CFD model validation of coolant flow in nuclear reactor fuel assembly outlet by means of measurements on a physical model of a fuel assembly

Jozef Bereznai[1], František Urban[2], Ľubor Kučák[3]

[1] Slovenská technická univerzita v Bratislave, Strojnícka Fakulta, Ústav tepelnej energetiky, Námestie Slobody 17, Bratislava, jozef.bereznai@stuba.sk

[2] Slovenská technická univerzita v Bratislave, Strojnícka Fakulta, Ústav tepelnej energetiky, Námestie Slobody 17, Bratislava, františek.urban@stuba.sk

[3] Slovenská technická univerzita v Bratislave, Strojnícka Fakulta, Ústav tepelnej energetiky, Námestie Slobody 17, Bratislava, lubor.kucak@stuba.sk

Abstract Safe and effective loading of nuclear reactor fuel assemblies demands qualitative and quantitative analysis of the relations between the coolant temperature in the fuel assembly outlet, measured by a thermocouple, and the mean temperature of the coolant in the thermocouple plane position. In the laboratory at the Institute of Thermal Power Engineering of the Slovak University of Technology in Bratislava, an experimental physical fuel assembly model of the VVER 440 nuclear power plant with V 213 nuclear reactors was installed. The aim of the measurements on the physical model was to analyze the temperature and velocity profiles in the fuel assembly outlet. The following article deals with CFD simulations of a fuel assembly outlet, validated by means of measurements on the physical model of the fuel assembly

1 Introduction

Coolant flows along a 126 fuel rod bundle in the VVER 440 nuclear reactor fuel assembly. During the nuclear reactor operation, the temperature fluctuation is irregular in each single fuel rod, depending on the fuel properties and the position of the fuel assembly in the reactor. In the fuel assembly outlet the coolant temperature profile is irregular as well. In the fuel assembly outlet a mixing grid, which flattens the coolant temperature profile, is located. Subsequently, the coolant flows through the fuel assembly outlet where the cross section shape and area change. The next step in smoothing the coolant temperature profile is performed by a catcher. Coolant temperature in the fuel assembly outlet is measured by a thermocouple located in the fuel assembly axis, and is positioned 300 mm from the end of the fuel rods (fig. 1, plane 3). Measurements are provided at one point.

It is not possible to perform the analysis directly in the reactor, so it is carried out using measurements on the physical model, and the CFD fuel assembly coolant flow models. The CFD models have to be verified and validated in line with the temperature and velocity profile obtained from the measurements of the cooling water flowing in the physical model of the fuel assembly.

2 Fuel assembly physical model

In the laboratory of the Insitute of Thermal Power Engineering of the Slovak University of Technology in Bratislava a physica,l experimental copy of the fuel assembly model of a VVER 440 nuclear power plant was installed. The model was created to a scale of 1:1,125. The temperature and velocity profiles were measured with 2 combined probes located in 3 planes at the outlet of the fuel assembly in the physical model (fig. 1).

Because of the fuel assembly model design, the water pressure and the temperature during experiments, the velocity and temperature profiles mesurements are taken near the automodelling flow region, while the coefficient of friction λ is constatnt, independent of the Reynolds number. Ensuring the compliance between the size and physical similarity of the fuel assembly model and the actual reactor fuel assembly is achieved by ensuring the results of measurements on the model are in agreement with the coolant flow in the real fuel assembly outlet.

3 Measurements on fuel assembly physical model

The physical model of the fuel assembly is designed for modeling coolant flow in the fuel assembly outlet. By mixing heated (primary) water that flows along the fuel rod bundle with cool (secondary) water flowing from one of the triplet tubes, α, β, γ and/or the central tube, temperature discontinuity can be modeled. Temperature and velocity profiles are measured with 2 combined probes for each plane. Thirty seven measurements were made on the physical model at three different flow rates, with eight variants of isokinetic secondary water distribution to the fuel assembly model, and four different configurations of the mixing grid. In this paper we examine measurement with a water flow rate of 11,08 $kg.s^{-1}$ with secondary water being distributed through β triplet tubes located between the fuel assembly axis and the wall. Measured temperature values in plane 3 range from 39,67 °C up to 40,01 °C, a difference of 0,34 °C. By integrating temperature and velocity profiles the mean temperature of the coolant t_{str2} = 39,79 °C is calculated in plane 3 where the thermocouple is located. The temperature of the water is measured by probes A3 and B3 in the fuel assembly axis and are t_{A3} = 39,63 °C and t_{B3} = 39,72 °C.

4 Validation of the CFD fuel assembly model

The computational fuel assembly model has to be validated and verified for the purpose of further computing analyses. The objective is to achieve the maximum accuracy of the CFD simulations compared to real-time experiments. Hence suitable computational mathematical models were sought which would represent the physical phenomena of the experiment with minimal deviations.

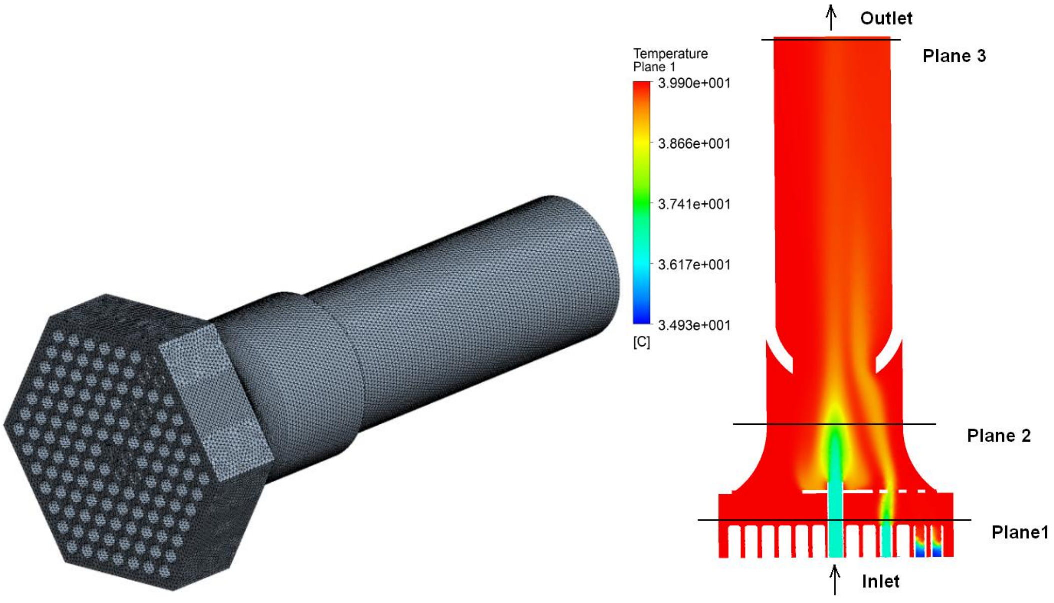

Fig. 1 Computational grid and temperature profile along cut of fuel assembly outlet

An analysis of three different, unstructured meshes was made: a coarse mesh grid with approximately 200 000 elements; a medium grid with 750 000 elements; and a fine grid with 2 000 000 elements(fig. 2). An adiabatic wall, with velocity inlet boundary condition and with pressure outlet boundary condition was used for the simulations.

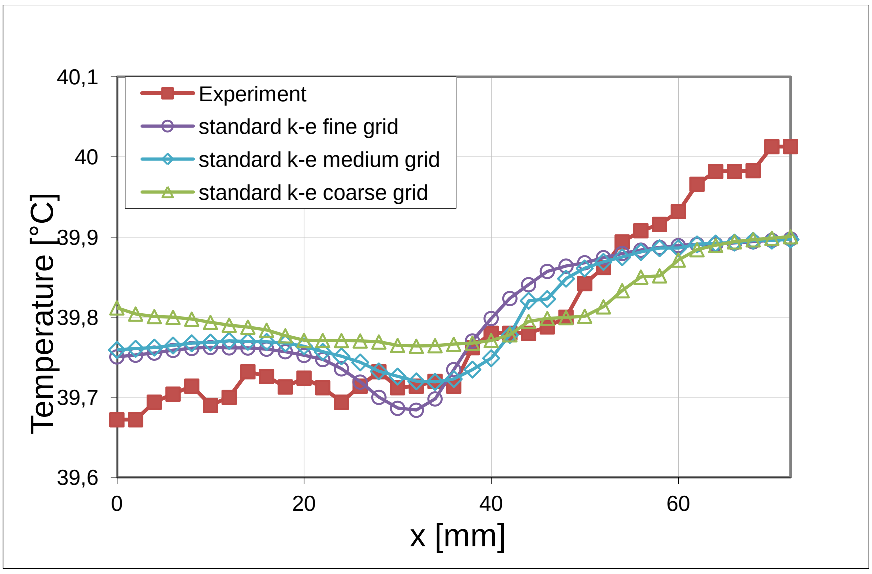

Fig. 2 Meshes comparison with calculated temperature profiles in plane 3 of fuel assembly outlet

For better correspondence of CFD simulations with the experiment, 4 mathematical turbulence models were tested - standard k-ε, RNG k-ε, standard k-ω and SST k-ω using the fine mesh (fig. 2). The results show large variations at the center of the cross section, with the standard k-ω and SST k-ω models matching the measured data more closely than the RNG model which showed a higher temperature drop at the center of the cross section. The greatest difference between the measured values is 0,34°C, and the difference between the calculated temperature values of the SST model is 0,20°C. The turbulence models were unable to match the temperature increase in last phase of probe traversation where the difference between the measured and calculated values is 0,11°C.

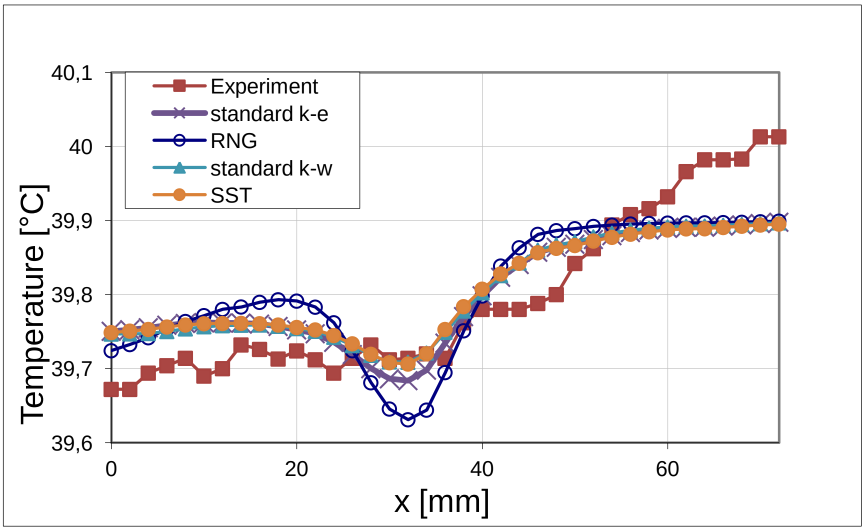

Fig. 3 Turbulence model comparison with calculated temperature profiles in plane 3 of fuel assembly outlet

An analysis of the influence of the inlet boundary turbulence intensity on the temperature profile in the fuel assembly outlet was made. Three levels of turbulence intensity were tested – low intensity (1%), medium intensity (5%) and high intensity (10%), all with the fine mesh. According to the analysis, the influence of different values of turbulence intensity has a minimal effect on the temperature profiles as shown in fig. 4.

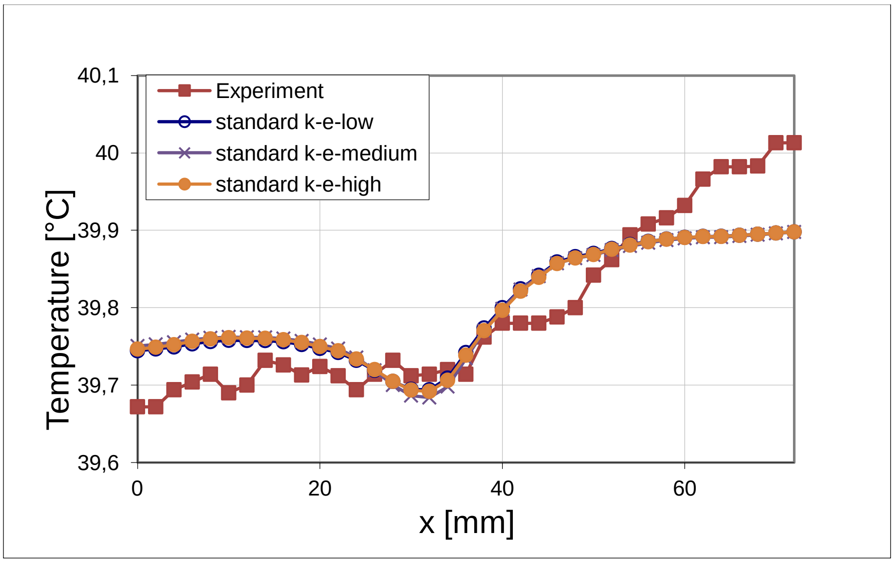

Fig. 4 Comparison of temperature profiles in plane 3 of the fuel assembly outlet calculated with different turbulence intensities in input boundary condition

5 Conclusion

In the laboratory of the Institute of Thermal Power Engineering of the Slovak University of Technology in Bratislava experiments were undertaken to determine the outlet coolant mixing on the fuel assembly physical model. CFD simulations and their validation are now ongoing. So far from the data acquired we can conclude that turbulence intensity change at the inlet boundary condition has minimal influence on the outlet temperature profile. Furthermore on the evidence of the turbulence models tested, the k-ω models match the experimental data more closely than the k-ε models. More effort will be put into investigating the size of the time steps, inlet boundary conditions,and other turbulence models.

Acknowledgement Preparation of this paper was supported by the Research and Development Operational Programme of the European Union's ERDF through the Centre of Competence in New Materials, Advanced Technologies and energy project, ITMS 26240220073.

Reference

[1] Muškát, P., Urban, F., Pulmann, M.: Merania na fyzikálnom modeli palivového článku jadrového reaktora. Strojnícky časopis = Journal of Mechanical engineering. 2008, č. 5-6, Roč. 59, s. 305-315 ISSN 0039-2472.

[2] Bereznai, J., Urban F., Kučák Ľ.: Influence of the mixing grid position on the coolant flow at the outflow part of the nuclear reactor fuel assembly physical model. XVIII. International Scientific Conference The Application of Experimental and Numerical Methods in Fluid Mechanics and Energy, Demänovská dolina, 2012, ISBN 978-80-554-0516-2.