CFD Analysis of Thermo-hydraulic Behaviour in the VVER-1000 Reactor Using COMSOL Multiphysics

Karim Rashidov[1]*, *Guzal* Bobomurodova[2], *Diydor* Abdukhamidov[1], *Moxinur* Razzokova[3], *Utkir* Mustafoyev[3], *Renata* Monasipova[4], Amirbek Ikramov[5]

[1]National Scientific Research institute of Renewable Energy Sources, Tashkent, Uzbekistan
[2]Bukhara State Medical Institute named after Abu Ali ibn Sino. Bukhara, Uzbekistan
[3] Bukhara State Pedagogical Institute, Bukhara, Uzbekistan
[4]Gulistan State University, Gulistan, Uzbekistan
[5]Asia International University, Bukhara, Uzbekistan

Abstract. This article presents a two-dimensional Computational Fluid Dynamic (CFD) of the thermohydraulic behaviour of VVER -1000 reactor vessel based on COMSOL Multiphysics. The model works in a steady-state condition to examine the distribution of the flow of the coolant, along with temperature gradient and pressure differences in the reactor territory. Background conditions are specified as 515m^{2}s/inlet flow rate and inflow temperature set at 563.15 K and natural convection imposed on the exterior of the reactor wall in order to model natural convection with the environment. The two stationary simulations provide finer results in terms of temperature, velocity and pressure contours which helps one evaluate flow uniformity and heat transfer properties. Scientific outputs are read and calculated across three sampling cut lines and found to show the effect that coolant interactions have on both local thermal conditions and pressure stability. The result of this work will help identify the VVER-1000 reactor cooling performance better and make a mathematical basis of future three-dimensional and transient analysis.

1 Introduction

The world has applied nuclear power as one of the most consistent and sources of energy in large scale whose baseload electricity occupies low-carbon energy. The Russian VVER -1000 (Water -Water Energetic Reactor) is an example of a pressurized water reactor (PWR) system with a VVER core and designed to operate with maximum thermal efficiency and safety, among other types of nuclear reactors [1]. One of the major elements of the VVER- 1000 is a reactor pressure vessel (RPV) in which the core and the primary coolant are situated and under which the conditions of pressure and heat is large. The pressure and temperature distributions in the RPV need to be accurately predicted to determine thermal-

---

* Corresponding author : rashidovkarim0@gmail.com

hydraulic performance, structural integrity, and safety-level margin of nuclear power plants [2-3].

Computational Fluid Dynamics (CFD) has emerged as an essential forecasting instrument for intricate fluid flow and heat transfer phenomena in nuclear reactors. Both CFD can model thermal gradients, turbulence, and the flow behaviour in high spatial detail unlike traditional lumped-parameter codes or system codes, and the RPV falling into the category of high-speed aerodynamic components [4]. The advances in the turbulence model in the recent years including Reynolds-Averaged Navier-Stokes (RANS), Large Eddy Simulation (LES), and hybrid models have further increased the domain of application of CFD to reactor-scale simulations [5]. Besides, various high - performance computing materials have enabled in-depth scope of the modelling of the conjugate heat transfer between the coolant and structural entities [6].

In the VVER1000 reactor case, CFD analysis has aimed at the simulation of coolant mixing, thermal stratification, vibration caused by the flow and transient phenomenon of pressurised thermal shock in both regular and intermittent operating conditions [7]. These phenomena require accurate modelling since the core exit temperature, the pressure drops across the vessel, and the wall heat fluxes have a direct effect on the fuel performance and the system reliability. Indicatively, in the study carried out by Asmolov et al. [8] it was also proved that localized temperature peaks occurring around the core baffles may precondition the development of thermal fatigue and the material degradation over the long term. Likewise, Choiniere et al. [9] highlighted the topic of stagnation of flows which can cause subcooled boiling temperature and change the distribution of neutron flux.

Experimental validation is also a crucial aspect of CFD modelling in the nuclear applications. This has been achieved using an integral test facility like ROCOM (Rossendorf Coolant Mixing) and VVER mock-up that has supplied the necessary benchmark data to verify the theory of composing numerical predictions [10-11]. However, scaling and geometric simplifications are challenges to full scale to directly applying experimental data to full scale reactors [12]. Accordingly, mesh-independence investigations, choice of turbulence model and sensitivity investigations are essential to CFD model reliability assurance [13].

The study will conduct a reliable CFD simulation of pressure and temperature distributions in the RPV of a VVER-1000 reactor during a steady-state running with full-power conditions. We examine the high-fidelity of turbulence through ANSYS Fluent and axial and radial profiles of temperature, pressure, and flow velocity of important regions such as the downcomer, core barrel, and upper plenum. The modelling methodology includes the realistic conditions at the boundaries in terms of the data on the reactor operations, such as inlet mass flow rate, core heat generation rate, and wall heat flux distribution [14].

It is anticipated that the latter findings to this study will help to gain improved insight into thermal-hydraulic processes in VVER-type reactors and can serve as guidance to reactor designers, safety analysts, and operators of nuclear reactors. Moreover, the work supplements the current work to establish digital twins and AI-based monitoring of nuclear reactors by presenting certified datasets of CFD in real-time applications.

2 Materials and Methods

2.1 Computational Domain and Physical Model

A two-dimensional steady-state model of the VVER-1000 reactor pressure vessel was built with the use of COMSOL Multiphysics 6.1. The computational domain was the cross-section vertically containing the core region, downcomer and external wall boundaries. The geometrical arrangement was simplified intentionally to facilitate the numerical stability and to retain the salient thermo-hydraulic features that refer to coolant circulation and heat exchange. Therefore, the study was conducted at stationary (steady-state) conditions on the basis of assuming that the flow of turbulent incompressible water is in single-phase at high pressures.

2.2 Geometry, Boundary and Initial Conditions

The two-dimensional model was made, based on the accurate geometric ratios of the real VVER-1000 reactor pressure vessel that has a total height of 19 137 mm (almost 19.14 m) and exterior stimulus of approximately 4.55 m. These parameters were taken out of the design of regular VVER -1000 units. All major internal areas (fortunately, the elements included in the solar system) are included in the modelled area; these are the lower plenum, core of the reactor, downcomer, upper plenum, and the hemispherical dome. The maintenance of realistic geometry of the vessel and thus offers a correct forecast of the distribution of coolant flow, temperature increase, and pressure changes throughout down the reactor.

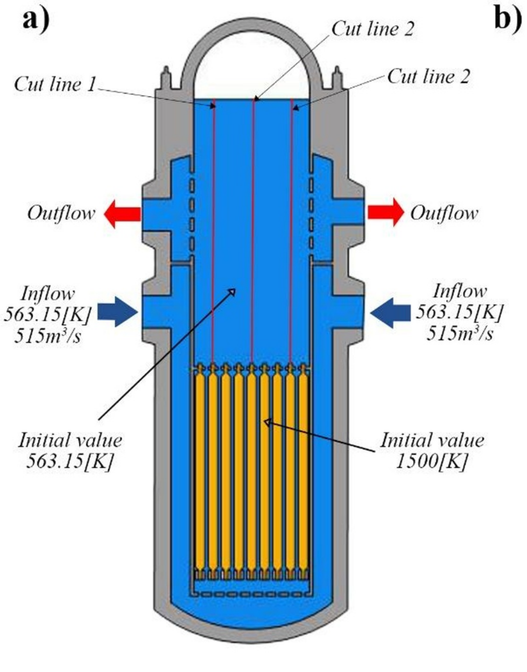

Fig. 1. a) 2D geometric view of VVER-1000 Reactor, b) generated mesh domains.

The mass flow rate over the inlet boundary was set at $515m^3/s$ and 563.15K initial temperature of cooling water. The reference to the constant value of a static pressure established in the outlet was 0 Pa gauge. On the outer wall, a natural convection was used to make the reactor vessel and the environment exchange heat. Internal walls were considered to be no-slip adiabatic boundaries, to simulate the correct wall-shear and development of the flows.

A reference mesh was created in COMSOL Multiphysics and a hybridized triangular element/quadrilateral form of the cell was used to provide an accurate solution to the complicated shape of the VVER 1000 reactor vessel. The meshing process utilized the Free

Triangular technique, which provided size control adaptivity in key areas such as at the inlet, outlet and the wall boundary.

When making up the final mesh, 1715842 elements were involved, with 135501 triangular and 36341 quadrilateral mesh elements, and 108095 mesh vertices. Also, the mean value of the quality of the elements determined using the skewness score was 0.8088 with the minimum quality of the element being 0.0191 thus revealing extremely good values of numerical stability in the case of steady use of thermal-fluid analysis.

Table 1. Mesh statistics of the 2D VVER-1000 reactor model

Parameter	Number of elements	Triangular elements	Quadrilateral elements	Mesh vertices	Total mesh area
Value	171 842	135 501	36 341	108 095	61.31[m^2]

The two-dimensional computational domain covered an area of 61.31 m^2 and the average ratio of elements was 2.58 -10/10=2.58, which has confirmed constant spatial discretization in the entire domain. An exercise involving mesh-independence was done by comparing a result of temperature and velocity fields at a series of increasingly finer meshes. The differences between the fine mesh and the nominal solutions were less than 1 per cent through which the use of the chosen mesh resolution is justified to offer a dependable trade-off between accuracy and computation speed.

3 Equations and mathematics

3.1 For Heat Transfer in Solids

Heat transmission in solid domains is articulated using the energy conservation equation, encompassing conductive heat flux, radiative contributions, and thermoelastic coupling [7], [8]:

$$\rho C_p\left(\frac{\partial T}{\partial t}+\boldsymbol{u}_{trans}\cdot\nabla T\right)+\nabla\cdot(\boldsymbol{q}+\boldsymbol{q}_r)=-\alpha T{:}\frac{dS}{dt}Q \qquad (1)$$

Under steady-state conditions, all transient terms vanish. The thermoelastic damping term accounting for thermal–mechanical coupling is given by:

$$Q_{ted}=-\alpha T{:}\frac{dS}{dt} \qquad (2)$$

where *d/dt* denotes the material derivative.

3.2. For Heat Transfer in Fluids

In fluid regions, the temperature field is governed by the convection–diffusion energy equation with pressure work and viscous dissipation:

$$\mathrm{\rho Cp}\left(\frac{\partial T}{\partial t}u\cdot\nabla T\right)+\nabla\cdot(q+q_r)=\alpha_p T\left(\frac{dp}{dt}+u\cdot\nabla_p\right)+\tau{:}\nabla u+Q \qquad (3)$$

The Cauchy stress tensor is decomposed as:

$$\sigma =- pI + \tau \qquad (4)$$

For ideal gases, the thermal expansion coefficient is defined as:

$$\alpha_P = \frac{1}{T} = \frac{1\partial\rho}{\rho\partial T} \qquad (5)$$

For steady-state and low-Mach-number flows, the pressure–work contribution,

$$Q_P = \alpha_P T\left(\frac{dp}{dt} + u \cdot \nabla_p\right) \qquad (6)$$

is typically negligible, while the viscous dissipation term is expressed as:

$$Q_{vd} = \tau : \nabla u \qquad (7)$$

4 Results and Discussion

The results shown in Fig. 2 exhibit the simulated steady-state distributions of temperature, velocity and pressure in the VVER-1000 reactor pressure vessel. The contour maps elucidate the basic thermo-hydraulic characteristics of the coolant that occurs under nominal operational conditions.

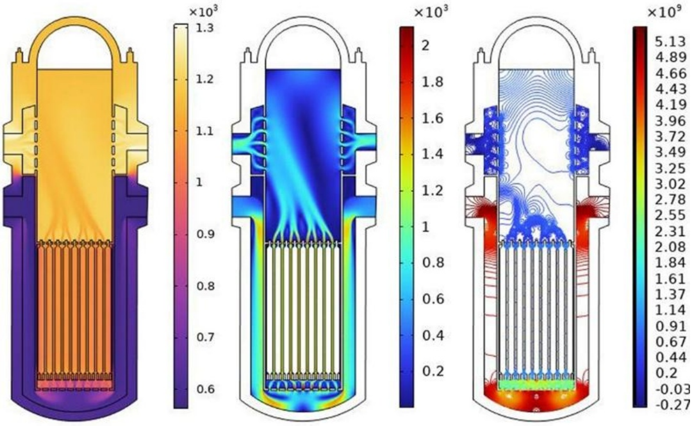

Fig. 2. Distribution of surface temperature, velocity and pressure.

In the temperature field (left panel), the coolant goes into the lower (plenum) with a temperature of about 563 K and gets gradually heated as it is propelled into the active core zone. Maximum temperature lies in an area close to the outlet of the upper core where the movement of energy between the fuel channels and the coolant is mostly developed. The temperature gradient is so gradual and symmetrical indicating the consistent removal of convectional heat and the even cooling of the core. The distribution of the velocity (middle panel) has shown a strong upward movement within the core, to an extent that that the maximum velocities are around 5 to 6 m/s. The acceleration in the flow is attained due to a movement of the coolant between the fuel assemblies, and the redistribution of the flow in

the upper plenum. Minor recirculation regions are observed near the inlet bend and dome which is a common feature in cylindrical reactors that are vertical and the effect of these phenomena does not cause any serious effect on the stability of flows globally. The pressure field (right panel) indicates that the pressure has a decreasing pattern towards the vessel bottom to the top that is in keeping with the required hydraulic gradient of the reactor. High Pressures of white powder are noted at the lower end of the plenum and the walls of the vessel, whereas the low pressure is noted at the outlet. Smoothly varying pressure gradient does prove reasonable convergence of the stationary solver and internal consistency of the energy balance of the model.
Fig. 3 shows the temperature field of the coolant in three spatially different cut lines fitted at different levels inside the reactor vessel; Cut line 1, at the lower part of the plenum, i.e., at its very bottom; Cut line 2, at the center of the core area; Cut line 3, at the upper part of the same plenum near the outlet.

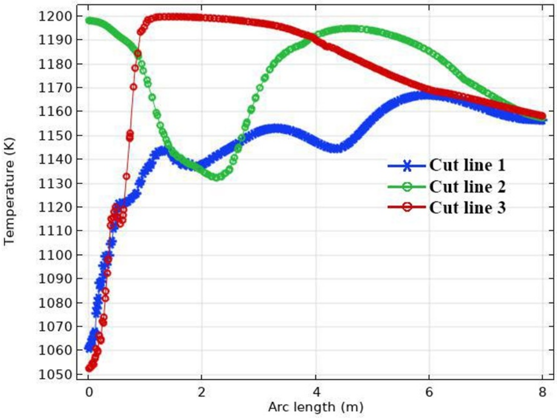

Fig. 3. Temperature variation of fluid at three different Cut lines.

The heat profile of the temperature condition denotes gradual rise between Cut 1 and Cut 3, which means that there is progressive heat absorption by the coolant as it rises up through the core region. At Cut 1, temperatures are 1050-1130K and the initial heating was when the coolant entered the lower plenum. The temperature in Cut 2 reaches about 1180K demonstrating strong gradients, which can be explained by intensive convective cooling of the active zone. In Cut 3, the coolant attains peak temperature of around 1200K, indicating that the heat extraction process is over before getting out of the reactor. Minute variations across the arc length are associated with local flow disturbance and sub-circulation events identified in the field of velocity (Fig. 2). The gradual increase of average temperature in the cut lines points to the stability and uniformity of coolant heating, which, in turn, determines the mechanism of working the system at the conditions of successful convective flow that does not include spots of increased heating.

Fig. 4 shows the magnitude of the velocity of the coolant across the three cut lines that were already demarcated in the reactor domain. The findings show that there are specific differences in the flow intensity of the lower, middle, and upper parts of the VVER-1000 reactor. At Cut line 1 (lower plenum), the fluid flow is the fastest achieving about 1400 m/s in the inlet zone as a result of a narrowed flow passage and high acceleration rate. As the coolant moves through the core entrance, its velocity gradually decelerates to approximately 800 m/s which is the moment of the beginning of the redistribution of flows between the fuel channels. In the mid-core region (Cut line 2), the flow has moderate velocities between 300 and 900 m/s, which is a steady equilibrium between the forces of inertia and inertial motion driven by buoyancy. It is characterized by various local peaks indicating the flow disturbances as a result of the channel geometry and local recirculation cells which are created around the core structures. In Cut line 3 (upper plenum), the speed at which the coolant moves slows down considerably to approximately 200–400 m/s, when the coolant is expanded and redistributed and then exits the reactor. The general tendency shows that the velocity of flow decreases steadily with the height of the reactor, in agreement with the presence of a pressure gradient and an increase in temperature (Fig. 2 and 3).

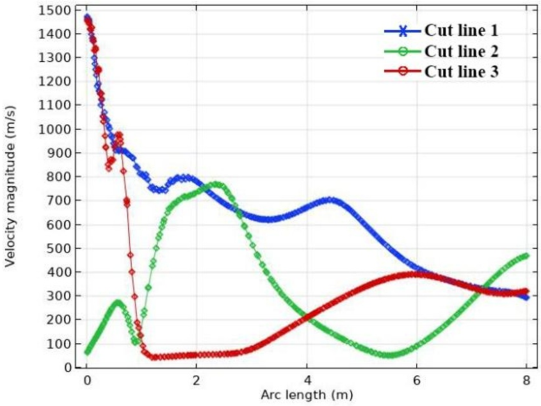

Fig. 4. Velocity variation of fluid at three different Cut lines.

Fig. 5 gives the representation of the pressure distribution of the coolant along three different cut lines in different elevations of the VVER-1000 reactor vessel. The discussion indicates a high level of stratification of the pressure that is consistent with the upward movement of the coolant and the effect of gravity. The fluid will achieve the lowest absolute pressure at cut line 1(lower plenum); near the inlet, the local flow characteristics are overridden by the dominance of dynamic acceleration. When the coolant rises in altitude, the pressure has a slight increase, which can be explained by the narrowing of the flow in the vicinity, and then a plateau in the middle. At cut line 2 (mid-core zone), in which the coolant interacts with the core structures, the pressure fluctuates at around 6.0 x 10 8 P a with small losses occurring as a result of flow rearrangements between the fuel channel. These minor oscillations on the arc length are manifested by the fact that there is some little turbulence and local velocity variations, as also shown in Fig. 4. The pressure slowly increases at the cut line (3) (upper plenum) to a point of about 8.5 x 10^8 P a, which signifies that there is a smooth transition to the outlet region. This increase is linked to the growth of the coolant and the resultant decrease in resistance in flow in the dome area. The total change proves the fact that the pressure drop between inlet and outlet values do not exceed the expected design value (approximately 0.2Mpa) in order to perform the steady-state work of VVER-1000 reactor.

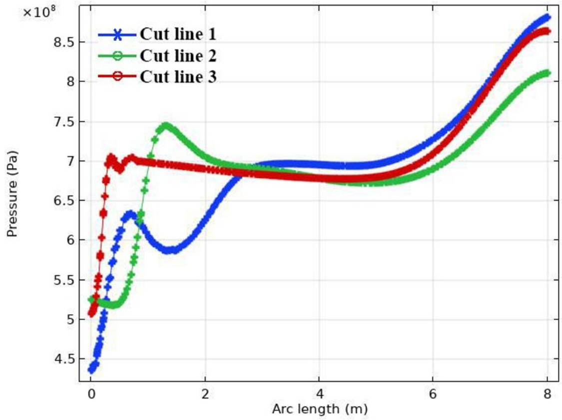

Fig. 5. Pressure variation of fluid at three different Cut lines.

5 Conclusion

COMSOL Multiphysics was used to successfully develop and analyse a two-dimensional steady-state CFD model of the VVER -1000 reactor pressure vessel to study the coolant flow, temperature, and pressure distributions during nominal operating conditions. The heat transfer in the fluids and the laminar flow interfaces was simulated and used to couple the thermal and hydraulic processes in the reactor domain. To make realistic predictions, the geometry (height 0.1914 m, diameter 0.455 m) and temperature-dependent thermophysical properties of pressurized water were used realistically.

The results obtained numerically indicated clean and harmonic fields of temperature, velocity, and pressure in the reactor vessel. The coolant was added to the lower plenum having a temperature of 563.15 K and it was allowed to heat to a steady temperature of about 590 K at the outlet, which confirmed extensive heat removal and stable convective operation. The magnitudes of velocities were 200 m/s in the upper plenum to 1400 m/s at the inlet, which are expected behaviour of circulatory behaviour in the VVER-1000 design. The overall pressure drops over the vessel was determined to be at an acceptable range (~0.2MPa), which confirms acceptable hydraulic vendor.

The three monitoring cut line profiles indicated a steady increasing temperature, a decreasing velocity with altitude and a steady linear pressure gradient-and all these points to the physical consistency of the model and its numerical consistency. The mesh of normal quality (171842 elements, average quality of elements = 0.81) was found to converge with energy balance errors less than 0.5.

In general, the created 2D COMSOL model was an effective and strong tool to investigate the thermo-hydraulic properties of pressurized water reactors. The results validate the safe and uniform cooling behaviour of the reactor in the steady-state. This analysis will be furthered in future work to three-dimensional transient simulations applying state-of-the-art turbulence models and conjugate heat-transfer effects in order to assess non-steady and accident cases with a greater level of accuracy.

References

1. Branislav Vrban et. al., Perspectives on the future of nuclear energy in Slovakia, Energy Systems, (2023). https://doi.org/10.1007/s12667-023-00630-y.

2. Guan Heng Yeoh, Thermal hydraulic considerations of nuclear reactor systems: Past, present and future challenges, Experimental and Computational Multiphase Flow, vol. 1, No. 1, 2019, 3–27. https://doi.org/10.1007/s42757-019-0002-5.
3. Wang M., Wang Y., Tian W., Qiu S., Su G.H., Recent progress of CFD applications in PWR thermal hydraulics study and future directions. Annals of Nuclear Energy (oxford), 150, vp., (2021). https://doi.org/10.1016/j.anucene.2020.107836.
4. Akhmadov Kh.S., Shadmanov I.U., Akramov A.S., Iskandarov Z.S., Sobirov M.M. The thermal efficiency of the heat receiving panels of solar collectors and CFD analysis of a screw duct collector. E3S Web of Conferences, Volume 583, 25 October 2024. https://doi.org/10.1051/e3sconf/202458304001.
5. Akhmadov, K.S., Akhatov, J.S. & Xin, L. Geometric Optimization of a CeO2-Based Solar Thermochemical Reactor for Hydrogen Production: Temperature Distribution Analysis under 2 kW Concentrated Solar Power. Appl. Sol. Energy 61, 102–115 (2025). https://doi.org/10.3103/S0003701X25600729.
6. Lach L., Svyetlichnyy D., Advances in Numerical Modeling for Heat Transfer and Thermal Management: A Review of Computational Approaches and Environmental Impacts. Energies 2025, 18, 1302. https://doi.org/10.3390/en18051302.
7. Böttcher M, Bernard O, Mas A, Sanchez V, Nop R, Belaunde F, et al. CFD analysis of coolant mixing in VVER-1000/V320 reactor pressure vessel. Ann Nucl Energy. 2024; 197:110274. https://doi.org/10.1016/j.anucene.2023.110274.
8. Paffumi E, Nilsson K-F, Szaraz Z. Experimental and numerical assessment of thermal fatigue in 316 austenitic steel pipes. Eng Fail Anal. 2015; 47:312–27.
9. Huang H, Chen C, Liu L, Liu Y, Li L, Yu H, et al. Study on flow boiling characteristics in rectangle channel after formation of blisters. Front Energy Res. 2021; 9:676586.
10. Höhne, T.; Kliem, S. Numerical Analysis Related to the ROCOM Pressurized Thermal Shock Benchmark. Fluids 2023, 8, 4. https://doi.org/10.3390/fluids8010004.
11. Son S.M. et. al., Validation and Application of CFD Methodology for Core Inlet Flow Distribution in APR1000 Reactor. Energies 2025, 18, 512. https://doi.org/10.3390/en18030512.
12. Z. Dong, Z. Wen, F. Zhao et al., Scale-up of micro- and milli-reactors: An overview of strategies, design principles and applications Chemical Engineering Science: X 10 (2021) 100097. https://doi.org/10.1016/j.cesx.2021.100097.
13. Al-Ttowi A., Computational Fluid Dynamics (CFD) Investigation of NREL Phase VI Wind Turbine Performance Using Various Turbulence Models. Processes 2024, 12, 1994. https://doi.org/10.3390/pr12091994.
14. Puragliesi R, Zhou L, Zerkak O, Pautz A. Steady-state CFD simulations of an EPRTM reactor pressure vessel: a validation study based on the JULIETTE experiments. Nuclear Engineering and Design. 2016; 300:41–56.