DE GRUYTER OPEN

*Journal of* MECHANICAL ENGINEERING – Strojnícky časopis,
VOL 67 (2017), NO 1, 69 - 76

STU SjF

# INFLUENCE OF BYPASS ON THERMO-HYDRAULICS OF VVER440 FUEL ASSEMBLY

**JAKUBEC Jakub[1], PAULECH Juraj[1], KUTIŠ Vladimír[1], GÁLIK Gabriel[1]**

*[1]Slovak University of Technology in Bratislava, Faculty of Electrical Engineering and Information Technology, Department of Applied Mechanics and Mechatronics, Ilkovičová 3, 812 19 Bratislava, Slovakia, email: jakub.jakubec@stuba.sk*

**Abstract:** The paper deals with CFD modelling and simulation of coolant flow within the nuclear reactor VVER 440 fuel assembly. The influence of coolant flow in bypass on the temperature distribution at the outlet of the fuel assembly and pressure drop was investigated. Only steady-state analyses were performed. Boundary conditions are based on operating conditions. ANSYS CFX is chosen as the main CFD software tool, where all analyses are performed.

**KEYWORDS:** CFD analysis, ANSYS CFX, Fuel assembly, VVER 440, thermal-hydraulics

## 1 Introduction

Nuclear reactor safety, thermohydraulics is a very important subject [1]. Thermohydraulics as a multiphysical domain influences not only the thermal conditions of nuclear fuel, but also the distribution of neutron flux within the reactor core, thermal and pressure loading of reactor pressure vessel and dictates the critical value of heat flux, which can flow form the fuel rod to coolant. For many years, thermohydraulics of nuclear reactors has been investigated only by specialized system codes, like RELAP and ATHLET. In the last decade, computational fluid dynamics - CFD [2] emerged as a very useful alternative tool to analyse thermohydraulics, where real 3D geometry can be considered. The paper presents the application of CFD for the investigation of fuel assembly bypass coolant mass flow and its influence on the coolant temperature distribution within the fuel assembly head.

## 2 Geometric model and discretization

To perform thermo-hydraulic analysis of the fuel assembly in the reactor VVER440, it is necessary to create an equivalent 3D geometric model of the coolant in the fuel assembly (FA). Creating the geometric model of coolant is divided into three steps (Fig.1).

In the first step, an accurate geometric model of the fuel assembly with all details is created. This model includes parts of the protective tubing known as the fixator, where the thermocouple housing is placed. This 3D geometric model represents real geometry of FA, which also can be used for structural analysis.

Fig.1 shows fully detailed 3D CAD model of fuel assembly. In the Fig.1 there is bypass outlet from fuel assembly in the bottom and bypass inlet in top, marked with blue circle.

Second step, detailed geometric model of fuel assembly is simplified because of the future mesh generation and computational hardware limitations. Simplifications are performed on input and also on output parts of fuel assembly. Those modifications won't have significant influence on the coolant flow (Fig.1).

DOI: 10.1515/scjme-2017-0007, Print ISSN 0039-2472, On-line ISSN 2450-5471 ©2017 SjF STU Bratislava

In third step, negative volume of fuel assembly, which represents the volume of coolant is created. In this step, also the geometry of fixator tube from upper core supporting plate is modelled, where the thermocouple housing is placed.

Final geometric model of the coolant in fuel assembly is shown in Fig.1 (3rd step). The final geometry model of coolant also contains the central tube, thermocouple housing and shroud, modelled as a solid part.

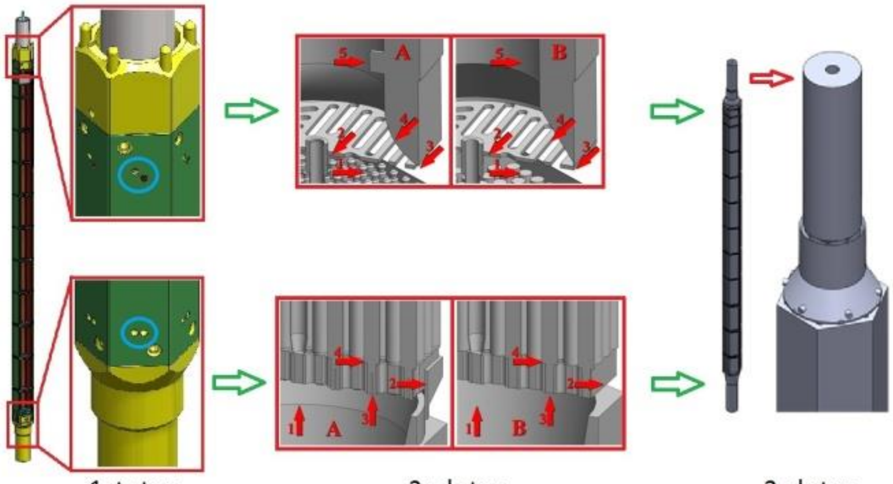
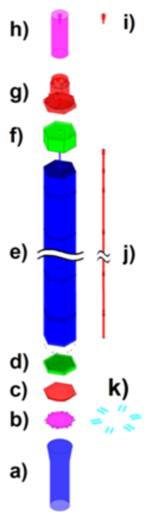



Fig. 1 3D CAD model of the Fuel assembly (1st step), simplifications in particular areas (2nd step) and final geometry model of coolant in fuel assembly (3rd step)

To solve Reynolds Averaged Navier-Stokes equations (RANS) by Finite Volume Method (FVM), division of the geometry of coolant into small cells is necessary. The process of discretization was performed in mesh tool ANSYS ICEM CFD where blocking strategy was mostly used. In order to use this strategy the whole geometry of coolant was divided into parts to provide better and easier way to create a suitable mesh (see Fig.2).

Fig.3 shows example of the most complicated part of the mesh created in the fuel rods area, which includes spacer grids and central tube.




| | elements |
|---|---|
| a | 1081584 |
| b | 110088 |
| c | 115764 |
| d | 343356 |
| e | 57159784 |
| f | 1218528 |
| g | 2723870 |
| h | 1648320 |
| i | 68040 |
| j | 293504 |
| k | 44928 |
| total | 64807845 |

Fig.2 Mesh parts with element counts

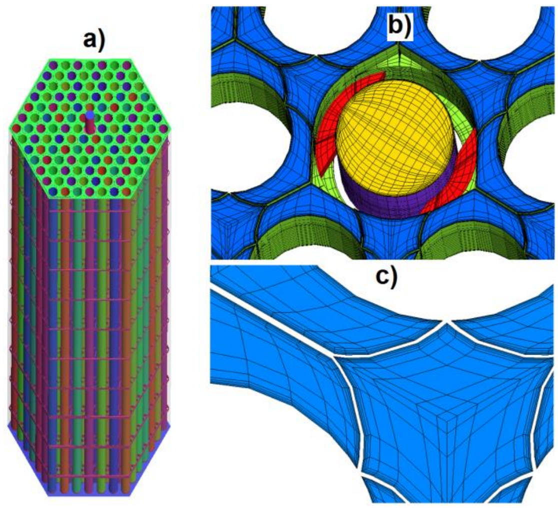



Fig.3 Mesh part (Fig.2 - e): (a) - geometry of the part, (b) – central tube perforations detail, (c) – detail of boundary layer

All meshed parts were connected by GGI connection in ANSYS CFX. The discretized model of fuel assembly coolant contains approximately 70 million nodes and 65 million elements (Fig.2). These numbers represent the limit of the hardware and software configuration, which was used for CFD computations.

## 3 CFD simulations and results

Very important parameter, which plays a crucial role in the heat removal of FA is mass flow of the coolant, which flows through individual assemblies. However, the entire mass of the coolant that enters the FA does not necessarily flow through all fuel rods. Minor part of the coolant leaves the FA at the lower part (still under the fuel rods) and enters the so called inner FA space, flows along FA and enters into the head above the fuel rods and mixing grid. This effect is known as FA bypass. Bypass coolant mass flow at the inlet to bypass and at the outlet from bypass could be uneven based on different hydraulic losses of nearby FAs.

The boundary conditions were based on Russian experiments [3]. This experiment was used to validate the utilized CFD model in our previous research [4].

As it was mentioned in the introduction, one of the problems of the VVER 440 fuel assembly is coolant temperature measurements. Coolant temperature measured by the thermocouple that is placed at the outlet of the fuel assembly in the reactor part, protective tube unit, could be slightly different from the average coolant temperature at the outlet. This could be caused by nuclear radiation heating of the thermocouple and of course by poor coolant mixing in the upper part of the fuel assembly [5]. This is the reason why the Kurchatov Institute built a test facility to examine the processes affecting mixing processes such as bypass and central tube flow.

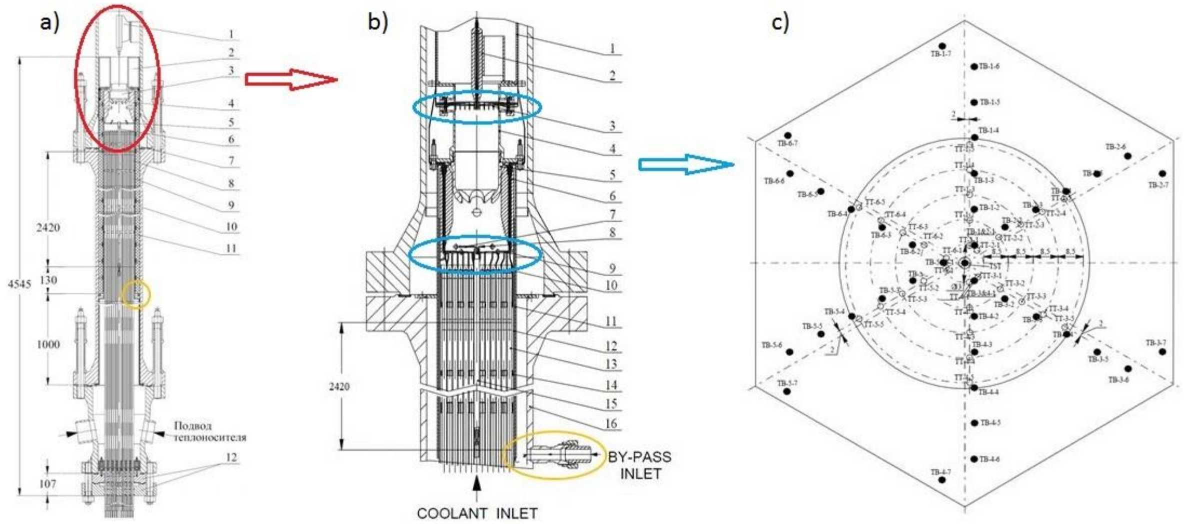



Fig.4 (a) – test facility cross-section, (b) – upper part detail with thermocouples, (c) – distribution of thermocouples

Test facility consists of one fuel assembly equipped with electrically heated fuel rods representing rods of fissile material, where each rod could have its own thermal performance. In the upper part there is 39 thermocouples at the fuel rod outlet area, 30 thermocouples in the real fuel assembly thermocouple plane to measure coolant temperature distribution and one at the central tube outlet to measure central tube outlet temperature. The test facility is able to cover fuel assembly bypass as well (Fig.4).

Bypass coolant mass flow was considered in the range of 0% - 4% of nominal coolant mass flow at the FA inlet and 0% - 5% at the bypass outlet. Coolant temperature at the bypass outlet was considered to be the same as the coolant temperature at the inlet to the FA + 10°C gain. Those bypass parameters were chosen to be able to examine its influence on FA output parameters. This means that they do not have to fit real operational conditions.

Boundary conditions (Fig.4):

- nominal inlet mass flow: 24.5kg/s
- inlet temperature: 268°C
- output pressure: 12.25MPa

Bypass parameters:

- inlet mass flow: 0-4% of FA nominal mass flow
- outlet mass flow: 0-5% of FA nominal mass flow
- outlet temperature: 278°C (FA inlet temp. +10°C gain)

Turbulent model:

- SST

Prescribed thermal power distribution:

- total thermal power = 5.77MW
- prescribed as the heat flux for each fuel rod

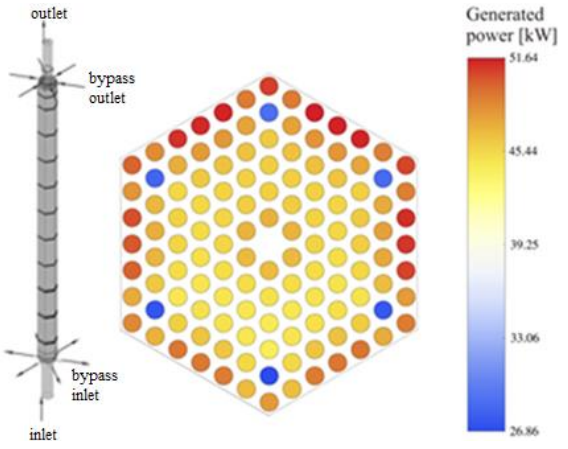



Fig.5: Boundary conditions – left, radial power distribution in fuel rods – right

All simulations were performed as steady state studies, ANSYS CFX was chosen as the CFD tool for all simulations. The model contains two domains: fluid and solid. The solid domain is used for modelling heat transfer across the central tube wall and thermocouple housing. The connection between individual mesh parts is realized by GGI connection. Material parameters of coolant (water) were defined by ANSYS CFX material library IAPWS-IF97.

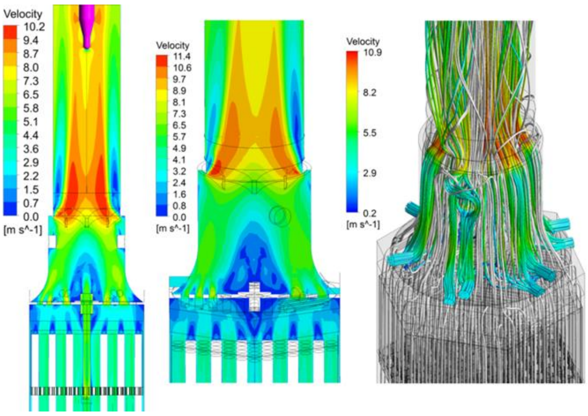


Fig.6 Coolant velocity distribution at the upper part of FA

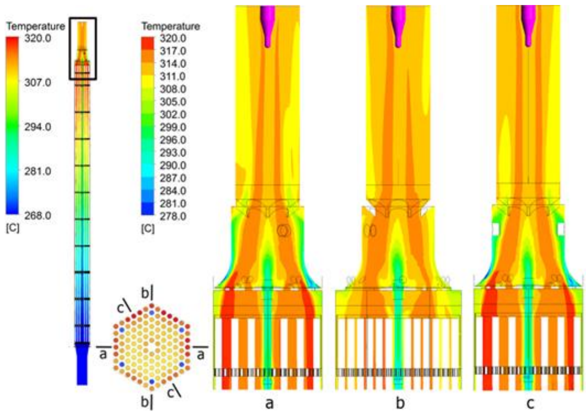



Fig.7 Coolant temperature distribution at the FA

Fig.6 shows the upper part of the FA with highest chosen values of bypass mass flow (4% inlet and 5% outlet of total FA inlet mass flow). It is obvious that the bottom part of the fixator increases coolant velocity with respect to the decrease in diameter by upto 10 m/s. Higher coolant flow velocities remain at the fixator tube centre and considering imperfect coolant mixing in the FA head (Fig.7), it is expected that the influence of this flow on coolant temperature measurement by thermocouple compared to average coolant temperature at the FA outlet. Fig.6, right, shows where bypass enters the FA head by velocity streamlines and is forced by the main stream into the fixator tube walls.

Detailed coolant temperature distribution in the upper part of the FA is shown in Fig.7 by contours (same boundary conditions as in Fig.6). All 3 cross-sections show how the main hot coolant stream is forced into the centre of the fixator tube by the bypass and even by the geometry. They also show great influence on the thermocouple housing since it is placed in the centre of the fixator tube. The effect of the main hot stream is even bigger considering weighting of the coolant flow velocities from the previous Fig. Another effect that could cause a difference between thermocouple temperature and average FA outlet coolant temperature is central tube coolant flow, since the central tube is placed right under the thermocouple housing. The central tube coolant outlet temperature is 19°C colder than the main coolant temperature in the fuel rod area (same boundary conditions as in Fig.6). Also, the central tube outlet mass flow is only approx. 1% of the FA inlet mass flow (see Fig.8).

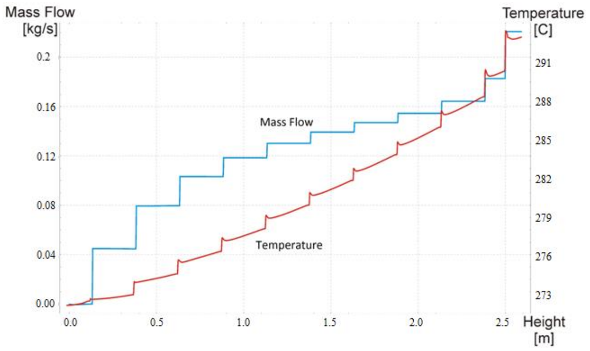



Fig.8 Mass flow and coolant temperature dependence on the height of the fuel assembly central tube

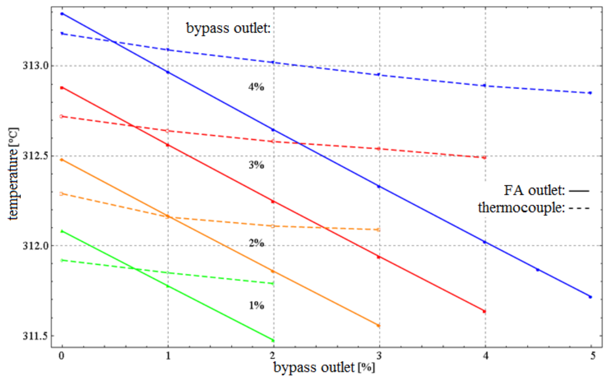



Fig.9 outlet coolant temperature and thermocouple dependence on bypass parameters

Fig.9 represents temperature (FA outlet and thermocouple) dependence on bypass mass flow parameters. Average coolant temperature at the FA outlet function and thermocouple temperature function are linear to bypass outlet mass flow parameters, but thermocouple temperature function has a lower slope compared to the outlet temperature function. It is caused by forcing the main hot stream to the coolant flow centre, closer to the thermocouple by the bypass mass flow at the inlet to the upper part of the FA. This event is also obvious in Fig.10.

Fig.10 shows coolant temperature homogenization in the upper part of the FA and in the fixator tube considering a 4% bypass mass flow at the inlet and 0-5% at the outlet within the nominal range of FA mass flow. Coolant mixing processes are not ideal and inhomogeneity at the outlet is in the range of 3.4 to 4.5°C. This figure also shows the influence of bypass and central tube flow on the coolant mixing processes.

Results from all simulations are summarized in Tab.1. It also shows differences between thermocouple temperatures and average outlet coolant temperatures and FA coolant heat up dependence on bypass parameters.

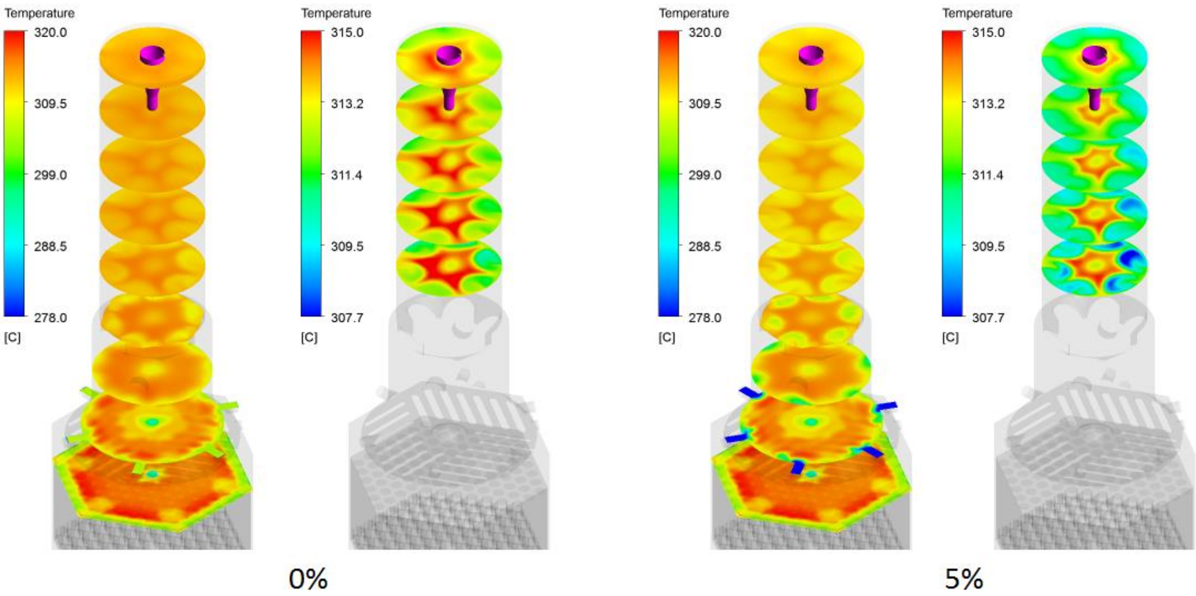



Fig.10 Coolant temperature homogenization in the upper part of the FA in two different temperature scales considering 4% bypass inlet and 0% - 5% bypass outlet

Tab.1 Thermocouple temperature and average outlet temperature dependence on bypass parameters

| bypass mass flow [%] | | average temperature [°C] | | thermocouple and FA outlet difference [°C] | FA coolant heat up [°C] |
|---|---|---|---|---|---|
| inlet | outlet | thermocouple | FA outlet | | |
| 0 | 0 | 311,57 | 311,69 | -0,12 | 43,69 |
| 1 | 0 | 311,97 | 312,08 | -0,11 | 44,08 |
| 1 | 1 | 311,85 | 311,78 | 0,07 | 43,78 |
| 1 | 2 | 311,79 | 311,47 | 0,32 | 43,47 |
| 2 | 0 | 312,29 | 312,48 | -0,19 | 44,48 |
| 2 | 1 | 312,16 | 312,17 | -0,01 | 44,17 |
| 2 | 2 | 312,11 | 311,86 | 0,25 | 43,86 |
| 2 | 3 | 312,09 | 311,56 | 0,53 | 43,56 |
| 3 | 0 | 312,72 | 312,88 | -0,14 | 44,88 |
| 3 | 1 | 312,64 | 312,56 | 0,08 | 44,56 |
| 3 | 2 | 312,58 | 312,25 | 0,33 | 44,25 |
| 3 | 3 | 312,54 | 311,94 | 0,60 | 43,94 |
| 3 | 4 | 312,49 | 311,64 | 0,85 | 43,64 |
| 4 | 0 | 313,18 | 313,29 | -0,11 | 45,29 |
| 4 | 1 | 313,09 | 312,97 | 0,12 | 44,97 |
| 4 | 2 | 313,02 | 312,65 | 0,37 | 44,65 |
| 4 | 3 | 312,95 | 312,33 | 0,61 | 44,33 |
| 4 | 4 | 312,89 | 312,02 | 0,87 | 44,02 |
| 4 | 5 | 312,85 | 311,72 | 1,13 | 43,72 |

## 4 Conclusion

The paper presents CFD modelling and simulation of coolant flow in the fuel assembly of the VVER 440 nuclear reactor. Main area of interest was the upper part of the fuel assembly and part of the protective tube unit with thermocouple. The goal was to investigate the influence of bypass mass flow on the coolant mixing processes and temperatures within the FA upper area. It is obvious that the FA bypass has a significant influence on the coolant flow profile and coolant temperatures registered by the thermocouple compared to average coolant temperature at the FA outlet. Even coolant flow from the central tube may affect measured coolant temperature.

This is the reason why it is necessary to determine all possible influences which causes differences between coolant temperature on the outlet and temperature data from the thermocouple especially by current projected thermal power increase of nuclear power reactor VVER440.


## ACKNOWLEDGEMENT

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 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] N.E. Todreas, M.S. Kazimi. Nuclear Systems Volume I: Thermal Hydraulic Fundamentals. CRC Press; 2 edition, **2011**. ISBN 1439808872.

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

[3] D.A. Oleksyuk L.L. Kobzar. "Experiments on simulation of coolant mixing in fuel assembly head and core exit channel of VVER-440 reactor," Kurchatov Institute, Moscow, **2006**

[4] J. Jakubec, V. Kutiš, G. Gálik, J. Paulech. Coolant mixing processes simulations of nuclear reactor VVER440 fuel assembly. SVSFEM ANSYS User's Group Meeting, **2015**.

[5] V. Kutiš, J. Jakubec, J. Paulech, G. Gálik, T. Sedlár. CFD Analysis of Downcomer of Nuclear Reactor VVER 440. *Journal of Mechanical Engineering – Strojnícky časopis*, **2016** (66), No. 2, 55 – 62.