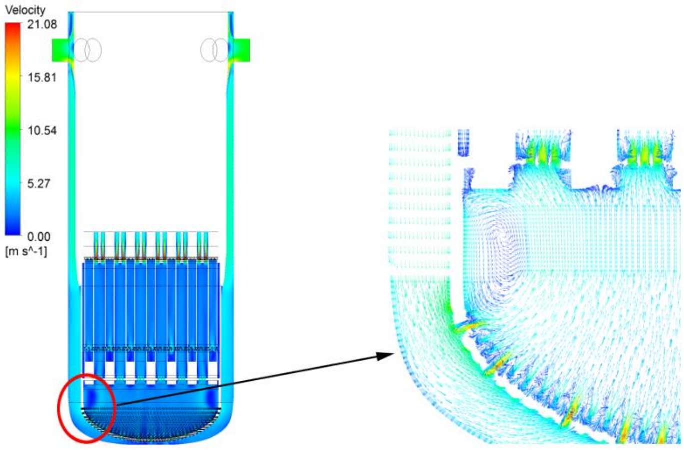

Fig. 8 Vortex structure in the coolant

Figure 9 shows the distribution of the coolant mass flow at the outlet of the investigated region, which also represents the inlet into the fuel assemblies nozzles (individual throttling orifices).

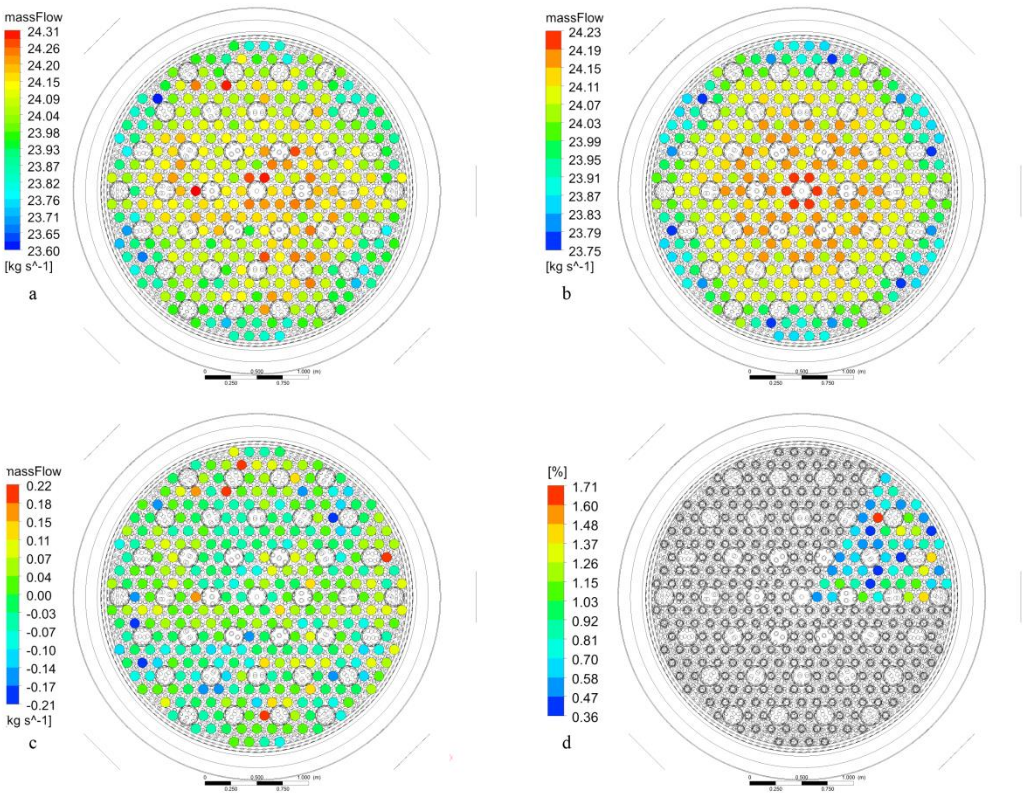

Fig. 9 Distribution of the coolant mass flow at the outlet of investigated region

There are shown only 312 orifices of fuel assemblies, but in the CFD calculation also the flow through protecting tubes for control rods were considered. Figure 9a shows the distribution of the coolant mass flow in individual orifices of fuel assembly inlet nozzles. In this picture we can see that the mass flow of the coolant is in the 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. Figure 9b shows the average coolant mass flow in individual orifices when 1/6 symmetry of the reactor core is considered. The difference between the actual coolant mass flow in the individual orifice and the average value of correspondent 6 orifices is shown in Figure 9c. In Figure 9d, there is shown the maximal difference of correspondent orifices expressed in percentage. As we can see from these last two figures, the difference between the actual and the average value of correspondent 6 orifices is in the range -0.21 kg/s to 0.22 kg/s and their maximal percentage difference is in the range from 0.36 % to 1.71 %. But most of the coolant mass flow percentage difference in considering 1/6 symmetry is below 1 %.

The influence of the orifice diameter change on the mass flow distribution through individual orifices was investigated in three different orifices, which also represents the inlet into three different FA (Figure 10 left):

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

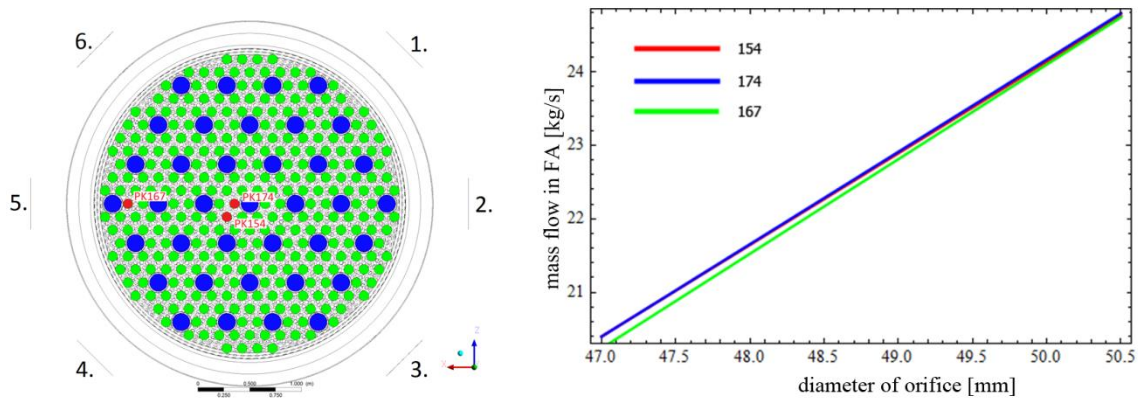

Fig. 10 Dependence of mass flow on diameter of selected orifices

The orifice diameter change was considered in the range from 47 mm to 50.5 mm with the 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:

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

Linear relationship between the coolant mass flow and the orifice diameter for all three investigated orifices is shown in Figure 10 right.

CONCLUSION

The presented paper dealt with the modelling of thermo-hydraulic conditions in the downcomer of the nuclear reactor VVER 440. In CFD analysis, the nominal boundary conditions were considered. Area of interest was the fuel assemblies inlet region where the distribution of the mass flow and the velocity of the coolant were investigated. From obtained results we can conclude, that the velocity in the downcomer region is not fully homogenized, that is caused by geometry locations of the inlet nozzles in the RPV. But the coolant flow is redistributed in the core barrel - bottom part and the coolant has maximal velocity at the individual orifices, where the velocity reach a value up to 21 m/s. The difference between the actual and the average mass flow of correspondent orifices in considering 1/6 symmetry of the reactor core is in the range -0.21 kg/s to 0.22 kg/s and the maximal percentage difference is in the range from 0.36 % to 1.71 %. Also the influence of the orifice diameter change on the mass flow distribution through individual orifices was investigated. The results show that the relationship between the coolant mass flow through the investigated orifice and the diameter of the orifice is practically linear with only local influence.

This CFD analysis can be considered as an introductory CFD analysis of real conditions in fuel assembly inlet nozzles. In our next research we will focused on real conditions in the reactor pressure vessel inlet nozzles, but also on the influence of a generated heat in individual fuel assemblies on the coolant mass flow distribution in individual orifices.

ACKNOWLEDGEMENT

This work was supported by the Slovak Research and Development Agency under the contract No. 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] Muškát, P., Urban, F., Pulmann, M.: Merania na fyzikálnom modeli palivového článku jadrového reaktora. In *Strojnícky časopis - Journal of Mechanical engineering*. Roč. 59, č. 5-6 (2008), s.305-315. ISSN 0039-2472.

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

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

[5] Donea, J., Huerta, A. Finite Element Methods for Flow Problems. John Wiley & Sons. ISBN 9780471496663. 2003.

[6] Oberkampf, W. L., Trucano, T. G. Verification and Validation in Computational Fluid Dynamics. Sandia National Laboratories. 2002.

[7] ANSYS CFX. Theory manual. 2016.