TRANSIENT COOLANT MIXING SIMULATION DURING A SMALL-LOCA EVENT IN VVER 440 REACTOR

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

[1] *Institute of Automobile Mechatronics, Faculty of Electrical Engineering and Information Technology, Slovak University of Technology in Bratislava*

Ilkovičova 3, 812 19 Bratislava

E-mail: gabriel.galik@stuba.sk

Received 05 May 2017; accepted 21 May 2017

1. Introduction

The reactor pressure vessel is considered the most reliable component of pressurised water reactors. The target of concurrent research is the extension of operating life of existing power plants end their components. The condition of the reactor pressure vessel is a major limiting factor for the operating life of a power plant. The pressure vessel is exposed to thermo-hydraulic transients and the embrittlement effect caused by hard radiation. The coupled impact of these effects increases the risk of structural damage to the pressure vessel during high transients by pressure thermal shock (PTS). Thermal shock damage within solid materials represents high risk of structural weakening or in severe cases total structural failure and its elimination represents a significant engineering challenge.

2. Pressure thermal shock phenomenon

Loss of coolant accidents create highly transient processes within the reactor pressure vessel. The two properties that influence the vessel wall are pressure and temperature, both experience rapid changes during a thermo-hydraulic transient [1]. This makes it necessary to perform a time transient thermo-hydraulic analysis to be able to capture the dynamic loading of the RPV in sufficient quality. Given the unstable and non symmetrical nature of the coolant flow, the analysis must also include a model without symmetrical reductions that describes the RPV and the governing coolant flow characteristics within [2,3].

3. Transient CFD Analysis

As described above, a transient thermo-hydraulic analysis is necessary to capture the dynamic loading of the RPV during a Small LOCA event. The transient thermo-hydraulic simulation was set up to calculate coolant flow and mixing in the fluid domain and to calculate heat transfer at the RPV inner wall and the temperature field within its solid domain.

3.1. Computational model

The CFD model represents the fluid domain within the RPV and the solid domain of the RPV itself. Although, the RPV and fluid layers directly in contact with it are modelled in detail, internal structures and components have been significantly simplified. The larger structural components (i.e. reactor shaft, core barrel, reactor bottom etc.) are not directly modelled, only their shape is defined in the fluid domain. The structure of more complex components (i.e. fuel assemblies, perforations of reactor shaft and bottom etc.) were described as parameters of porous regions. The fully assembled model is shown in Fig. 1.

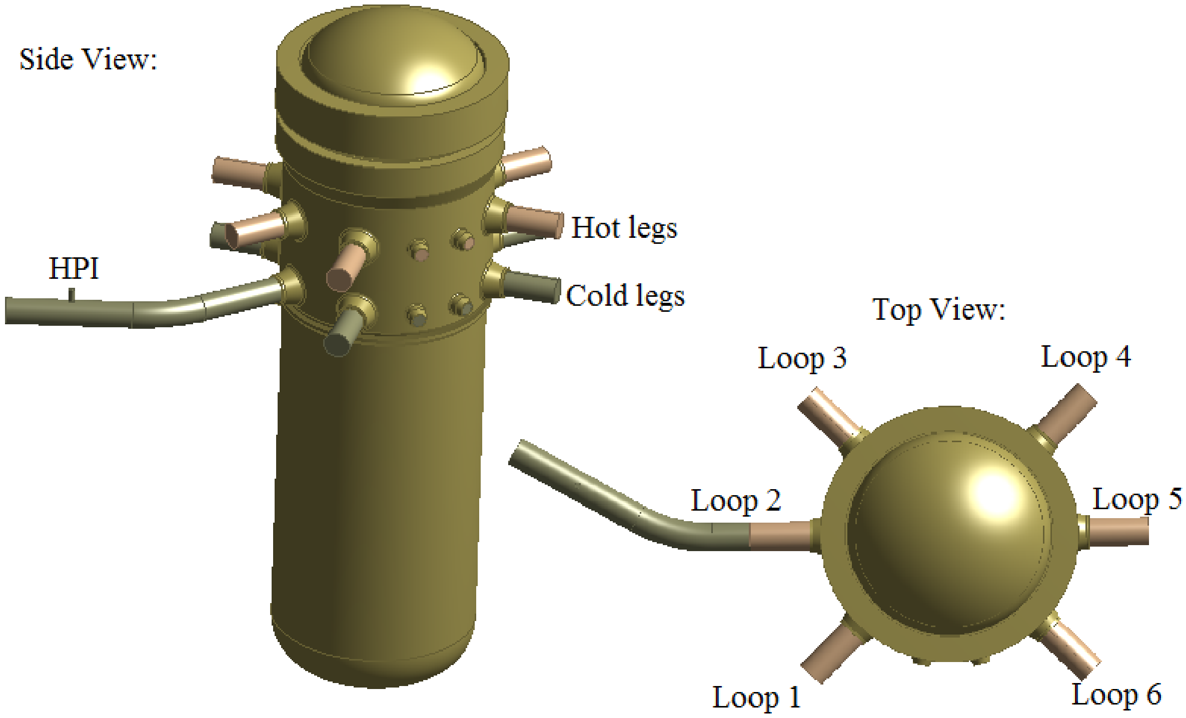

Fig.1:*Fully assembled CFD model.*

3.2. Analysis setup and Boundary conditions

The transient analysis simulates the initiation of high pressure coolant injection into the primary circuit cold leg. In the beginning of the simulation, the primary circuit is in nominal operational state. Water is pumped through the cold leg into the downcomer region by the main circulatory pump. Cold water injection is initiated by the decrease in pressure at the beginning of the simulation caused by a small break LOCA. Total simulated time 1700s. Specific parameters of the SB-LOCA case:

- Initial condition is standard operating state
- Break of ϕ20mm located in Loop 1 (outside of the modeled domains)
- All other Loops are considered undamaged
- Single HPI pump active on CL 2
- Main circulatory pumps and core shut down at t=0 s
- No phase change (water-steam) during event, no water level decrease in RPV

Boundary conditions were set up based on the specific case parameters and based on data acquired from a system level thermo-hydraulic analysis:

- On undamaged loops (e.g. 2,3,4,5,6) inlet mass flows on CL equal the outlet mass flow on HL and are specified based on data
- CL1 defined as pressure inlet (provides pressure information and also represents the leak), HL1 set up as mass flow from data
- Inlet temperatures on all CLs defined based on data
- Core remnant power defined from data

Fig.2 shows the location of individual boundary conditions.

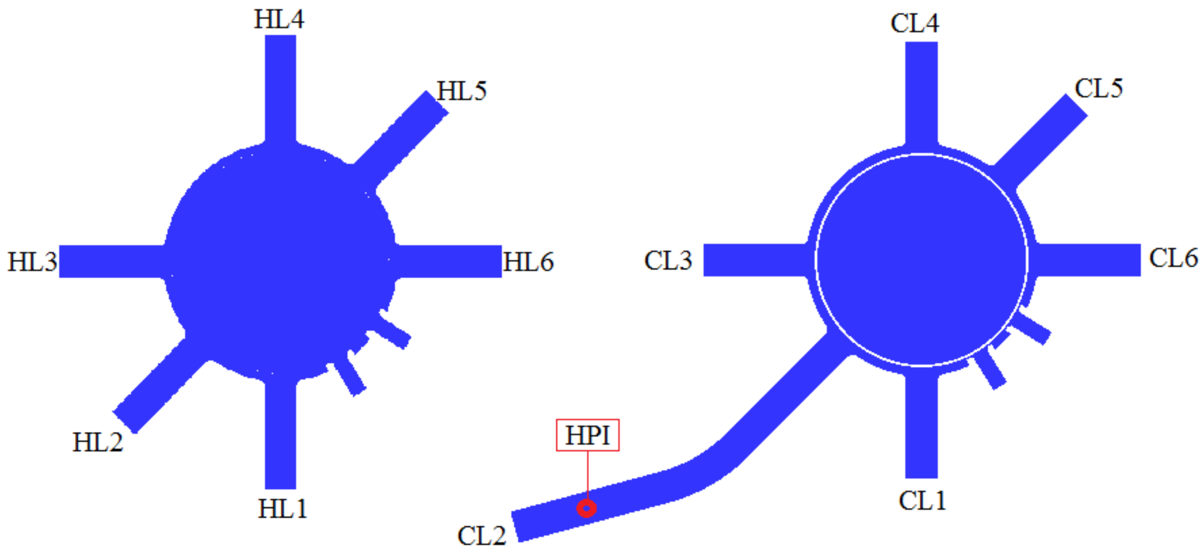

Fig.2: *Boundary condition locations*

3.3. Results

The above described analysis was solved in Ansys CFX on a High performance computing (HPC) cluster. The total solution time for the 1700s transient took 7days and 14 hours to solve. The final solution contains 270GBs of data. As such a large database cannot be fully included in this article, the following figures represent some of the most relevant data.

Fig.3:*Temperature distribution of RPV*

Fig. 3 shows the overtime development and change in the temperature distribution of the RPV wall inner surface. Fluid flow and mixing creates a strip cooling effect, the RPV is cooled in a long thin strip under the nozzle. Results show that this strip is also unstable and has a slight oscillation.

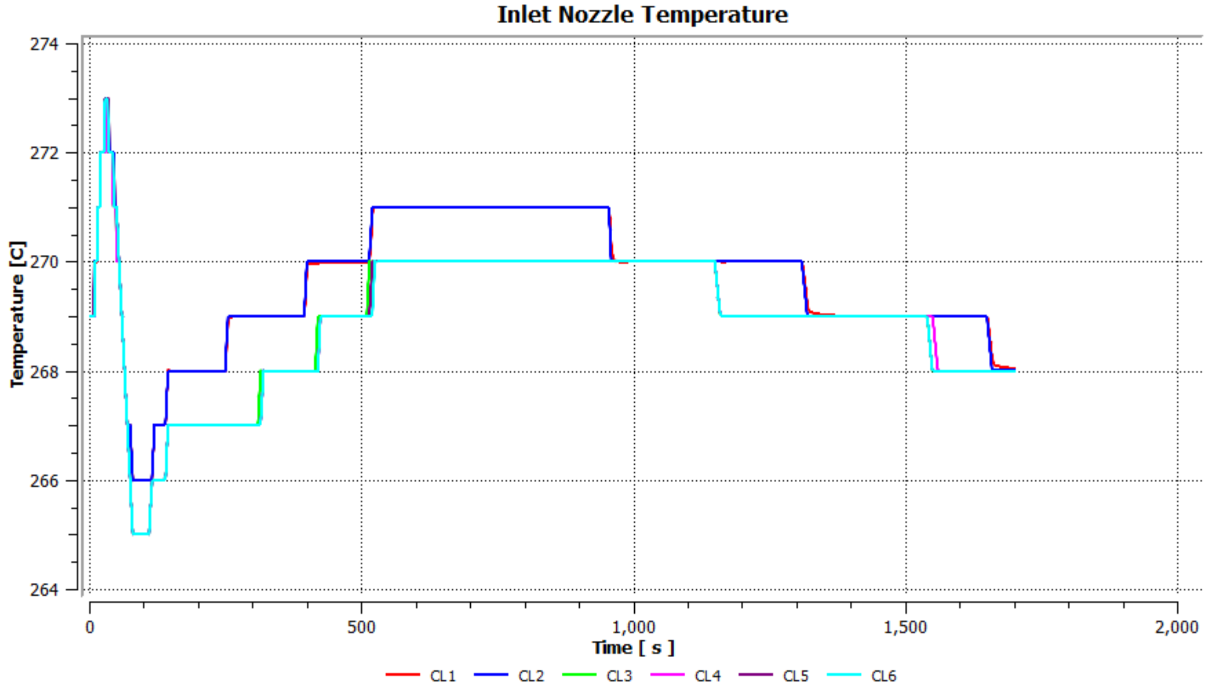

Fig.4: *Averaged coolant temperature in Cold Legs*

Fig. 4 shows the overtime development of inlet coolant temperatures on all six cold legs of the reactor. These inlet temperatures are given as parameters of the inlet boundary condition. The individual data series copy their values from the source data, which explains the discontinuous steps shown in Fig. 4.

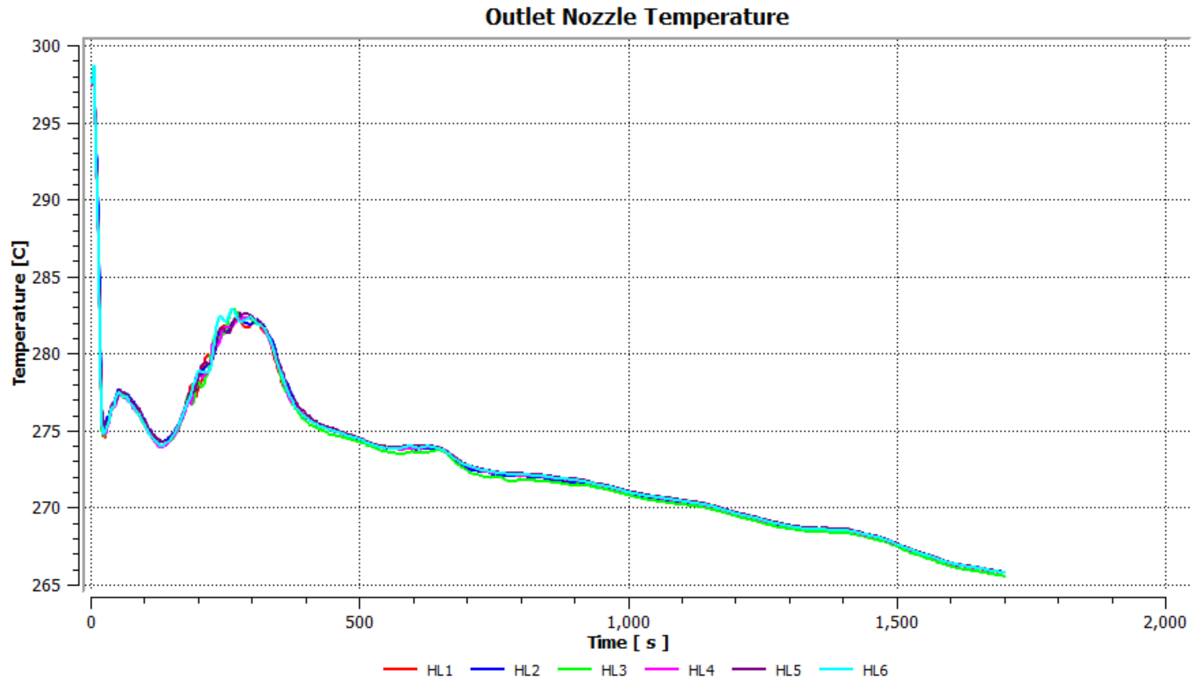

Fig.5: *Averaged coolant temperature in Hot Legs*

Fig. 5 shows the overtime development of outlet coolant temperatures on all six cold legs of the reactor. These outlet temperatures represent the coolant temperature as a result of coolant mixing and heating within the reactor.

4. Conclusion

As the results show the SB-LOCA event causes strip cooling of the RPV as expected. However, the cooling strip was shown to be unstable and to oscillate over time. This oscillation could result in cyclical loading of the RPV wall and its fractures [4,5]. Continued investigation into the cause of this instability and into its effects is needed.

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] Apanasevich P., Coste P., Ničeno B., Heib C., Lucas D.: Comparison of CFD simulations on two-phase Pressurized Thermal Shock scenarios. *Nuclear Engineering and Desing* Vol. **266**, Pages 112-128

[2] Quian G., Niffenegger M.: Integrity analysis of a reactor pressure vessel subjected to pressurized thermal shocks by considering constraint effect. *Engineering Fracture Mechanics* Vol. **112-113**, Pages 14-25

[3] Ferrara P., Araneo D., Moretti F., D´Auria: Development of a Finite Element Model of ATUCHA II NPP Reactor Pressure Vessel for Pressurized Thermal Shock Analysis, Nuclear Energy For New Europe 2008, Pages 702.1-702.10

[4] Wang Yanlong, Liang Shuhua, Xiao Peng, Zou Juntao: Experimental and simulation analysis of thermal shock with rapid heating followed by water quenching for CuW70 alloys. *Rare Metal Materials and Engineering* Vol. **41**, No. 3

[5] Carta G., Jones I.S., Brun M., Movchan N.V., Movchan A.B.: Crack propagation induced by thermal shocks in structured media. *International Journal of Solids and Structures* Vol. **50**, 2725-273