Id.: EN15

CFD ANALYSIS OF THE AP1000 REACTOR PRESSURE VESSEL

Lucas Telino Borges[1], Daniel González Rodríguez[1], Caio Július César Miranda Rodrigues da Cunha[2], Jair de Lima Bezerra[1], Carlos Alberto Brayner de Oliveira Lira[2]

[1]Universidade Federal de Pernambuco (Av. Prof. Luiz Freire, 1000 – Cidade Universitária,
Recife - PE, 50.740-545, Brazil)
[2]Centro Regional de Ciências Nucleares do Nordeste (CRCN-NE/CNEN) (Av. Prof. Luís
Freire, n°200 – Curado, Recife – PE, 50740-437, Brazil)

lucas.telino@ufpe.br
danielgonro@gmail.com
cjcmrcunha@gmail.com
jairbezerra@gmail.com
carlos.brayner@cnen.gov.br

The Keywords: AP1000; CFD; RPV; Thermal-hydraulic.

ABSTRACT

This paper proposes a thermal and hydraulic analysis of the AP1000 Reactor Pressure Vessel under normal operating conditions through a three-dimensional CFD model. The AP1000 advanced nuclear reactor stands out among other PWRs in terms of technology generation due to its innovative characteristics regarding reliability, modularity, and mainly its passive safety systems. We use the AutoCAD software to build the 3D model of the Reactor Pressure Vessel, and the Ansys Meshing tool for domain discretization. We develop the physical modeling with the Ansys FLUENT software. The paper aims to develop a computational model to assist thermohydraulic investigations of new Reactor Pressure Vessel (RPV) designs. The CFD model estimates the average coolant outlet temperature (576.498 K) and pressure drop (0.46 MPa). The model can offer valuable assistance in advanced nuclear reactor projects, including Small Modular Reactors (SMRs) and Large Reactors (LRs).

1. INTRODUCTION

The use of Computational Fluid Dynamics (CFD) to solve problems related to fluid flow, heat transfer, and chemical reactions has attracted significant attention from both academic and industrial sectors. Its applications are wide-ranging, encompassing fields such as aerodynamics, hydrodynamics, power generation, meteorology, biomedical engineering, maritime engineering, electronic engineering, electrical engineering, and other scientific and engineering disciplines [1].

The use of CFD in nuclear reactor engineering has been growing over the last few years. They can provide detailed insights into phenomena that occur in steady-state or transitory scenarios for both single-phase and multi-phase conditions. CFD is experiencing rapid advancement, including their application to nuclear engineering, which has been extensively discussed at international conferences [2].

CFD enables the detailed prediction of crucial parameters in complex systems, which include the nuclear power plant's operation under different operating conditions. They apply continuously improved mathematical models and high computing processing technologies [3]. CFD is highly effective for predicting complex 3D processes involving fluid flow and heat transfer phenomena. It provides a deeper understanding of the thermo-hydraulic mechanisms in nuclear reactor cores [4].

The Westinghouse AP1000 nuclear reactor, which is a pressurized water reactor (PWR) model rooted in proven technology, has attracted considerable attention as a subject of CFD-based modeling. Countless scholarly publications use the AP1000 reactor as a benchmark for comprehensive analyses across different operational conditions.

For instance, Tao et al. (2019) [5] investigated the transient thermal hydraulics of reduced-scale passive residual heat removal heat exchangers in the water tank. They developed a three-dimensional numerical model using the porous medium approach to analyze the fluid temperature and velocity. Tong et al. (2020) [6] performed the flow distribution and mixing characteristics analysis for the AP1000 Reactor Pressure Vessel through a CFD model.

Furthermore, Xu et al. (2012) [7] examined the fuel performance between the AP1000 reactor and other nuclear reactor projects. They assessed the impact of the AP1000 reactor vessel's upper internals design on fuel performance and included the reactor's upper plenum modeling using CFD. This paper presents a thermal-hydraulic analysis of the AP1000 Reactor Pressure Vessel (RPV) under normal operating conditions employing a 3D CFD model.

2. METHODOLOGY

We performed a thermal analysis of the AP1000 Reactor Pressure Vessel (RPV) using the CFD technique. Initially, we developed a detailed 3D model of the RPV using AutoCAD, assuming the standard configurations outlined in the Westinghouse AP1000 Design Control Document Rev. 19 [8]. Additionally, we optimized the CFD model by taking advantage of the RPV's symmetric characteristics. Fig. 1. illustrates the detailed CAD model and the simplified geometries.

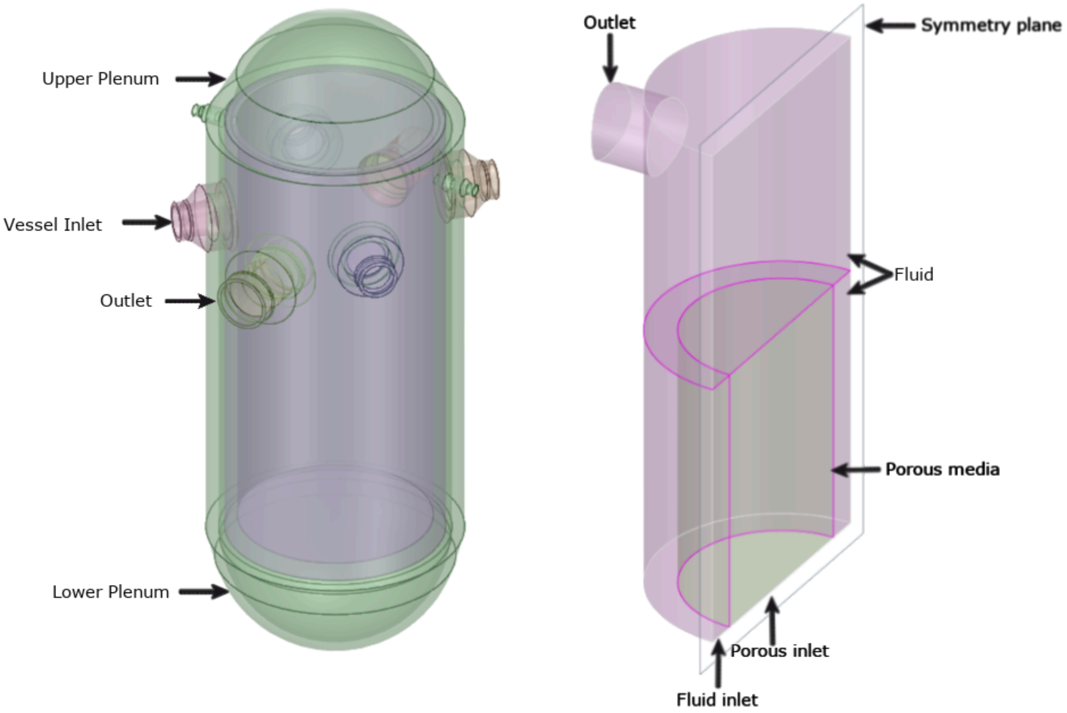

Fig. 1 AP1000 RPV detailed CAD model (Left) and Simplified CAD model (Right)

We employed the Ansys Design Modeler and SpaceClaim tools to create the simplified CAD model. We applied the Ansys Meshing tool to the RPV domain discretization, generating a mesh with 426493 nodes and 1237311 elements, primarily prismatic. We acquired the properties for the water employed as the coolant through a Python script utilizing the IAPWS water properties library, and the parameters derived from the code are outlined in Tab. 1. Subsequently, we integrated these properties into the FLUENT software.

According to data from the International Atomic Energy Agency [9][10], the specific heat, thermal conductivity, density, and molar mass of uranium and zircaloy were weighted at 15/85, taking into account the average temperature of the coolant in the core, which is 576.539 K [6]. The average fuel rods core properties presented in Tab. 1. were assumed for the porous region.

Tab. 1. Average AP1000 reactor fuel rods core and water properties

Properties	**Water**	**Reactor Core (Porous Domain)**
Cp (J/kgK)	5484.3702	119.2662
Thermal conductivity (W/mK)	0.5612	6.8031
Density (kg/m^3)	722.64	10301.0067
Molar Weight (kg/mol)	-	243.2090

The porous medium factors were set as 8.17x107 m-2 and 5.37 m-1 for the viscous and inertial terms, respectively, based on the findings of Yizhou Yan [11]. We define the laminar zone condition for the porous medium and the k-epsilon realizable for the coolant turbulence modeling. Inlet and outlet boundary conditions were set as the mass flow and outlet pressure, respectively, with symmetry conditions at the radial plane faces of the RPV. According to the

AP1000 project documentation, the Reactor Pressure Vessel operates at 13456.57 kg/s effective mass flow rate and a 552.55 K inlet temperature, with a nominal system pressure of 17.13 MPa.

We assume 18.66 kW/m for the reactor's average linear power under normal operation conditions, as reported in the Westinghouse Design Control Document [8]. We implement a Python script to fit a polynomial function to be utilized as input data in Ansys FLUENT, considering an axial power density distribution Fig. 2. shows the power density distribution resulting from the polynomial.

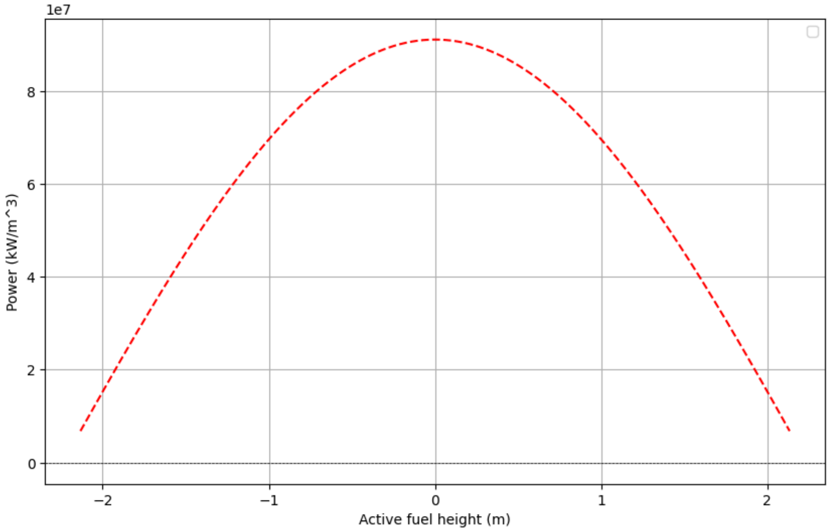

Fig 2. Axial power distribution

The power generation term in Ansys FLUENT was assumed based on the sinusoidal approach defined in Todreas and Kazimi [12], with a maximum power density value of $9.1075*10^7$ kW/m^3. The simplified CFD model developed for the thermal-hydraulic analysis of the AP1000 RPV Westinghouse standard design focuses on the global operating parameters for normal operating conditions.

3. RESULTS

The CFD model shows a pressure drop of 0.46 MPa along the Reactor Pressure Vessel, slightly higher than the 0.43 MPa specified in the Westinghouse Design Control Document [8]. Fig. 3. highlights the RPV pressure drop and the reactor core axial power density distribution.

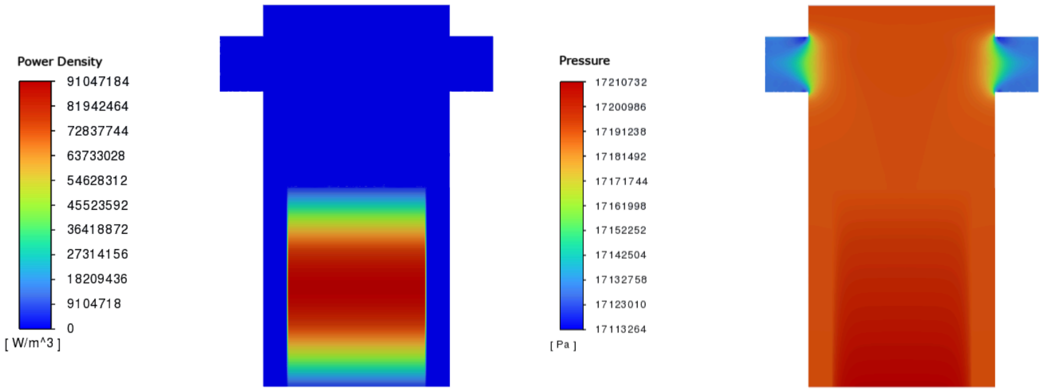

Fig. 3. Contours of power density (Left) and pressure (Right)

Despite employing a conservative approach for the core’s structures and incorporating a porous medium approach, the official AP1000 reactor documents inform that the pressure drop calculation for the RPV is not something trivial. Thus, the findings hold the potential for a more comprehensive analysis. Fig. 4. highlights the temperature contours for the Reactor Pressure Vessel RPV CFD model. The determined average outlet temperature of the RPV for the current CFD model stands at 576.498 K.

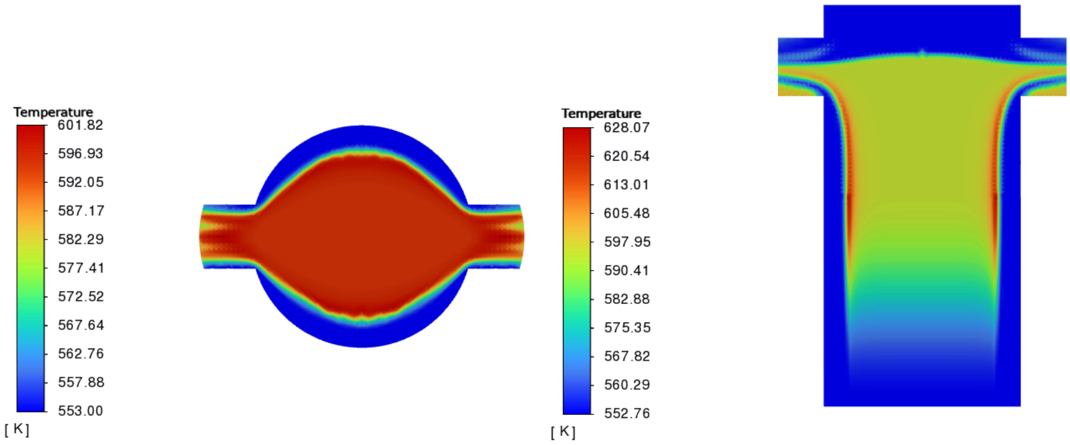

Fig 4. Contours of temperature

The AP1000's standard design specifications indicate 42.90 K as an average temperature increase of the RPV, resulting in an outlet temperature of 595.45 K. This difference is because the simulation was carried out under average operating conditions. The model produced results that aligned with expectations despite the numerous simplifications made. Accounting for the variation in the water temperatures due to changes in operating conditions is one of the improvements that will be made in future analyses.

4. CONCLUSION

We developed a detailed CAD model for the AP1000 Reactor Pressure Vessel based on the structural specifications from the Westinghouse AP1000 Design Control Document [8]. The CFD model uses the porous medium approach for the reactor core without considering the lower and upper plenum structures, as they are not part of the current paper's objective. Our findings showed a pressure drop of 0.46 MPa and an average outlet coolant temperature of 576.498 K, contrasting with Westinghouse's reported values of 0.43 MPa and 595.45 K.

Overall, the CFD model showed satisfactory thermal and hydraulic results, allowing a reasonable initial and simplified analysis of the AP1000 RPV. Despite these promising findings, any future fluid-structure interaction (FSI) analysis of the AP1000 RPV is contingent upon enhancing the CFD model presented in this paper

ACKNOWLEDGEMENTS

The authors would like to thank the CNPq (Conselho Nacional de Desenvolvimento Científico e Tecnológico) and CRCN-NE (Centro Regional de Ciências Nucleares do Nordeste).

BIBLIOGRAPHIC REFERENCES

[1] Versteeg, H.K. and Malalasekera, W., An Introduction to Computational Fluid Dynamics, 2nd ed., London, Ed. Pearson (2007)

[2] Tian et al., Editorial: CFD Applications in Nuclear Engineering, Frontiers in Energy Research, Vol. 9 (2021)

[3] G.H. Yeoh, Thermal hydraulic considerations of nuclear reactor systems: Past, present and future challenges, Exp. Comput. Multiph. Flow 1, pp. 3–27 (2019)

[4] Mingjun Wang et al., Recent progress of CFD applications in PWR thermal hydraulics study and future directions, Annals of Nuclear Energy, Vol. 150, pp. 107836 (2021)

[5] Jiaqi Tao et al., Investigation on transient thermal hydraulics of reduced scale passive residual heat removal heat exchanger in tank, Annals of Nuclear Energy, Vol. 130, pp. 402-410 (2019)

[6] L.L. Tong et al., Analysis of the flow distribution and mixing characteristics in the reactor pressure vessel, Nuclear Engineering and Technology, Vol. 53, n. 1, pp. 93-102 (2021)

[7] Yiban Xu et al., Study of impact of the AP1000® reactor vessel upper internals design on fuel performance, Nuclear Engineering and Design, Vol. 252, pp. 128-134 (2012)

[8] WESTINGHOUSE ELECTRIC COMPANY, Westinghouse AP1000 Design Control Document Rev.19, U.S. Nuclear Regulatory Commission (2019)

[9] Y.E. Kim et al., Thermophysical properties database of materials for light water reactors and heavy water reactors, Vienna, IAEA (2006)

[10] BOBKOV, V. et al. Thermophysical properties of materials for nuclear engineering: a tutorial and collection of data, Vienna, IAEA (2008)

[11] Yizhou Yan, Development of a coupled CFD system-code capability (with a modified porous media model) and its applications to simulate current and next generation reactors, Thesis, University of Illinois, USA (2011)

[12] Todreas, Neil E. and Kazimi, Mujid S., Nuclear Systems Volume I: Thermal Hydraulic Fundamentals, 2nd ed, Ed. CRC Press (2011)