PARAMETRIC STUDY OF BYPASS INFLUENCE ON MIXING PROCESSES OF VVER440 FUEL ASSEMBLY

***Jakub Jakubec*[1], *Vladimír Kutiš*[1], *Gabriel Gálik*[1], *Juraj Paulech*[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, Bratislava, Slovak Republic*

E-mail: jakub.jakubec@stuba.sk

Received 05 May 2016; accepted 18 May 2016

1. Introduction

In nuclear reactor safety, thermo-hydraulics is very important subject [1]. Thermo-hydraulics as multi-physical domain has influence not only on thermal conditions of nuclear fuel, but also influences on distribution of neutron flux in reactor core, thermal and pressure loading of reactor pressure vessel and sets up critical value of 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 [2] emerged as very useful alternative tool to analyze thermo-hydraulics, where real 3D geometry can be considered. The paper presents the application of CFD on investigation of bypass flow influence on coolant temperature distribution in fuel assembly head.

2. Geometric model and discretization

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

In the first step, very accurate geometric model of fuel assembly with all details is created. This 3D geometric model represents real geometry of FA, which also can be used for structural analysis. 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). In third step, negative volume of fuel assembly, which represents the volume of coolant is created.

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

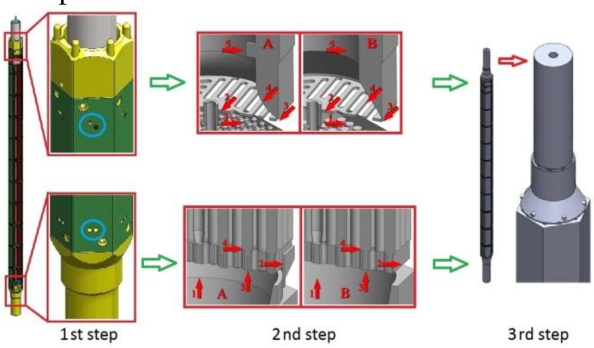
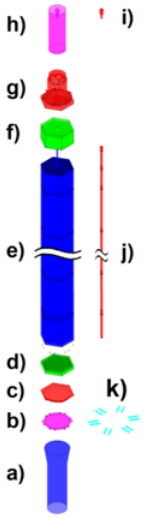


Fig.1: *3D CAD model of Fuel assembly (1st step), simplifications in particular areas (2nd step) and geometry model of coolant in fuel assembly (3rd step)*

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 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*

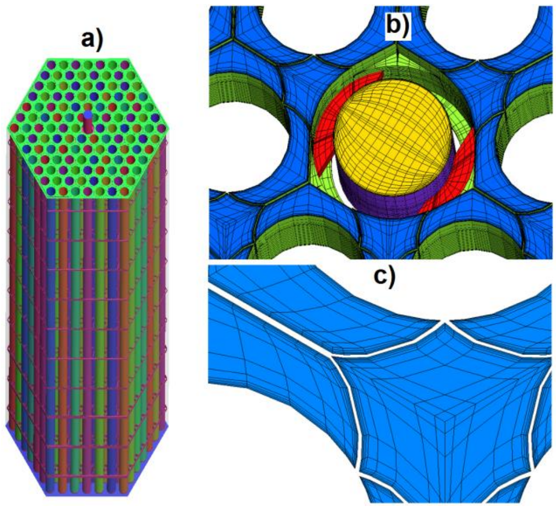


Fig.3 *Mesh part (Fig.2 - e): (a) - geometry of the part, (b) – central tube perforations detail, (c) – detail of boundary layer*

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

3. CFD simulations and obtained results

Very important parameter, which plays crucial role in heat removal from FA is mass flow of coolant, which flows through individual fuel assemblies. Not entire mass of the coolant which enters FA flows through fuel rods. Minor part of the coolant leaves FA in lower part (still under the fuel rods) and enters so called inner FA space, flows along FA and back enters to its head above fuel rods and mixing grid. It is called 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.

Used boundary conditions were based on the Russian experiment [3]. This experiment was used for validating used CFD model in our previous researches [4]. Test facility is basically fuel assembly equipped with electric heated fuel rods replacing rods with fissile material, where each rod could have its own thermal performance. In the upper part there are 69 thermocouples placed in order to monitor coolant temperature distribution.

Bypass coolant mass flow was considered in range 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 as coolant temperature at the inlet to FA + 10°C gain. Those bypass parameters were chosen to be able to examine its influence on FA output parameters. It means they don't 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

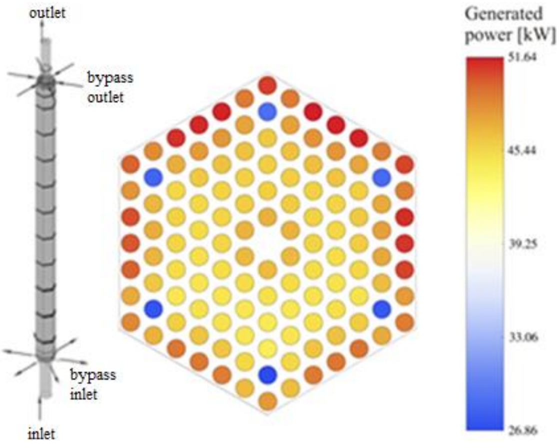


Fig.4: *Boundary conditions – left, radial power distribution in fuel rods – right*

All simulations were performed as steady state, ANSYS CFX was chosen as CFD tool for all simulations. The model contains two domains: fluid and solid. 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.5 shows upper part of FA with highest chosen values of bypass mass flow. As it is obvious bottom part of fixator increases coolant velocity by diameter decrease up to 10 m/s. Higher coolant flow velocities remains in the fixator tube centre and considering imperfect coolant mixing in FA head (Fig.6), it is expected the influence of this flow on coolant temperature measurement by thermocouple comparing to the average coolant temperature at the FA outlet. Right side of Fig.5 shows where bypass enters FA head by velocity streamlines and how it is forced by the main stream to the fixator tube walls.

Detailed coolant temperature distribution in upper part of FA is shown in Fig.6 by contours. All 3 cross-sections show how main hot coolant stream is forced to centre of 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 fixator tube. The effect of main hot stream is even bigger considering weighting of the coolant flow velocities from previous Fig.

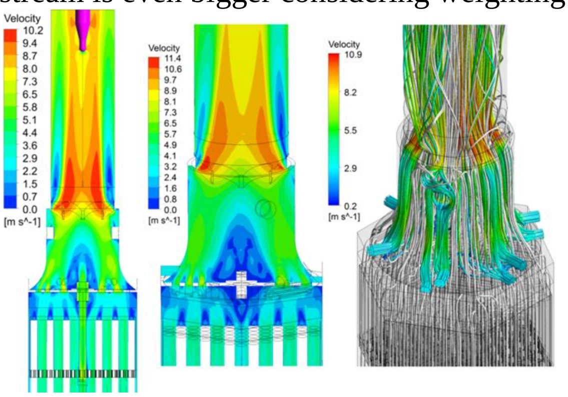


Fig.5: *Coolant velocity distribution at the upper part of FA, left and middle – velocity distribution by contour in different views, right – velocity streamlines from bypass*

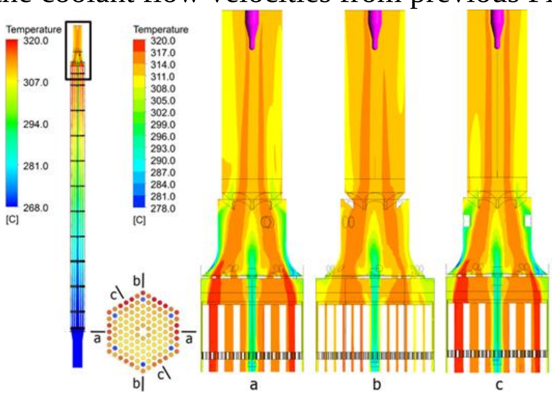


Fig.6: *Coolant temperature distribution: left – along whole FA, right – in FA upper part in cross-sections*

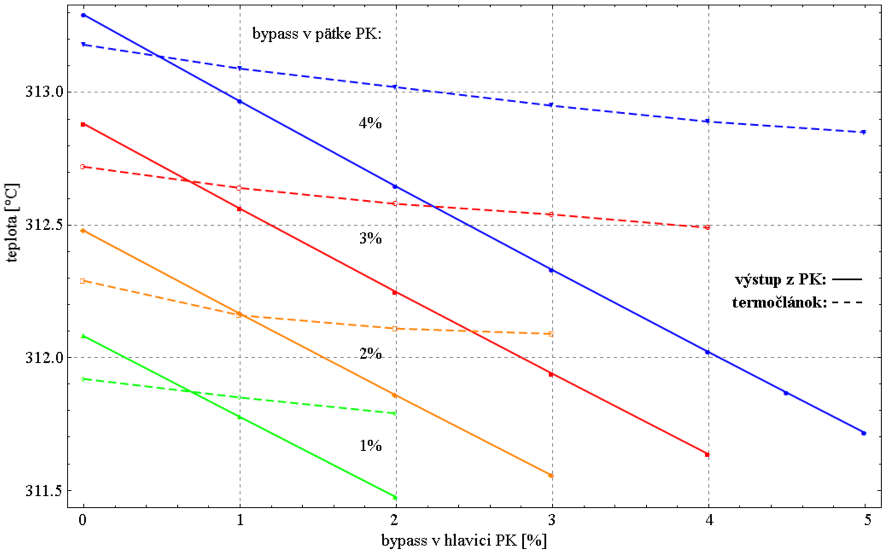


Fig.7: *Outlet coolant temperature and thermocouple dependence on bypass parameters*

Fig.7 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 lower slope compared to outlet temperature function. It is caused by forcing main hot stream to the coolant flow centre, closer to the thermocouple by the bypass mass flow at the inlet to upper part of FA

4. Conclusions

The paper presents CFD modelling and simulation of coolant flow in fuel assembly of nuclear reactor VVER 440. Goal was to investigate influence of bypass mass flow on the coolant mixing processes and temperatures in FA upper area. It is obvious that the FA bypass has 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 by the thermocouple registered by the thermocouple. This is the reason why to determine all possible influences which causes differences in coolant temperature measurement 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.

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.