CFD MODELLING OF BYPASS IN FUEL ASSEMBLY OF NUCLEAR REACTOR VVER 440

Jakub Jakubec[1], Vladimír Kutiš[1], Juraj Paulech[1], Gabriel Gálik[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
E-mail: jakub.jakubec@stuba.sk

Received 07 May 2015; accepted 14 May 2015

1. Introduction

Detailed knowledge of the thermo-hydraulic processes in fuel assembly of nuclear reactor is very important from operational safety point of view. Modern computer simulation techniques, like CFD or FEM, can be very useful in detail study of such processes, because after verification and validation processes of CFD model, you can relatively easily change boundary and initial conditions, or other input parameters of the model.

The flow field in fuel assembly of nuclear reactor VVER 440 is very complex mostly in fuel assembly head, where the thermocouple is placed. In the fuel assembly head main hot coolant stream from the fuel rods area is mixed with colder coolant from central tube and bypass. Thermocouple as the only point of coolant temperature measurement in fuel assembly has to register average coolant temperature at the outlet. In our research, we focused on modelling and simulation of thermo-hydraulic processes in fuel assembly. All CFD analyses were performed by ANSYS CFX software [1].

2. Geometric model and discretization

To perform thermo-hydraulic analysis of fuel assembly of reactor VVER440, it is necessary to create 3D geometry model of coolant in the fuel assembly. Creating of geometry model of coolant is divided into three steps (Fig.1). In the first step, geometry model of fuel assembly with all details is created. This first geometry model represents the detailed geometry model, that can be used not only for geometry creation for CFD analysis, but also for structural analysis of individual components of fuel assembly. 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.

In second step, detailed geometry model of fuel assembly is simplified. The simplification of detailed fuel assembly geometry model is necessary, because our discretized models are limited by hardware, that are used to CFD computations. Simplifications are performed on input and also on output part of fuel assembly (Fig.1)

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

Final geometry model of coolant in fuel assembly with thermocouple housing is shown in Fig.1. The final geometry model of coolant contains not only all internal fuel assembly components like supporting grid, spacer grid or mixing grid, but there is also modelled coolant flowing across central tube and central tube itself as a solid part.

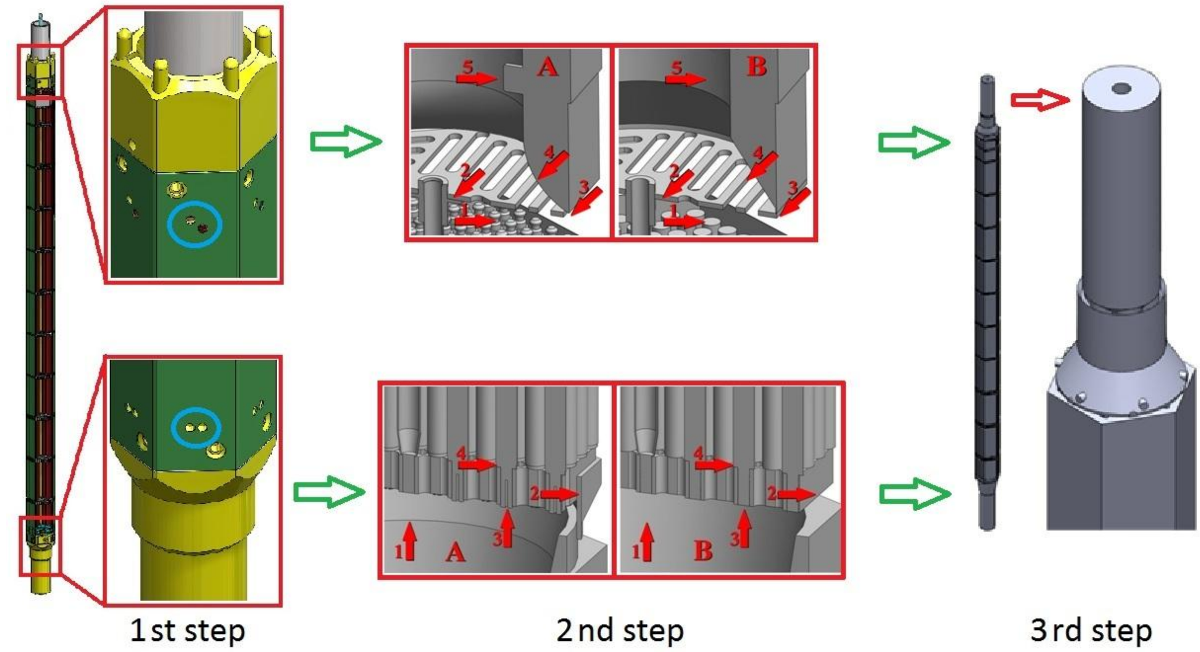

Fig.1: *Detailed 3D CAD model of Fuel assembly (1st step), simplifications in particular areas (2nd step) and geometry model of coolant in fuel assembly (3rd step)*

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.

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

3. CFD simulations and obtained results

Bondary conditions for the analyses are defined by following Russian experiment investigating same problems [2]:

- nominal inlet mass flow: 10.88 kg/s
- inlet temperature: 195.6 °C
- output pressure: 9.16 MPa

Bypass parameters:

- inlet mass flow: 0.52 kg/s
- inlet temperature: 195.6 °C

Turbulent models:

- SST, k-ω, k-epsilon, BSL [3]

Prescribed thermal power distribution in fuel rods (Fig.2):

- total thermal power = 1242 kW
- prescribed as the heat flux for each fuel rod

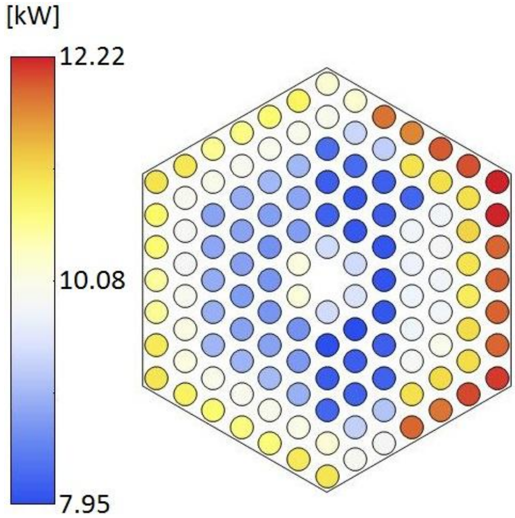

Fig.2: *Prescribed thermal power distribution*

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 shroud. Material parameter of coolant (water) were defined by ANSYS CFX material library IAPWS-IF97.

Tab. 1. *Monitor points of experiment and simulations.*

		outlet [°C]	thermocouple [°C]	central tube outlet [°C]	ΔT (thermo-inlet) [°C]
experiment		220	219.7	207.2	24.3
Turbulent models	SST	220.67	218.81	208.63	23.21
	k-ω	220.68	219.04	208.39	23.44
	BSL	220.67	220.12	208.08	24.53
	k-epsilon	220.68	219.74	208.9	24.14

Tab.1 shows monitor points of temperatures (outlet from fuel assembly, thermocouple, central tube outlet) for used turbulent models compared to the experiment. Outlet temperatures in all used turbulent models are very similar what was expected. Differences between outlet temperatures from experiment and simulations are almost 0.7 °C which points out difference of total thermal power in experiment and simulations where it is prescribed as heat flux. This problem may be caused by inaccurate definition of the fuel rods height. Coolant temperatures from central tube outlet is highly dependent on pressure condition in fuel assembly (see Fig.3). Pressure conditions defines amount of coolant which enters central tube so it has great influence on mass flow in central tube. Thermocouple itself is placed right above central tube outlet and therefore colder coolant flow from the central tube than coolant from the fuel rod area affect temperature measured by the thermocouple (see Tab.1., Fig.4).

Coolant flow in upper part of fuel assembly is shown in Fig.4. Fig.1 shows in blue circles (left side of the Fig.1) areas where in the bottom of fuel assembly part of the coolant leaves fuel assembly and enters co called inter fuel assemblies space, flows upwards to fuel assembly head and back enters fuel assembly. Fig.4 also shows how colder stream of coolant from fuel assemblies space enters main hot coolant stream. By mixing colder coolant stream from the bypass is forced to the edge and main hot coolant stream is forced to the centre of the flow.

It is obvious that colder coolant stream from central tube is mixed with hot coolant from fuel rod area but it is still able to affect coolant temperature in the area of thermocouple to be lower than average coolant temperature measured on the outlet.

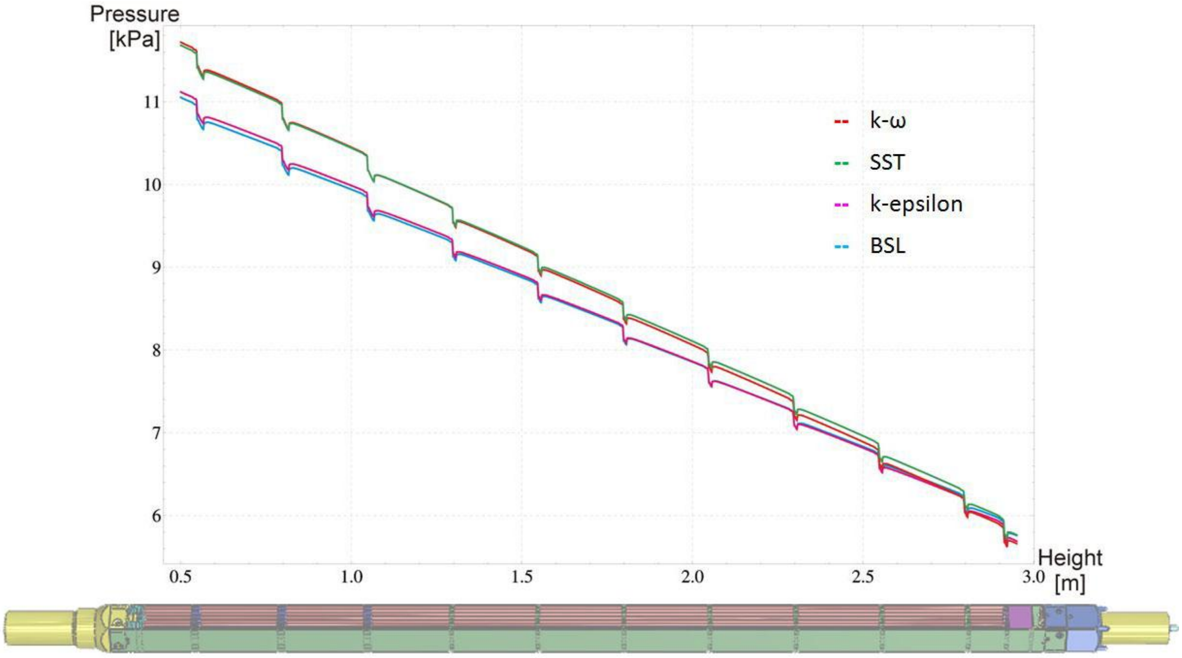

Fig.3: *Dependence of pressure drop along the central tube height for used turbulent models*

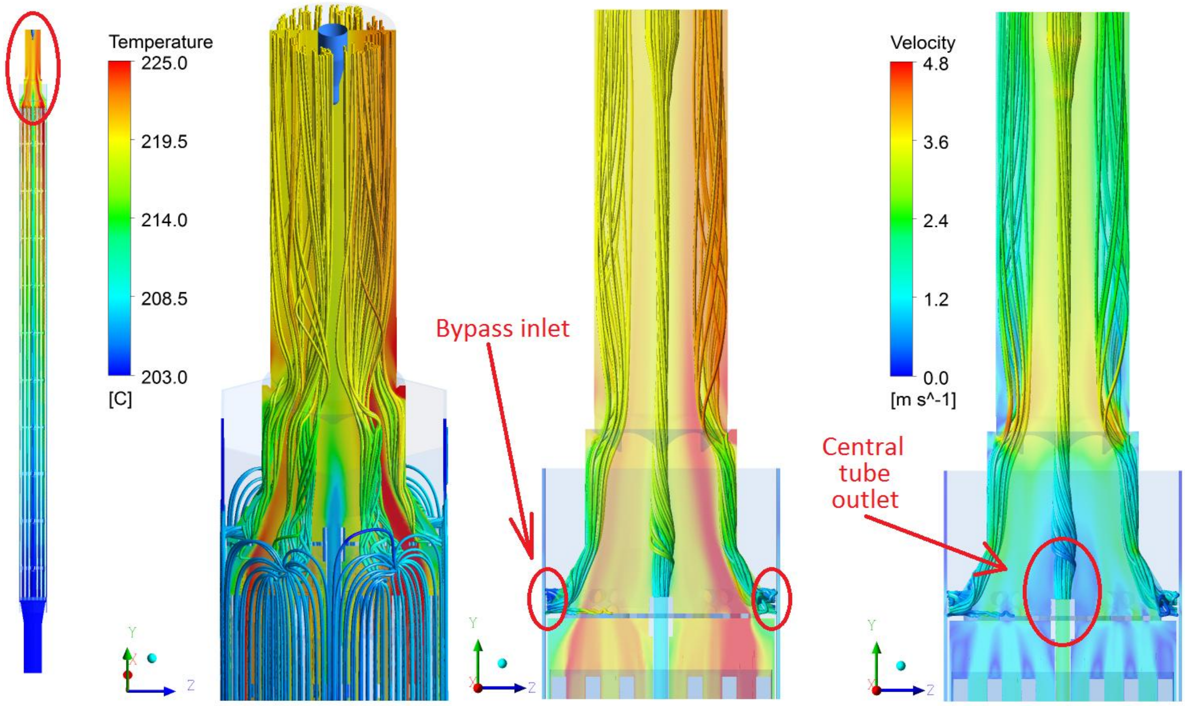

Fig.4: *Bypass inlet to FA and central tube outlet coolant streamlines, coolant temperature and coolant velocity distribution in top part of FA (SST turbulence model)*

4. Conclusions

The paper presents CFD modelling and simulation of coolant flow in fuel assembly of nuclear reactor VVER 440. Area of interest was the top part of fuel assembly. Goal was to investigate the bypass influence of different turbulent models on the output parameters such as coolant temperatures and temperature registered by the thermocouple. It is obvious that the bypass has significant influence on the coolant flow profile and also coolant flow from the central tube may affect temperature registered by the thermocouple. This is the reason why it is necessary to determine influences which may cause differences between coolant temperature on the outlet and temperature data from the thermocouple.

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] ANSYS CFX. Help manual, (2015).
[2] L.L. Kobzar, D.A. Oleksyuk, Experiments on simulation of coolant mixing in fuel assembly head and core exit channel of VVER-440 reactor, 16th Symposium of AER on VVER Reactor Physics and Reactor Safety.
[3] D.C. Wilcox: Turbulence Modeling for CFD, DCW Industries, Inc, (1993).