THERMO-HYDRAULICS OF SPENT FUEL STORAGE POOL OF NUCLEAR REACTOR VVER 440

***Gabriel Gálik*[1], *Vladimir Kutiš*[1], *Jakub Jakubec*[1], *Juraj Paulech*[1], *Tibor Sedlár*[1]**

[1] *Department of Applied Mechanics and Mechatronics, IAM, Faculty of Electrical Engineering and Information Technology, Slovak Technical University in Bratislava*
E-mail: gabriel.galik@stuba.sk

Received 05 May 2016; accepted 14 May 2016

1. INTRODUCTION

The aim of this paper is to model the steady state coolant flow within the storage pool. The spent fuel storage pool is used for long term storage and cooling of spent fuel assemblies of a VVER 440 reactor and it is an important part of the refuelling process. Computational Fluid Dynamics solutions [1,2] were calculated using ANSYS CFX.

2. GEOMETRIC MODEL

The spent fuel storage pool consists of the main pool, two inlet pipelines located at the bottom of the storage pool, two outlet pipe lines located at the top, including the submerged storage grid and its support structures. The storage grid contains hexagonal absorbers around every fuel assembly storage location. The model contains a total of 290 spent fuel assemblies stored in all storage locations. Heat transfer through the solid walls of absorbers and fuel assembly shroud was not considered and these solids are not represented in the final model.

A number of details were simplified on the remaining structures that were deemed to have insignificant impact on flow profiles. Simplified fuel assemblies were used, where individual fuel rods in fuel assemblies were represented by an equivalent assembly model.

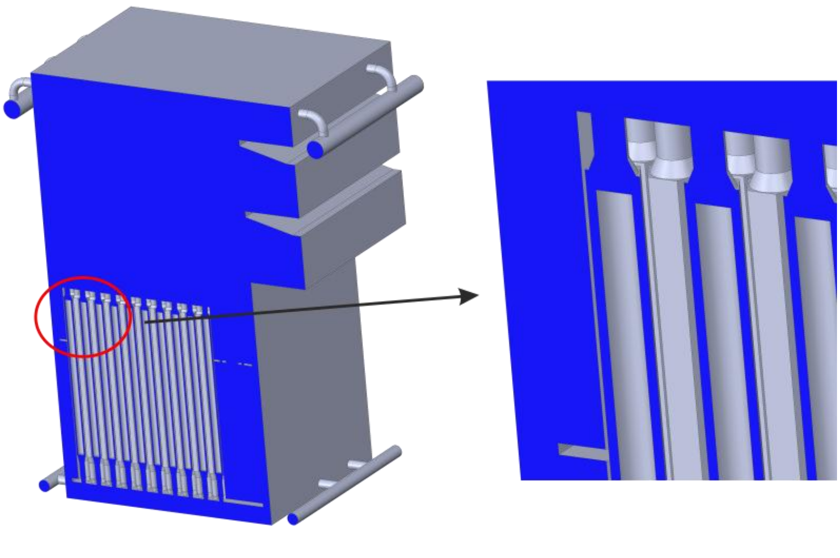

Fig.1: *Section of the final coolant volume model with a detailed view of fuel assemblies.*

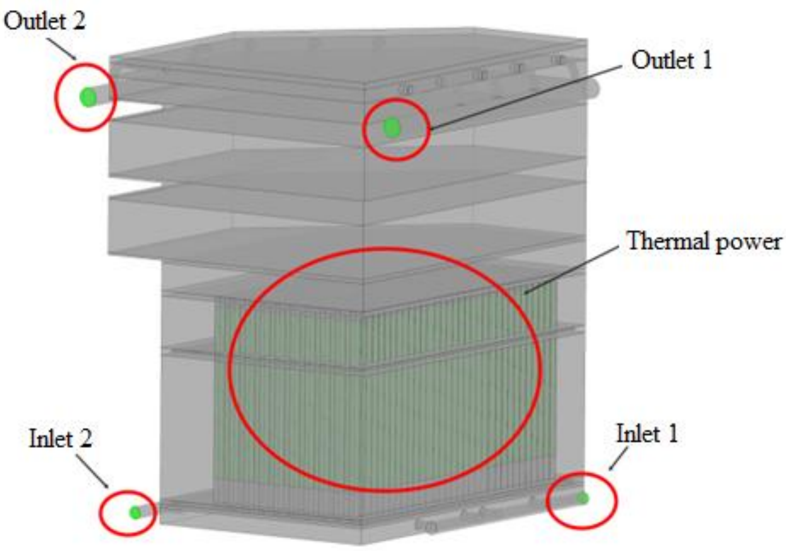

Fig.2: *Model and boundary conditions.*

3. BOUNDARY CONDITIONS

The total coolant level was 14.7m above the reactor zero reference level. Nominal pump properties were set for total inlet mass flow (Inlet 1 and Inlet 2) values at 90 kg/s. While the inlet coolant was injected at a constant temperature of 35 °C. The total remaining thermal power of all stored fuel assemblies was 1.85MWt this power represents the thermal loading after reactor refuelling. All remaining faces were set to a "No Slip Wall" boundary condition, except the top horizontal face, which was set to "Free Slip Wall" condition to simulate the properties of free surface.

4. MODEL DISCRETIZATION AND NUMERICAL MESH

The geometrical model was divided into multiple parts to ease the discretization process. The resulting divided model is shown in Figure 3.

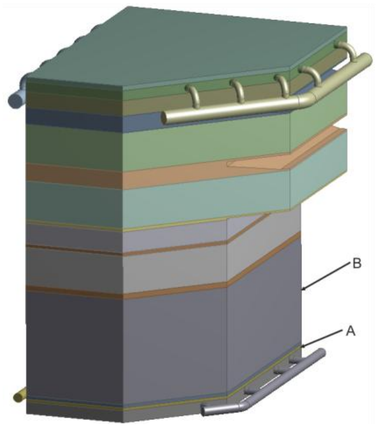

Fig.3: *Geometrical model division, location of parts A and B.*

The resulting parts were subsequently individually discretised using different meshing methods and elements. The total number of elements in the discretised model was app. 63.7mil., while the total number of nodes was app. 54.9 mil.

5. CFD SIMULATION

The goal of the CFD simulation is to determine the steady-state coolant flow in the spent fuel storage pool. Shear stress transport (SST) turbulent model was used during the solution process. The settled mean value of average outlet fluid temperature was used as the condition for steady-state solution. This condition was fulfilled after 2145 iterations.

6. SIMULATION RESULTS

The distribution of temperature and flow velocities were calculated for the whole fluid domain. However, for clarity, three different section planes were used through rows 1, 6 and 13 (as shown in Figure 4), to represent the spatial distribution of given flow properties. Additionally, outlet streamlines of 7 selected fuel assemblies were analysed to characterise the fluid flow structure within the storage pool.

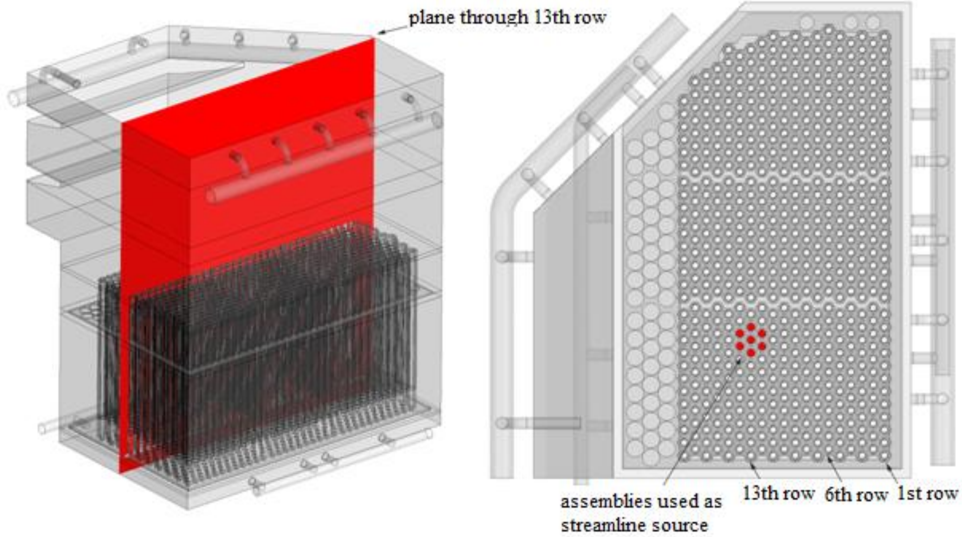

Fig.4: *Geometrical model division, location of parts A and B.*

Figure 5 shows the distribution of inlet streamlines with mapped flow velocities at the bottom of the pool. The mapped velocities show the approximate maximum flow velocity at the pool bottom to be 0.86m/s. Additionally, the distribution of streamlines characterises the induced vortices during flow interaction.

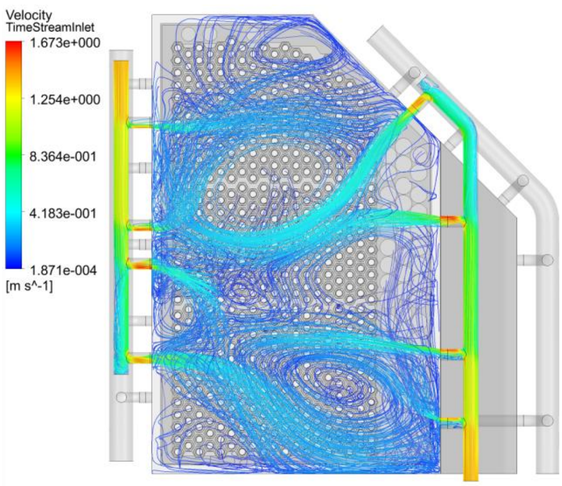

Fig.5: *Inlet streamlines with mapped flow velocity*

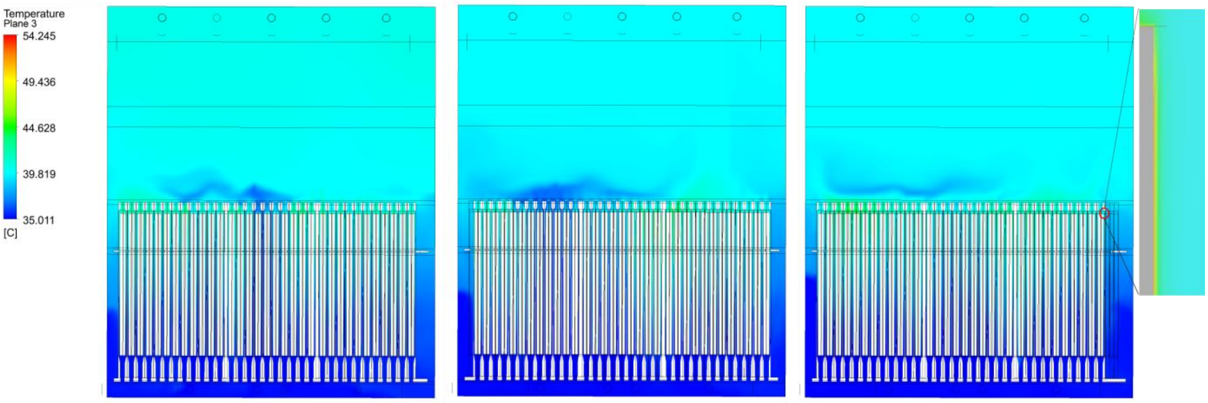

Fig.6: *Distribution of temperatures in the vertical section planes through stored assembly rows(1st 6th and 13th) with a detailed view of an equivalent fuel rod*

Figure 6 shows the distribution of coolant temperature in the vertical section planes through rows 1, 6 and 13 (shown in Figure 6). Despite the equally distributed thermal power, the coolant temperature in stored assemblies is non-uniform. This is caused by different mass flow rates through individual assemblies. Local temperature peaks are formed in layers that are in direct contact with the equivalent fuel rods, as shown in the detailed view of Figure 6.

7. CONCLUSION

Forced flow and buoyancy effects create comparatively low flow velocities. Such low flow velocities result in relatively large eddies, which cause the recirculation and mixing of the coolant fluid through fuel assemblies.

Acknowledgement

This work was supported by the Slovak Research and 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