FUEL ASSEMBLY BYPASS CALCULATIONS OF VVER440 IN CFD MODEL

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 2017; accepted 24 May 2017

1. Introduction

Detailed knowledge of the thermal-hydraulic processes in the fuel assembly of the nuclear reactor is very important from operational safety point of view. Modern computer simulation techniques, like finite volume method [1,2] or finite element method [3], can be very useful in detail study of such processes, because after verification and validation processes [4] of computational fluid dynamics (CFD) model, you can relatively easily change boundary and initial conditions, or other input parameters of the model.

In our research, we focused on modeling and simulation of thermal-hydraulic processes in fuel assembly of nuclear reactor VVER 440, where the distribution of temperature field in coolant is investigated. All CFD analyses were performed by ANSYS CFX software [5], which computes basic thermo-hydraulic differential equations by finite volume method [6].

2. Geometric model of assembly

To perform thermal-hydraulic analysis of fuel assembly (FA) with six surrounding FAs of reactor VVER440, it is necessary to create 3D geometry model of coolant in the central FA and in six surrounding FAs, but also geometry model of inner space between individual FAs, which is called bypass, must be created. Reactor core topology with locations of individual FAs (marked as PK) and safe rods (marked as HRK) is shown in Fig.1.

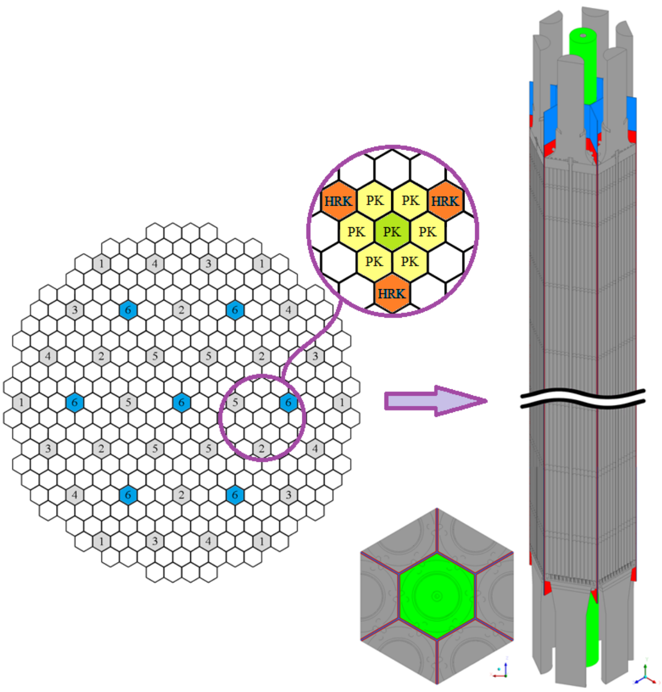

Fig. 1 *Investigated region – central FA with 6 surrounding FAs*

In this figure, investigated region of 7 FAs is shown in detailed circle. Central FA (green color in detailed circle) is modeled with all internal components, 6 surrounding FAs (yellow color in detailed circle) are modeled with symmetric boundary conditions and geometry simplifications, i.e. only half of surrounding FAs are considered and inner space between individual FAs is also modeled. Created 3D CAD geometry model is then used in mesh tool ANSYS ICEM CFD, where discretization of investigated region was performed. Due to size and complexity of model even with mentioned geometrical simplifications mainly in surrounding FAs it was necessary to replace fuel spacer grids in surrounding FAs by porous material (Fig.2) to shrink mesh size because of computational software and hardware limitations. Porous material parameters were calculated to numerically represent pressure losses and velocity changes in spacer grids [7]. Discretized model contains of over 126 million cells and over 132 million nodes (central FA itself has over 65 million cells).

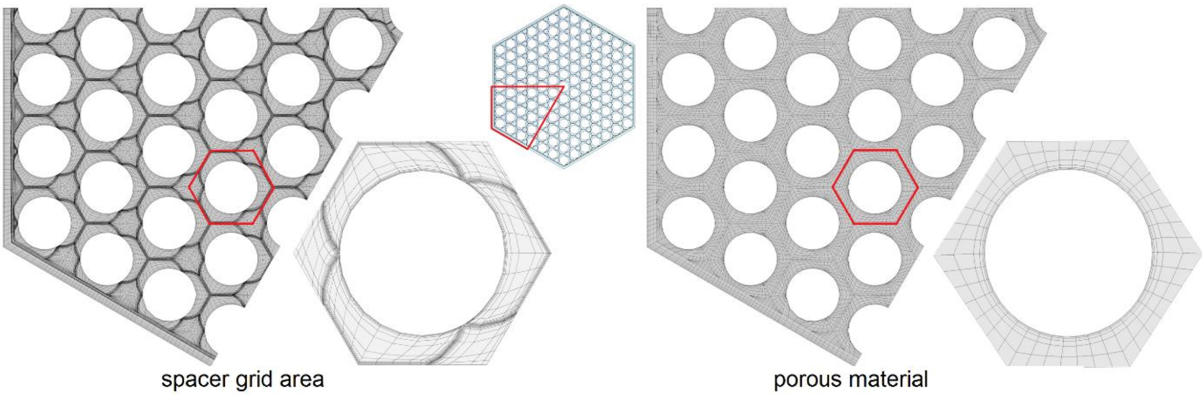

Fig. 2 *Mesh density comparison of spacer grid area and porous material*

3. CFD calculations and results

Steady-state CFD analyses of central FA with surrounding FAs were performed with CFD code ANSYS CFX with following conditions:

- coolant inlet temperature: all FAs 268 °C
- coolant inlet mass flow: all FAs 24.37 kg/s
- coolant output pressure: all FAs 12.25 MPa
- nominal thermal power in individual fuel rods: see Fig. 3
- turbulent model: SST
- material parameters of coolant: water from IAPWS-IF97 library

Presented CFD analyses were focused on influence of surrounding FAs thermal power on distribution of mass flow and coolant temperature in investigated central FAs. Thermal power of surrounding FAs was considered in rage 70-110% of nominal thermal power.

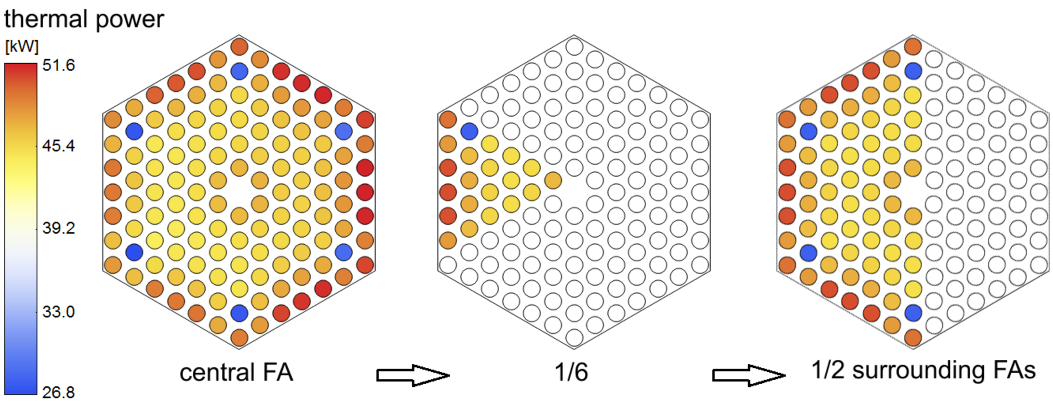

Fig. 3 *Prescribed nominal thermal power in central FA and surrounding FAs*

The distribution of temperature field in coolant at the output of investigated region for three different thermal power levels in surrounding FAs – 70, 100 and 110% of nominal thermal power (see Fig. 3) is shown in Fig. 4

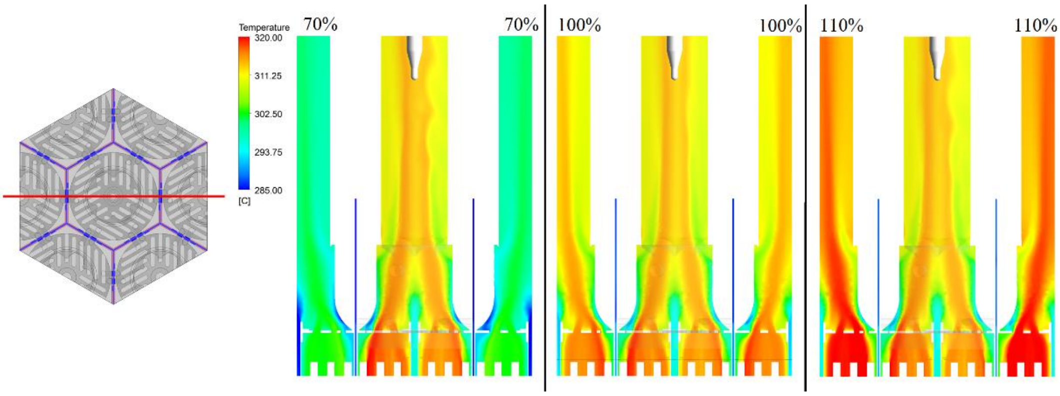

Fig. 4 *Distribution of temperature in vertical cut plane at the output of investigated region*

As we can see from this figure, the flow of coolant from bypass (thin region between individual FAs) influences adjacent FAs, but the coolant temperature in central FA is influenced by different thermal power of surrounding FAs only a little (see Fig. 5) – different thermal power in surrounding FAs causes different coolant temperature rise in bypass and different mass flow distribution between adjacent FAs.

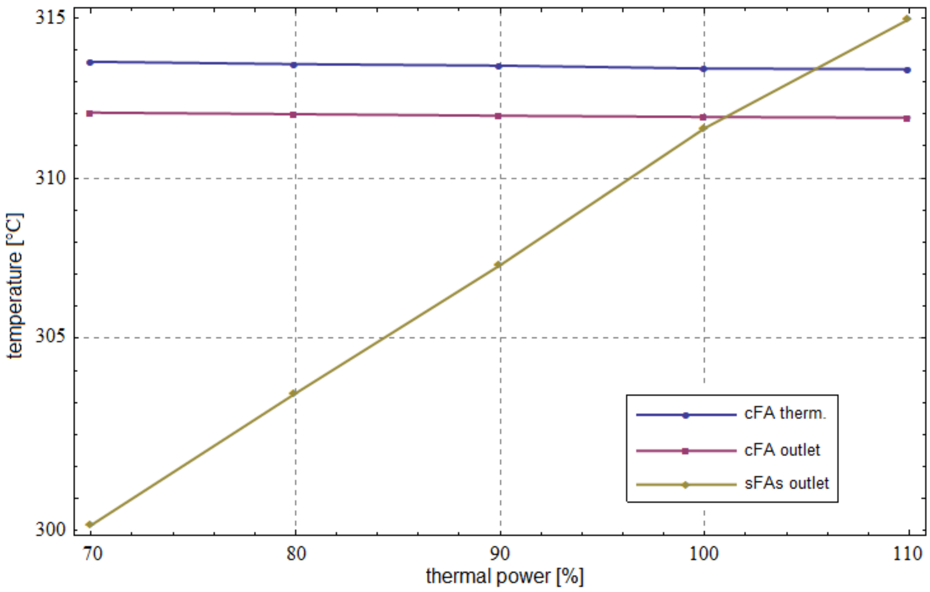

Fig. 5 *Average temperature dependence on thermal power of sFAs*

The distribution of temperature in different horizontal planes in central FA for different thermal power of surrounding FAs is show in Fig. 6. Nature of coolant flow in upper part of cFA remains very similar in all cases

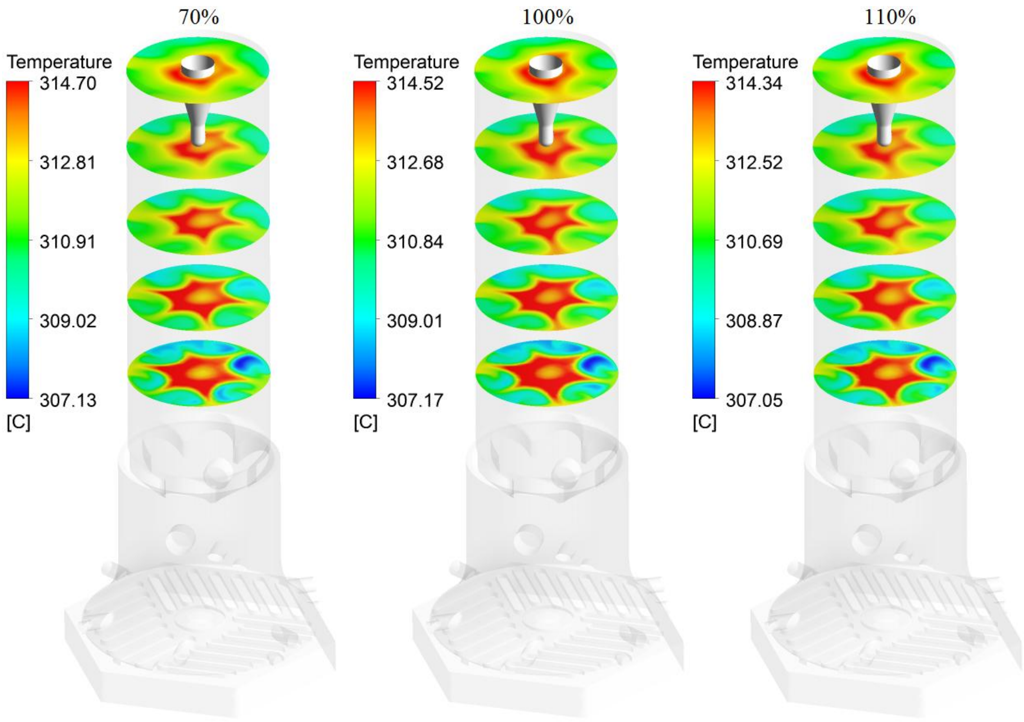

Fig. 6 *Distribution of temperature in central FA for three different thermal power of surrounding FAs*

Fig. 7 shows pressure losses in cFA and sFAs depending on prescribed thermal power of sFAs.

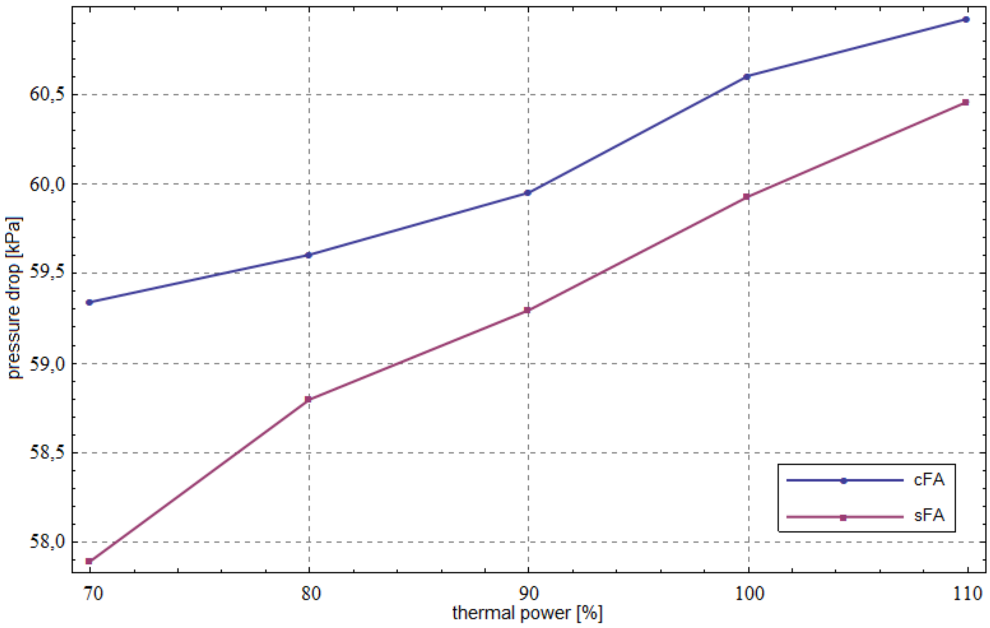

Fig. 7 *Pressure losses dependence on thermal power of sFAs*

Different pressure losses cause different bypass mass flows between cFA and sFAs what is shows Fig. 8.

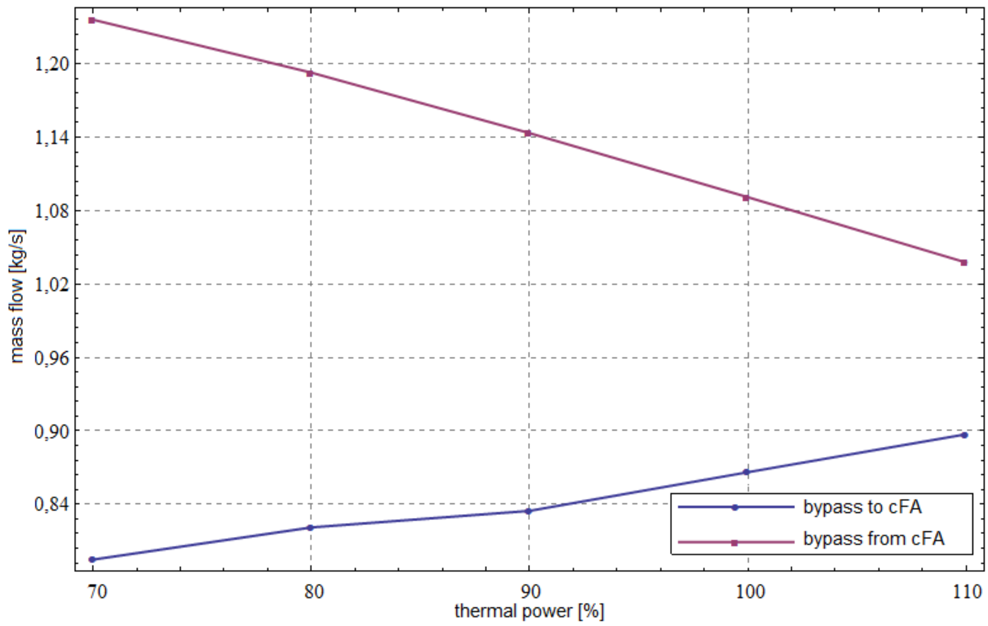

Fig. 8 *Bypass mass flow dependence on thermal power of sFAs*

Same pressure losses in cFA and sFAs were expected in equal thermal power (Fig. 7). Also same amounts of coolant were expected on bypass inlet and outlet to cFA in equal thermal power (Fig. 8).

Major problem of used CFD model is in its simplifications which cause slight differences in pressure losses comparing cFA and sFAs. this is caused by using just half of FA in surroundings FAs so the flow in upper has different character and second reason is used porous material which would be best to replace by full geometry of spacer grids if possible in software and hardware configuration.

4. Conclusion

The paper presents CFD modeling and simulation of coolant flow in fuel assembly of nuclear reactor VVER 440. The discretized model of coolant geometry contains over 126 million cells. To perform CFD analyses ANSYS CFX software was chosen. The influence of thermal power of surrounding fuel assemblies was investigated.

In the upper part of the central FA the coolant temperature distribution and average coolant temperature (registered by the thermocouple) from prescribed thermal power change of surrounding FAs shows only slight change. Great asset of this calculation are inner space coolant flow behavior and its parameters (i.e. bypass parameters) such as coolant mass flow and coolant temperature on inlets and outlets for central FA.

Even with mentioned uncertainties this CFD model gives great view on behavior of bypass flow between FAs. CFD model could be improved and used for future investigations in this field.

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. 1/0453/15. 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] Ch. Hirsch. Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics. Butterworth-Heinemann. 2 edition, 2007.
[2] T. J. Chung. Computational Fluid Dynamics. Cambridge Iniversity Press, 2002.
[3] J. Donea, A. Huerta. Finite Element Methods for Flow Problems. John Wiley & Sons, 2003.
[4] W. L. Oberkampf, T. G. Trucano. Verification and Validation in Computational Fluid Dynamics. Sandia National Laboratories, 2002.
[5] ANSYS CFX 15, Theory manual.
[6] H. K. Versteeg, W. Malalasekera. An introduction to Computational Fluid Dynamics, The Finite Volume Method, Longman Scientific & Technical, 1995.
[7] Daniele L. Marchisio, Federica Lince, Rajandrea Sethi Tiziana Tosco, Transport in Porous Media, vol. 96, no. 1, January 2013.