Paper ID: 435

Temperature Distribution of Reactor Pressure Vessel Wall for a Generic VVER 1200 Reactor

Mohammed Sarim Salman Karim[1], K M Rakib Al Hasan[1], Altab Hossain[2], Debashis Datta[1]
[1]Bangladesh Atomic Energy Regulatory Authority (BAERA)
[2]Dept. of Mechanical Engineering, Faculty of Engineering (MIST), Bangladesh Military Academy (BMA)
[1]Bangladesh Atomic Energy Regulatory Authority (BAERA)
E-mail: ssksarim@gmail.com; rkkhan1835@gmail.com; altab76@gmail.com; datta05@gmail.com

Abstract

The structural integrity and time-dependent degradation mechanism of the Reactor Pressure Vessel (RPV) of Water-Water Energy Reactor (VVER) - 1200 are the substantial part for the safe operation of a Nuclear Power Plant (NPP). Pressurized Thermal Shock (PTS) due to loss-of-coolant accidents (LOCA) transient is subject to a significant challenge to its structural integrity due to its high-pressure conditions and rapid temperature variations experienced by the RPV. Therefore, this study analyzes the 3D temperature distribution of reactor pressure vessel wall for LOCA induced single-phase pressurized thermal shock scenario for a generic VVER- 1200 reactor by ANSYS FLUENT. This scenario has the potential to be significant and impactful. The main conclusions drawn from the analysis is that in a LOCA scenario with stratified primary coolant and high pressure, the injection of ECCS results in two cooling plumes that begin to cool the vessel surface and can generate thermal stresses due to the abrupt thermal gradient between the colder ECCS water and the hotter inner wall of the vessel. These induced stresses can potentially exceed the material's fracture toughness, particularly if there are pre-existing flaws or cracks, leading to a risk of crack propagation and RPV failure. The model used in this study was a simplified symmetric VVER-1200, considering the material properties.

Keywords: Reactor pressure vessel, thermal shock, structural integrity, VVER

1. Introduction

During the assessment of the integrity of RPV, pressurized thermal shock (PTS) that can occur during emergency water cooling requires careful consideration. One of the most critical challenges that could potentially impact the RPV's structural integrity is referred as PTS (Ruan and Morishita, 2021). When a system with a higher temperature suddenly undergoes rapid cooling under high pressure, followed by re-pressurization, this event can potentially result in a thermal shock due to a temperature gradient. Furthermore, the increase in stress caused by the re-pressurization event can be referred to as pressurized thermal shock (PTS) [1]. For multi-dimensional effects analyzing in the downcomer region, CFD analyses is done by different simulation software e.g., OpenFOAM, STAR-CCM+, ANSYS etc. for more accuracy to predict the temperature and heat transfer in the RPV wall during thermal and mechanical loading. During the transient, ECCS injection is highly 3D complex mixing phenomena which requires 3D analyses [1]. CFD results for a hypothetical medium break loss of coolant accident (MBLOCA) happening in the hot leg were analyzed by Sharabi et al [2]. They pointed out how exceptionally time-consuming CFD models are. High-resolution CFD approaches like LES or DNS offer improved accuracy in resolving flow patterns and heat transfer, with examples of LES simulations is observed in Bieder and Giovanna Rodio [3-4] and a recent DNS example in [5]. To analyze temperature distribution of a potential PTS scenario created by LOCA event, a case has been carefully chosen from U.S. Nuclear Regulatory Commission [6]. ECCS injection into the primary coolant system is modeled by different CFD simulation (ANSYS FLUENT, ANSYS CFX, NEPTUNE CFD) and compared to validate the experimental setup of TOPFLOW-PTS [7]. Kral and Vyskocil used FLUENT

with URANS modeling to simulate a PTS scenario in the VVER-440 and VVER-1000 reactors. They subsequently compared these findings with the outcomes of their RELAP5 system TH calculations [8]. Main Steam Line Breaks (MSLB) are a significant risk factor for wall crack propagation. This study uses the Beaver Valley MSLB case-105 (BV case-105) as a reference scenario due to its severe temperature and pressure changes and potential for PTS. The goal is to analyze the impact of such a transient on a VVER-1200 reactor [9].

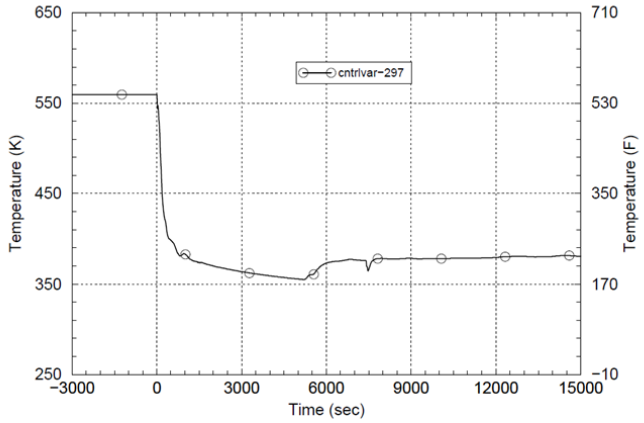


Fig. 1. Average Downcomer Fluid Temperature during Transient Scenario: LOCA- Main Steam Line Break from Hot Zero Power (BV Case 105) [6].

BV Case-105 involves a Main Steam Line Break from Hot Zero Power, tripping the Reactor Control Pumps (RCPs). Initial rapid drops in temperature and pressure are followed by a pressure rise to approximately 16.2 MPa. Downcomer fluid temperature reaches a minimum of 355 K at this pressure (Fig.1).

2. Computational Model and CFD simulation

During any transient event such as MSLB, the flow of hot primary coolant and cold ECCS water in the downcomer region as well as the inlet region is anisotropic. This flow requires turbulence modelling which is done in this present study. The temperature profile of fluid in downcomer region is analysis by computational fluid dynamics modeling. The primary fluid is heated by the nuclear reactor from the core and heat transfer is done by convection to reactor pressure vessel wall. ANSYS FLUENT solver is used to determine the temperature profile of fluid and vessel inner wall [10]. To solve the CFD analysis by ANSYS FLUENT, it is necessary to carry out Navier-Stokes equations and other relevant equations by fluent solver. The applied energy, continuity and momentum equation are showing in Equation (1) to (3):

Continuity Equation:

$$\rho\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)=0 \tag{1}$$

Momentum Equation:

$$\begin{aligned}\rho\left(\frac{\partial u}{\partial x}+\frac{\partial u}{\partial y}+\frac{\partial u}{\partial z}\right)&=\frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu\frac{\partial u}{\partial z}\right)-\frac{\partial p}{\partial x}\\ \rho\left(\frac{\partial v}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial v}{\partial z}\right)&=\frac{\partial}{\partial x}\left(\mu\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu\frac{\partial v}{\partial z}\right)-\frac{\partial p}{\partial y}\\ \rho\left(\frac{\partial w}{\partial x}+\frac{\partial w}{\partial y}+\frac{\partial w}{\partial z}\right)&=\frac{\partial}{\partial x}\left(\mu\frac{\partial w}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu\frac{\partial w}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu\frac{\partial w}{\partial z}\right)-\frac{\partial p}{\partial z}\end{aligned} \tag{2}$$

Energy equation:

$$\rho\left(u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}\right)=\frac{\partial}{\partial x}\left(\frac{k_f}{c_p}\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(\frac{k_f}{c_p}\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(\frac{k_f}{c_p}\frac{\partial T}{\partial z}\right) \tag{3}$$

The transport equations used to obtain the turbulence kinetic energy (k) and its dissipation rate (ε) for Standard k-ε model are as follows as Equation (4) and Equation (5):

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right) + G_k + G_b - \rho\varepsilon - Y_M + S_k \quad (4)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right) + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \quad (5)$$

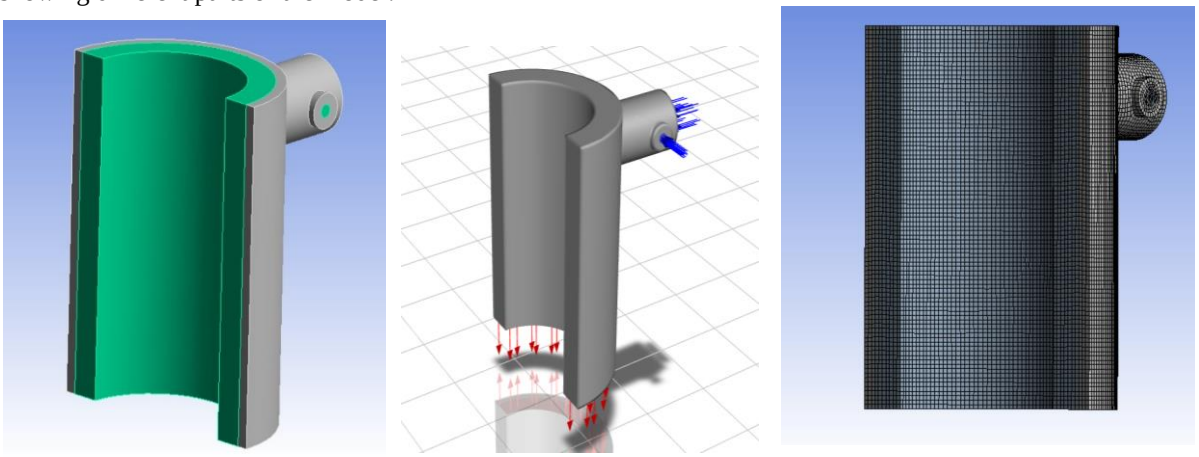
A simplified 3D model of a generic VVER-1200 reactor pressure vessel (RPV) has been designed using ANSYS Design Modular. The model includes the beltline region of the vessel, the cold leg, the ECCS injection region connected to the cold leg, and the associated primary and ECCS coolant. The various parameters of the RPV computational model used in this study are presented in Table 1.

In order to solve the turbulent flow considered in this research analysis, standard k-ε model is used in single phase fluid flow. The simulation is performed for 200 sec using Unsteady Reynolds Averaged Navier-Stokes (URANS) equations for time dependent solution by ANSYS FLUENT CFD simulation.

Table 1. Different parameters of generic VVER-1200 used for computational model [11]

Parameters	VVER-1200	Computational Model
Vessel diameter	4.232 m	4.2 m
Vessel height	11735 mm	6000 mm (Belt line region)
Thickness	197.5 mm	200 mm
Cold leg diameter	850 mm	850 mm
ECCS inlet diameter	225 mm	225 mm

The inlets of the system are defined as the inlet of the cold leg coolant and the ECCS injection. The outlet of the domain is set at the end of the downcomer fluid. In this 3D model, the hot leg is not considered. In the simulation, it is considered that the entire flow ultimately directed toward the downcomer. Fig. 2 (a) & (b) shows the simplified symmetric 3D model of VVER-1200 which is considered for simulation with detailed description showing different parts of the model.



(a) (b) (c)

Fig. 2. (a) Simplified symmetric 3D model, (b) direction of primary coolant, ECCS water and (c) meshed configuration of VVER-1200 RPV

During MSLB, the flow of hot primary coolant and cold ECCS water in the downcomer and inlet regions is anisotropic. This flow requires turbulence modeling, which is conducted in this study. The temperature profile of the fluid in the downcomer region is analyzed using CFD modeling. The primary fluid, heated by the nuclear reactor core, transfers heat by convection to the reactor pressure vessel wall. ANSYS FLUENT solver is used to determine the temperature profile of the fluid and the vessel's inner wall [10].

Mesh Configuration & boundary condition

For the mesh setup, the hex-dominant method is used for the CFD model. To achieve better accuracy, edge sizing is applied to the concerned nozzle region. The element order is selected as quadratic with a maximum size of 70 mm. The number of nodes and elements are 584,627 and 164,874, respectively. The orthogonal quality and skewness factor obtained during meshing are 0.78 and 0.32, respectively.
Pressure based solver is used for this transient CFD analyses with a time step size of 0.25 sec. The simulation is run for 200 sec. The low average CFL values for the simulation is kept below the 0.5 for the base mesh. In FLUENT setup, the velocity inlet- pressure outlet combination is used for boundary condition. The solver implemented a second-order upwind discretization method to solve the governing PDEs. The pressure-velocity coupling was evaluated using the SIMPLEC scheme, and a convergence criterion lower than 10^6 was employed. The CFD simulation is executed by ANSYS FLUENT considering the following computational setup given in Table 2.

Table 2. Main Parameters of initial and boundary conditions for CFD simulation

Parameters (Unit)	Value
Transient Pressure (MPa)	7
Inlet velocity of ECCS (m/s)	0.4
Inlet temperature of ECCS (◦C)	10
Inlet temperature (◦C)	298
Flow type of primary coolant	Stagnant
Void fraction	0
Inlet mass flow rate of ECCS (kg/s)	80
Transient primary coolant temperature (◦C)	298
Viscosity (Pa-s)	0.2
Coolant Density (kg/m^3)	1000

3. Result & Discussion

During PTS event, the non-uniform mixing phenomena of cold ECCS water and hot primary coolant is analyzed by 3D CFD simulation modelling with the help of ANSYS FLUENT. After simulation of 200 sec temperature distribution of inner vessel due to mixing of coolants is obtained. The study investigated the 3D temperature profile evolution along the downcomer and inner wall surface of the RPV during the coolant mixing event due to transient event. The ECCS water is injected into the primary circuit within the 225 mm diameter nozzle. From Fig. 3, it shows that the initial center line temperature of inlet nozzle of primary circuit which is around 298 ◦C or 571 K decreases with time. And after 200 sec simulations, it reaches to 167◦C or 440 K for cold (10 ◦C) ECCS water mixing. 3D temperature evolution of fluid at different time spans up to 200 sec is analyzed with the constant pressure of 7 MPa. No pressure drop & rise event is considered in this simulation approach.

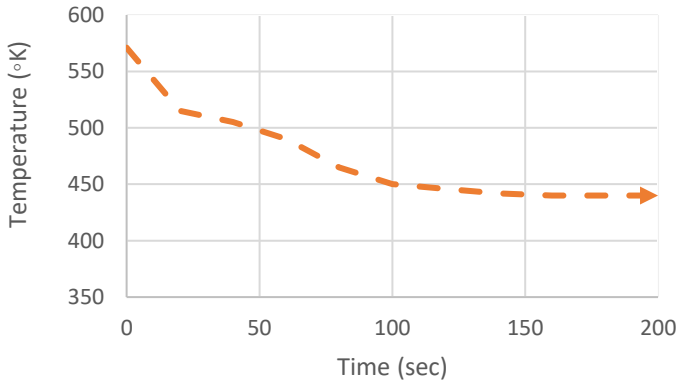
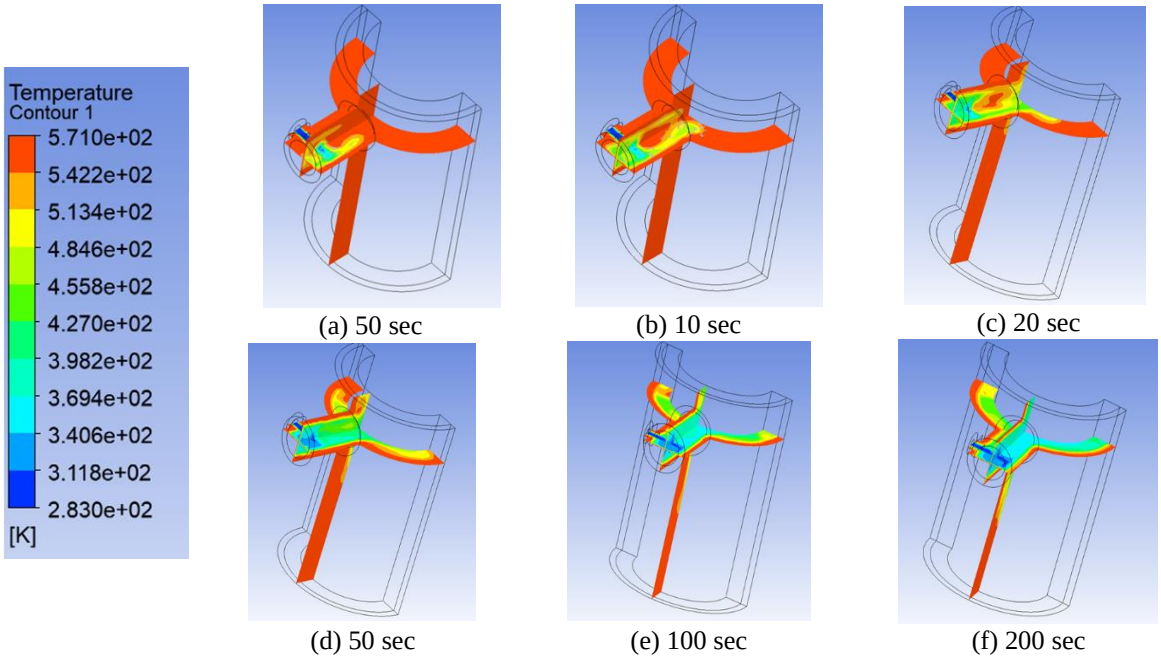


Fig. 3. Center line temperature of inlet nozzle region with respect to time (sec)

Cross sectional plane view at different time step of temperature contour is showed in Figure 4.2. The cold ECCS water entering into the primary loop with the velocity of 0.4 m/s and how it cools down the stagnant hot primary coolant in downcomer region can be demonstrated by Fig. 4. The minimum temperature can be found at the inlet of ECCS.

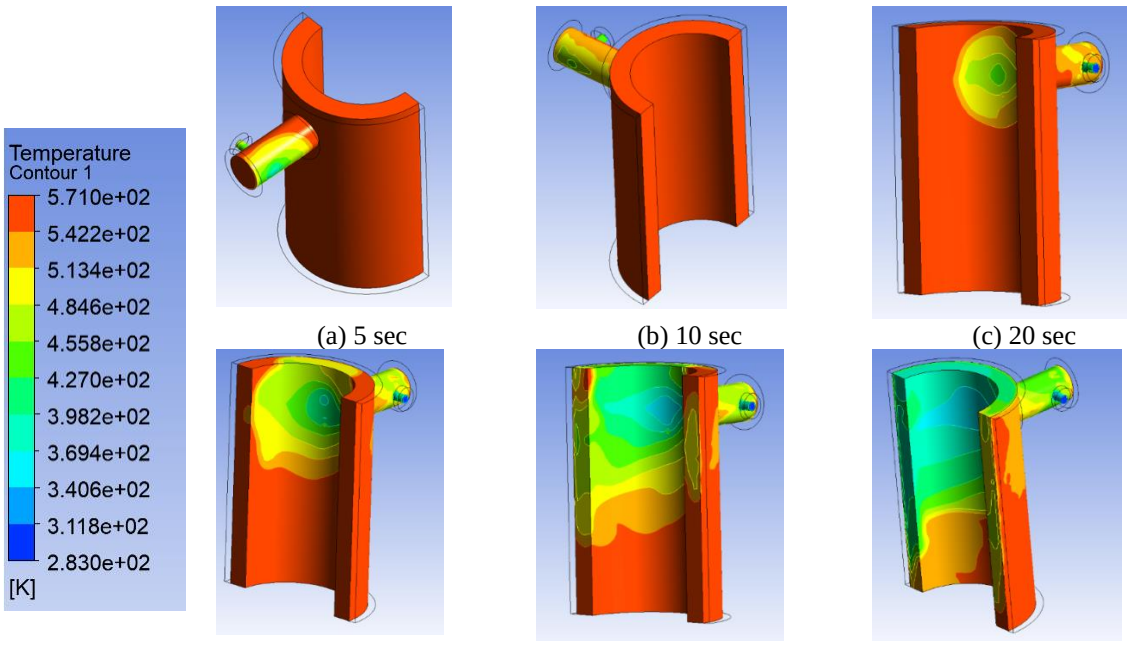


(a) 50 sec (b) 10 sec (c) 20 sec

(d) 50 sec (e) 100 sec (f) 200 sec

Fig. 4. Cross sectional plane contour at different time step

From Fig. 5, temperature development of the fluid region in downcomer and cold leg part due to ECCS injection up to 200 sec can be obtained from simulation results. The temperature distribution in fluid region is non-homogeneous. The lowest temperature obtained in this analysis is found to be 361.4 K or 88.4 ◦C after 200 sec.



(a) 5 sec (b) 10 sec (c) 20 sec

(d) 50 sec (e) 100 sec (f) 200 sec

Fig. 5. Temperature evolution contour of fluid profile at different time step

And the maximum temperature difference reaches at 210 ◦C at the fluid region. Temperature in the whole computational domain varies from 10 °C up to 298°C. Surface temperature of inner RPV wall is obtained in ANSYS FLUENT simulation. Temperature development in fluid part due to ECCS injection and mixing, the inner wall also undergoes a temperature change and experience thermal gradients. Fig. 6 shows the temperature evolution contour of RPV wall surface at different timesteps up to 200 sec. It is observed that up to 50 sec, the cooling effect is only shown at the cold leg part and nozzle area. Additionally, no cooling plume is experienced on the inner surface of the RPV. Two cooling plumes are observed on the inner surface of the RPV wall at 100 seconds and afterwards. Thermal gradients at the inner wall increase with an increase in ECCS injection mixing time and total mass injection. The temperature variation at inner wall surface also varies within 10 °C to 298°C.

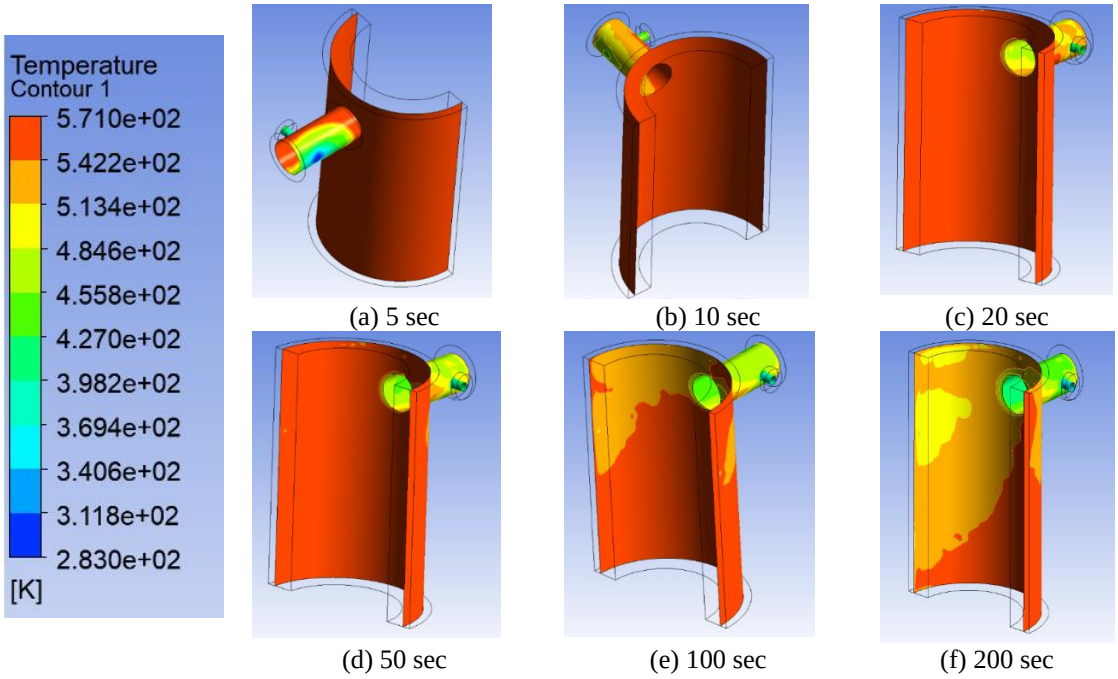


Fig. 6. Temperature evolution contour of RPV wall surface at different time step

4. Conclusion

This study investigated that temperature distribution of beltline region of RPV with the injection of ECCS water mixed with primary coolant for generic VVER 1200 of BV-105 case. The simulation results are aligned with the results from USNRC, NUREG/CR-6858, 2004. The 3D temperature distribution over time obtained from the simulation form the base of subsequent analyses of this PTS case. The analysis also reveals the presence of two cooling plumes around the nozzle area of the RPV wall, indicating the cooling pathway for the vessel wall. The study also demonstrates the non-uniform temperature distribution of the primary coolant. The rapid temperature drop at the vessel surface can introduce thermal stresses that may exceed the material's fracture toughness, especially if pre-existing flaws or cracks are present, increasing the risk of crack propagation and potential RPV failure. Consequently, understanding the cooling dynamics caused by ECCS plumes in a LOCA scenario is crucial for assessing PTS risk and evaluating the RPV's structural integrity under transient thermal loading. This relationship highlights the importance of ECCS behavior in predicting PTS events and enhancing reactor safety in accident scenarios.

5. References

[1] Uitslag-Doolaard, H.J., Stefanini, L., Shams, A. and Blom, F.J., "Numerical prediction of a single phase Pressurized Thermal Shock scenario for crack assessment in a Reactor Pressure Vessel wall", *Annals of Nuclear Energy*, 144, p.107563, 2020.

[2] Sharabi, M., González-Albuixech, V.F., Lafferty, N., Niceno, B. and Niffenegger, M., "Computational fluid dynamics study of pressurized thermal shock phenomena in the reactor pressure vessel", *Nuclear Engineering and Design*, 299, pp.136-145, 2016.
[3] Bieder, U. and Rodio, M.G., "Large Eddy Simulation of the injection of cold ECC water into the cold leg of a pressurized water reactor", *Nuclear Engineering and Design*, 341, pp.186-197, 2019.
[4] Loginov, M.S., Komen, E.M.J. and Höhne, T., "Application of large-eddy simulation to pressurized thermal shock: Assessment of the accuracy", *Nuclear engineering and design*, 241(8), pp.3097-3110, 2011.
[5] Lai, J.K., Merzari, E., Hassan, Y.A., Fischer, P. and Marin, O., "Verification and Validation of large eddy simulation with Nek5000 for cold leg mixing benchmark", *Nuclear Engineering and Design*, 358, p.110427, 2020.
[6] US Nuclear Regulatory Commission, "Technical Basis for Revision of the Pressurized Thermal Shock (PTS) Screening Limit in the PTS Rule (10 CFR 50.61)", Washington, DC, Report No. *NUREG- 1806*, Vol. 1, 2007.
[7] Apanasevich, P., Coste, P., Ničeno, B., Heib, C. and Lucas, D., "Comparison of CFD Simulations on two-phase Pressurized Thermal Shock scenarios", *Nuclear Engineering and Design*, 266, pp.112-128, 2014.
[8] Kral, P. and Vyskocil, L., "Thermal Hydraulic Analyses for PTS Evaluation: Comparison of Temperature Fields at RPV Predicted by System TH Code and CFD Code", *International Conference on Nuclear Engineering* (Vol. 51487, p. V06AT08A001). American Society of Mechanical Engineers, July, 2018.
[9] Arcieri, W.C., Beaton, R.M., Fletcher, C.D. and Bessette, D.E., "RELAP5 thermal Hydraulic analysis to support PTS evaluations for the Oconee-1, Beaver Valley-1, and Palisades nuclear power plants," *NUREG/CR-6858*, 2004.
[10] Manual, U.D.F., "ANSYS FLUENT 12.0. Theory Guide", p.67, 2009.
[11] IAEA, "Status report 108, VVER-1200 (V-491)", International Atomic Energy Agency, Vienna, Austria, 2011.