15th World Congress on Computational Mechanics (WCCM-XV)
8th Asian Pacific Congress on Computational Mechanics (APCOM-VIII)
Virtual Congress: 31 July – 5 August 2022
S. Koshizuka (Ed.)

TRANSIENT COOLING OF REACTOR VESSEL WALL DURING LOCA

GABRIEL GÁLIK[1], VLADIMÍR KUTIŠ[1], JURAJ PAULECH[1] AND VLADIMÍR GOGA[1]

[1]Department of Applied Mechanics and Mechatronics
Institute of Automotive Mechatronics
Faculty of Electrical Engineering and Information Technology
Slovak University of Technology
Ilkovičova 3, 81219 Bratislava, Slovak Republic
Email: gabriel.galik@stuba.sk - Web page: http://www.uamt.fei.stuba.sk

Key words: FEM, CFD, RPV, Fracture Mechanics

Abstract. *The article describes a multistage modelling methodology proposed by the author for the modelling of emergency core cooling processes. The methodology is based on the best practice guidelines presented by the IAEA, it is applied to a specific scenario of emergency core cooling during a loss of coolant accident [1] with an effective break diameter of 20mm. A 3D thermohydraulic analysis was performed as the first step in the solution process, where the transient changes in the pressure, velocity and temperature fields within the reactor pressure vessel were studied [2]. The primary knowledge learned when processing the results of the first step, was the presence of an oscillating cold coolant stripe in close proximity to the pressure vessel wall. The next step in the methodology consisted of a three-dimensional thermo-mechanical analysis of the reactor pressure vessel [3]. In this step, pressure thermal shock induced critical zones of mechanical loading were identified and the influence of the oscillatory character of the cold stripe on the pressure vessel was studied. The last step of the methodology consisted of a fracture mechanics analysis of postulated defects during the pressure thermal shock. Acquired results from the final step shown, that the postulated defects' sensitivity to the oscillatory nature of the cold stripe is highly dependent on the postulated defect's orientation.*

1 INTRODUCTION

The target of concurrent research is the extension of operating life of existing power plants and their components [2, 3, 4]. The condition of the reactor pressure vessel (RPV) is a major limiting factor for the operating life of a power plant [2]. The pressure vessel is exposed to thermo-hydraulic transients and the embrittlement effect caused by long time exposure to fast neutron radiation [4]. The coupled impact of these effects increases the risk of structural damage to the pressure vessel during pressure thermal shock (PTS) transients [3]. 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.

This article describes the part of the solution process of a multistage methodology. It describes the third stage, where the described simulation is based on the results of previous stages. While precursor stages simulated coolant mixing in a PWR (specifically WWER 440) during a simulated SB-LOCA transient and its direct impact on mechanical loading of the RPV.

The step described in this article focuses on determining fracture loading states in postulated defects.

2 DEFINITION OF THE STUDIED PROBLEM

The initial transient thermo-hydraulic analysis [5] simulated the initiation of high-pressure coolant injection into the primary circuit cold leg. In the beginning of the simulation, the primary circuit was in nominal operational state. Water was pumped through the cold leg into the downcomer region by the main coolant pump. Cold water injection was initiated by pressure decrease at the beginning of the simulation caused by a SB-LOCA. The above described analysis was performed using Ansys CFX on a High-performance computing (HPC) cluster. The main results of the above described analysis were pressure and temperature conditions on the reactor pressure vessel. Results of this analysis shown the formation of an unstable coolant stripe, that oscillated during the modelled scenario.

The second step consisted of a thermo- mechanical analysis with the goal of determining the RPV loading state during this scenario. This step was used to determine the critical stress intensity regions and the influence of the unstable cooling stripe on the RPV wall itself. The thermo-mechanical analysis consisted of three steady-state simulations representing the three chosen timepoints. The individual simulations represent loading state snapshots at given timepoints of the loading scenario. Figures 1 and 2 show the imported and mapped temperature (left), the calculated mechanical strain (middle) and calculated equivalent mechanical stress (right) in a longitudinal section showing the internal wall faces.

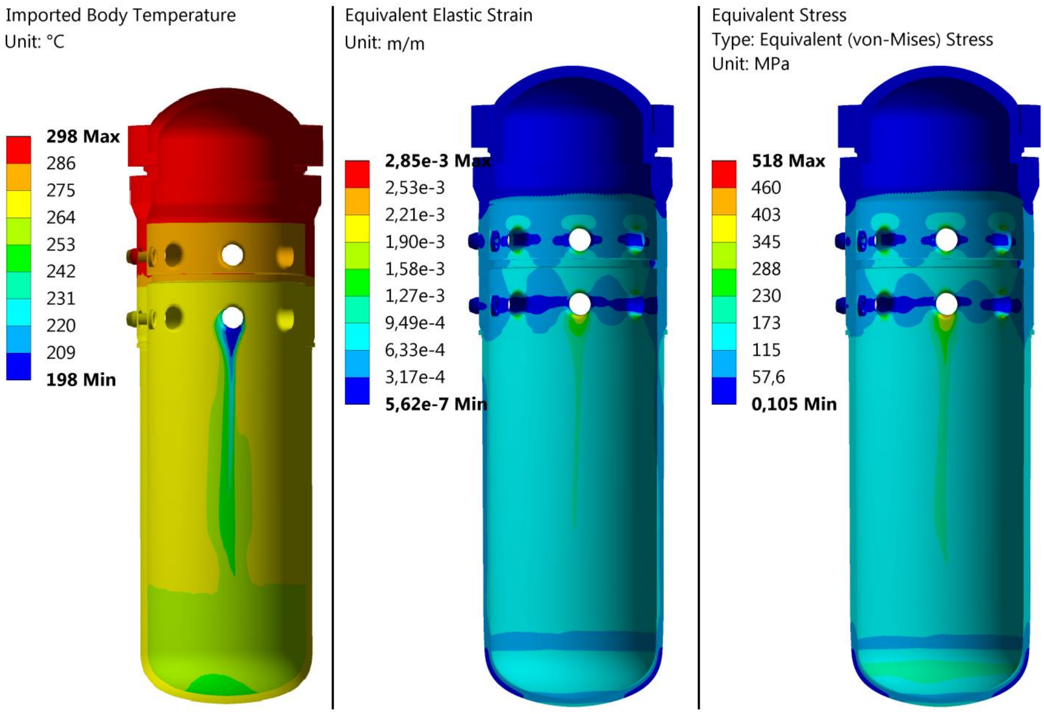

Figure 1: Results of thermo-mechanical analysis for timepoint 360s

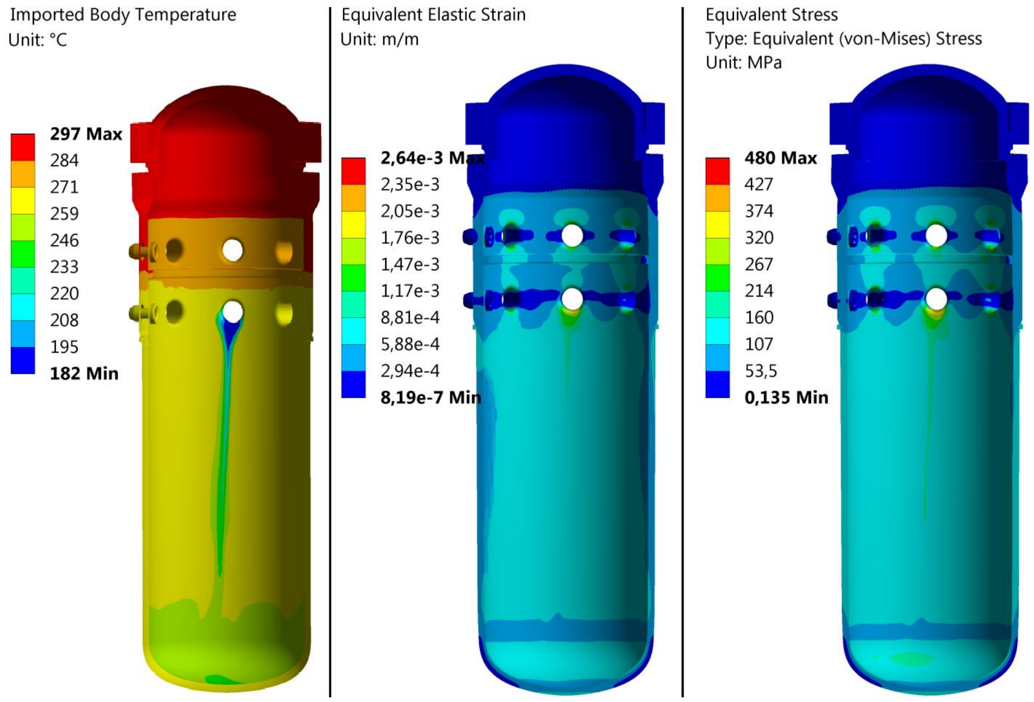

Figure 2: Results of thermo-mechanical analysis for timepoint 1053s

3 FRACTURE MECHANICS ANALYSIS

Numerical fracture mechanics methods are based on the concept of postulated defects, which usually are represented by elliptical or semi-elliptical cracks. These cracks need to be manually incorporated into the numerical model of the studied body. For the specific modelled case, there are best-practice guidelines for the location and type of postulated defects published by the International Atomic Energy Agency (IAEA)[8]. The numerical model was created using the geometric model of an RPV with a semi-elliptical fracture included with two orientations: axial and circumferential.

Figure 3 shows the mesh of the final numerical model with the two differing postulated defect orientations.

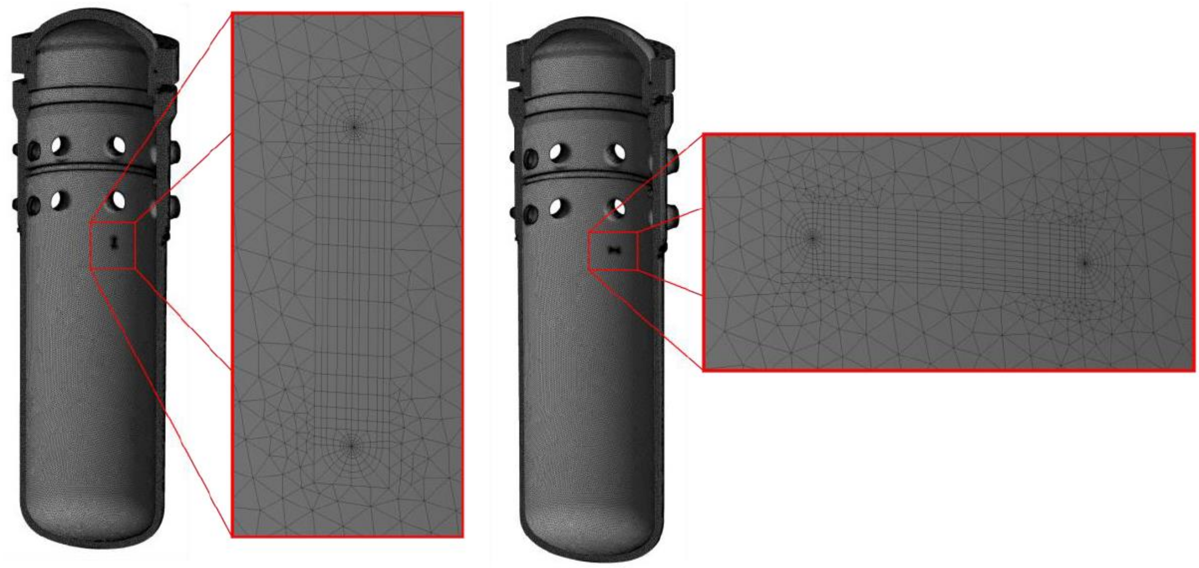

Figure 3: Tetrahedral mesh with postulated defects, axial (left) and circumferential (right)

Discretization of the semi-elliptical crack itself was identical for both cases, with 25 divisions along the crack front. Material discretization in the semi-elliptical fracture region is shown on Fig. 4.

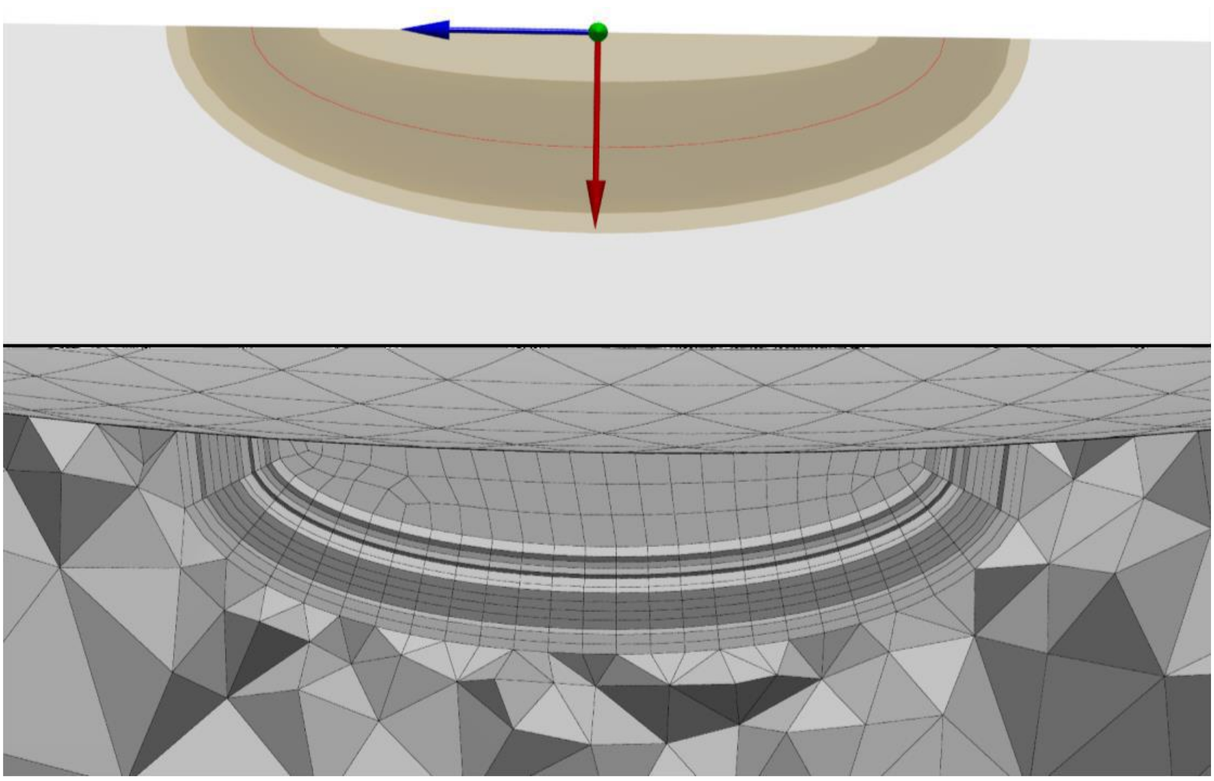

Figure 4: Schematic semi-elliptical fracture representation (top) and fracture discretization (bottom)

Basic parameters of the computational model and solver parameters used in simulations:

- model created using ICEM CFD, analysis performed in ANSYS APDL [6, 7] through ANSYS Workbench interface
- the final computational model contains 7.5 million nodes and 5.2 million elements
- the mesh is assembled from 10 node quadratic tetrahedral elements and in the fracture region with a combination of 15 node quadratic prismatic and 20 node hexahedral elements.
- RPV modelled with homogenous isotropic material, 15Cr2MoVA steel. Linear elastic material model with temperature dependent material properties.
- model assembled as a single solid domain

Figure 5 shows the mechanical boundary conditions defined for the thermo-mechanical analysis. The individual conditions are:

- A – RPV seating lip, fixed vertical degree of freedom
- B and C – cold leg connections, fixed translational degree of freedom in the axial direction, required for numerical stability.
- D – pressure conditions applied on all internal faces of the RPV, mapped from the results of the thermo-hydraulic analysis

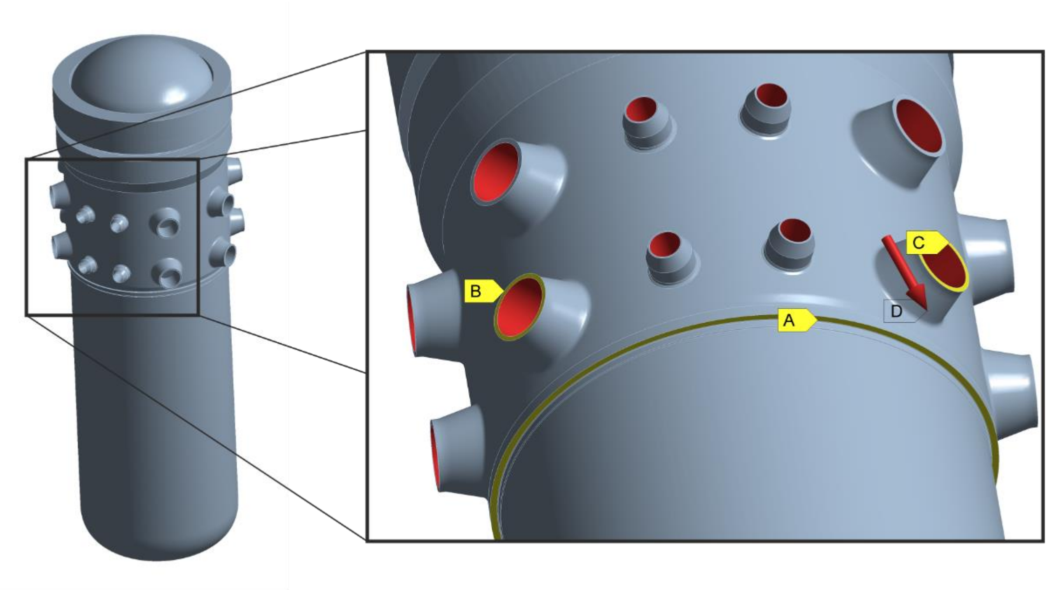

Figure 5: Boundary conditions

Imported data with temperature conditions were mapped on the whole model volume of the RPV. Temperature and pressure conditions were mapped on the RPV for three different timepoints. The total number of analyses performed was six.

4 RESULTS

The study comprised of six different steady state analyses, that corresponded to three different loading timepoints for the two corresponding fracture orientations each. The primary loading factors were pressure and temperature distribution imported from the previous stages.

Maximum stress intensities acquired for the individual cases are shown in Table 1.

Table 1: Maximum stress intensities

		Axial	Circumferential	
360s	K_I	1612,20	860,38	[MPa/mm^{0,5}]
	K_{II}	27,55	36,17	
	K_{III}	28,48	31,38	
	J	11886,00	3458,60	[J/m^2]
1053s	K_I	1406,40	1146,80	[MPa/mm^{0,5}]
	K_{II}	35,55	6,68	
	K_{III}	36,89	5,84	
	J	8040,00	6081,70	[J/m^2]
1218s	K_I	1335,70	1157,00	[MPa/mm^{0,5}]
	K_{II}	40,03	69,19	
	K_{III}	39,72	55,90	
	J	8155,00	6193,30	[J/m^2]

As is shown in Tab 1, the postulated defects are under tension in an opening mode, given by the significantly higher magnitude of KI in comparison with other components. While the above table shows a relatively simple relation between stress intensity maximums and time, the facture loading state must be evaluated with regard to its spatial distribution. Figure 6 shows the lengthwise distribution of stress intensity of the axial fracture from the three different loading states. As the chart shows, the stress intensity distribution is symmetrical along the crack front and it shows a declining trend over time. Figure 7 shows the lengthwise distribution of stress intensity of the circumferential fracture from the three different loading states. The stress intensity distribution in this case is highly asymmetrical and its maximum follows a sideways oscillatory movement, following the position of the cooling stripe.

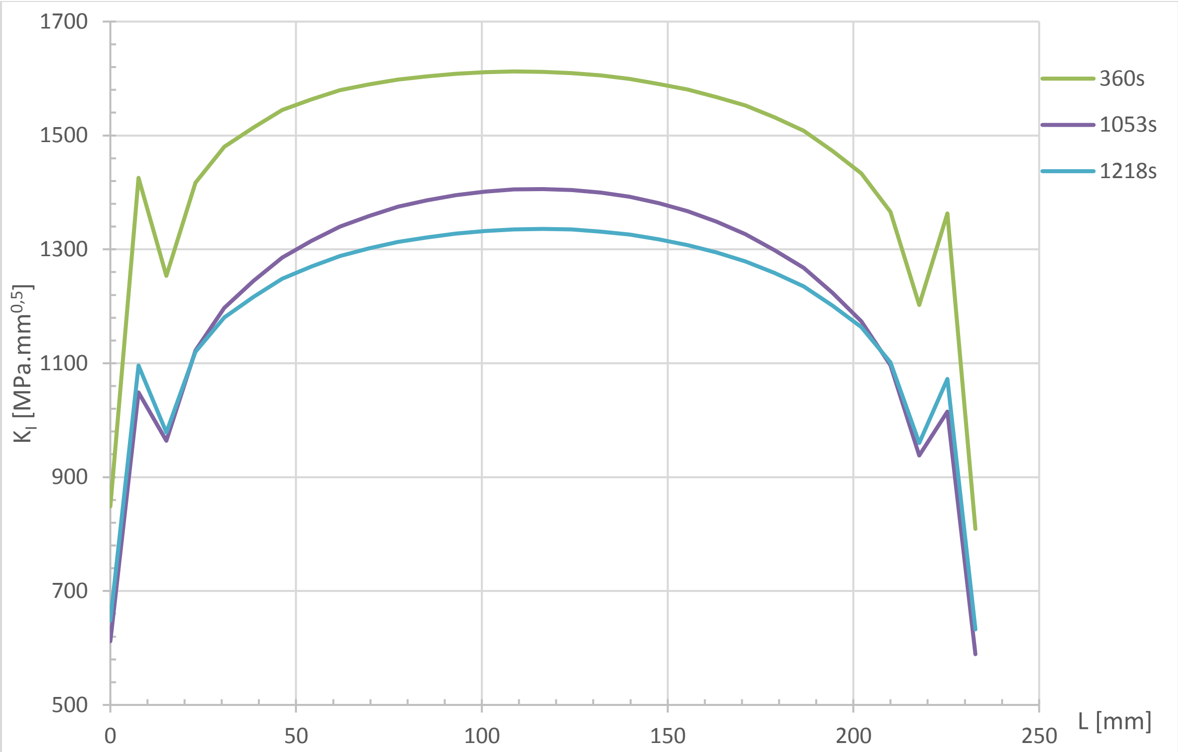

Figure 6: Lengthwise distribution of stress intensity of the axial fracture

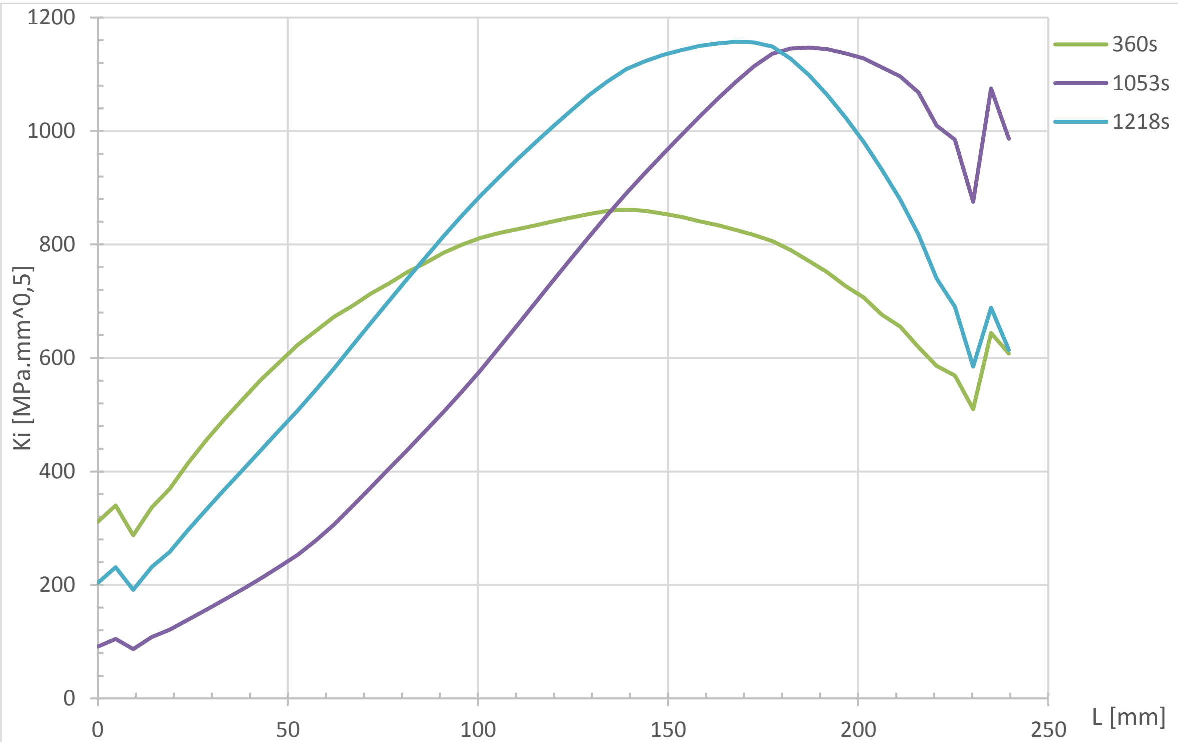

Figure 7: Lengthwise distribution of stress intensity of the circumferential fracture

CONCLUSION

In conclusion, the axially oriented fracture is minimally sensitive to the oscillatory character of the cooling stripe. The circumferentially oriented fracture on the other hand, shows significant sensitivity to the position of the cooling stripe in the form of stress intensity distribution along the crack front. While the local temperature determines the stress maximum, only one of the cases shown changes to stress distribution. This "wandering" maximum could result in cyclical loading alongside the crack front increasing fatigue, without showing significant changes in maximum stress intensity values.

ACKNOWLEDGEMENT

This work was supported by the Slovak Grant Agency: VEGA No. 1/0081/18, KEGA 011STU-4/2020 and APVV-19-0406.

REFERENCES

[1] G. F. Hewitt, C. L. Tien, Introduction to Nuclear Power.: Taylor & Francis, 2000.
[2] B. Yamaji, A. Aszódi, "CFD analysis of coolant mixing in VVER-1000 pressure vessel," Nuclear Engineering and Design, vol. 4, no. 238, pp. 872-889, 2008.
[3] L. Debarberis, a. Kryukov, d. Erak, yu. Kevorkyan, d. Zhurko, "advanced method for WWER RPV embrittlement assessment," international journal of pressure vessels and piping, no. 81, pp. 695-701, 2004.
[4] International Atomic Energy Agency, "Guidelines for prediction of irradiation embrittlement of operating WWER-440 reactor pressure vessels," IAEA-TECDOC-1442, 2005.
[5] Gálik G., Kutiš V., Paulech J., Murin J., "Modelling the effects of oscillating stripe cooling during a LOCA event in a VVER440/213 reactor pressure vessel," in 8th International Conference on Computational Methods for Coupled Problems in Science and Engineering, COUPLED PROBLEMS 2019, Barcelona, 2021.
[6] "ANSYS Mechanical APDL Technology Demonstration Guide," ANSYS, Inc., 2011.
[7] ANSYS, "Theory manual," 2014.
[8] International Atomic Energy Agency, „Guidelines on Pressurized Thermal Shock Analysis for WWER Nuclear Power Plants," 2006.