American Journal of Science, Engineering and Technology
2024; Vol. 9, No. 2, pp. 42-49
https://doi.org/10.11648/j.ajset.20240902.11

Review Article

CFD Analysis of Coolant Flow Characteristics in Reactor Pressure Vessel: A Comprehensive Review

Md. Wazihur Rahman[1, *], **Md. Saadbin Chowdhury**[1, 2], **Mohammad Zoynal Abedin**[1]

[1]Department of Mechanical Engineering, Dhaka University of Engineering and Technology, Gazipur, Bangladesh
[2]Department of Mechanical Engineering, Bangladesh University of Engineering and Technology, Dhaka, Bangladesh

Abstract

This comprehensive review explores the application of Computational Fluid Dynamics (CFD) in analyzing coolant flow within Reactor Pressure Vessels (RPVs) of nuclear power plants. By synthesizing existing research, methodologies, and advancements specific to RPVs, the paper offers in-depth insights into critical aspects such as boundary conditions, turbulence modeling, heat transfer mechanisms, and validation techniques. Examining a range of studies encompassing various reactor types from Pressurized Water Reactors (PWRs) to Integral Pressurized Water Reactors (IPWRs), the review underscores CFD's pivotal role in enhancing safety, efficiency, and performance optimization in nuclear reactors. Through systematic exploration, this study underscores the critical importance of precise modeling in facilitating safety assessments, operational optimization, and design enhancements across various reactor systems. Accurate modeling serves as a cornerstone for informed decision-making processes aimed at maximizing reactor performance while ensuring the highest standards of safety and reliability. The paper navigates through challenges such as computational limitations and turbulence modeling intricacies, while also discussing emerging trends like the porous media method aimed at improving computational efficiency. By offering a comprehensive understanding of thermal-hydraulic behavior in nuclear reactors, the review underscores CFD's contribution to enhancing safety and reliability in nuclear power generation. Overall, this review underscores the indispensable role of CFD in advancing our understanding of nuclear reactor dynamics, thereby contributing significantly to the overarching goals of improved safety and reliability in nuclear power generation.

Keywords

Computational Fluid Dynamics, Reactor Pressure Vessel, Coolant Flow, Thermal-Hydraulic Behavior, Safety, Nuclear Power Plant

1. Introduction

In the realm of nuclear engineering, understanding the behavior of coolant flow within a reactor pressure vessel (RPV) is crucial for ensuring the safety and efficiency of nuclear power plants. Computational Fluid Dynamics (CFD) has emerged as a powerful tool for simulating and analyzing coolant flow phenomena within RPVs. This comprehensive review aims to

*Corresponding author: wazihurr@gmail.com (Md. Wazihur Rahman)

Received: 12 March 2024; **Accepted:** 25 March 2024; **Published:** 11 April 2024

delve into the intricacies of CFD analysis applied to coolant flow in RPVs. By synthesizing existing research, methodologies, and advancements, this review provides a holistic understanding of the challenges, methodologies, and applications pertinent to this field. The review will explore various aspects including boundary conditions, turbulence modeling, heat transfer mechanisms and validation techniques specific to CFD simulations of coolant flow in RPVs. Furthermore, it will critically assess the strengths and limitations of different CFD approaches, highlighting areas for future research and development. Through this comprehensive review, researchers, engineers, and stakeholders in the nuclear industry will gain valuable insights into the state-of-the-art techniques and best practices for conducting CFD analysis of coolant flow in RPVs, ultimately contributing to enhanced safety and performance of nuclear power plants.

2. Nuclear Power Plant

Nuclear energy is a form of energy released from the nucleus, the core of atoms, made up of protons and neutrons. This source of energy can be produced in two ways: fission – when nuclei of atoms split into several parts or fusion when nuclei fuse together. The nuclear energy harnessed around the world today to produce electricity is through nuclear fission, while technology to generate electricity from fusion is at the R&D phase [1]. A nuclear power plant is a thermal power plant, in which a nuclear reactor is used to generate large amounts of heat. This heat is used to generate steam (directly or via steam generator) which drives a steam turbine connected to a generator that produces electricity.

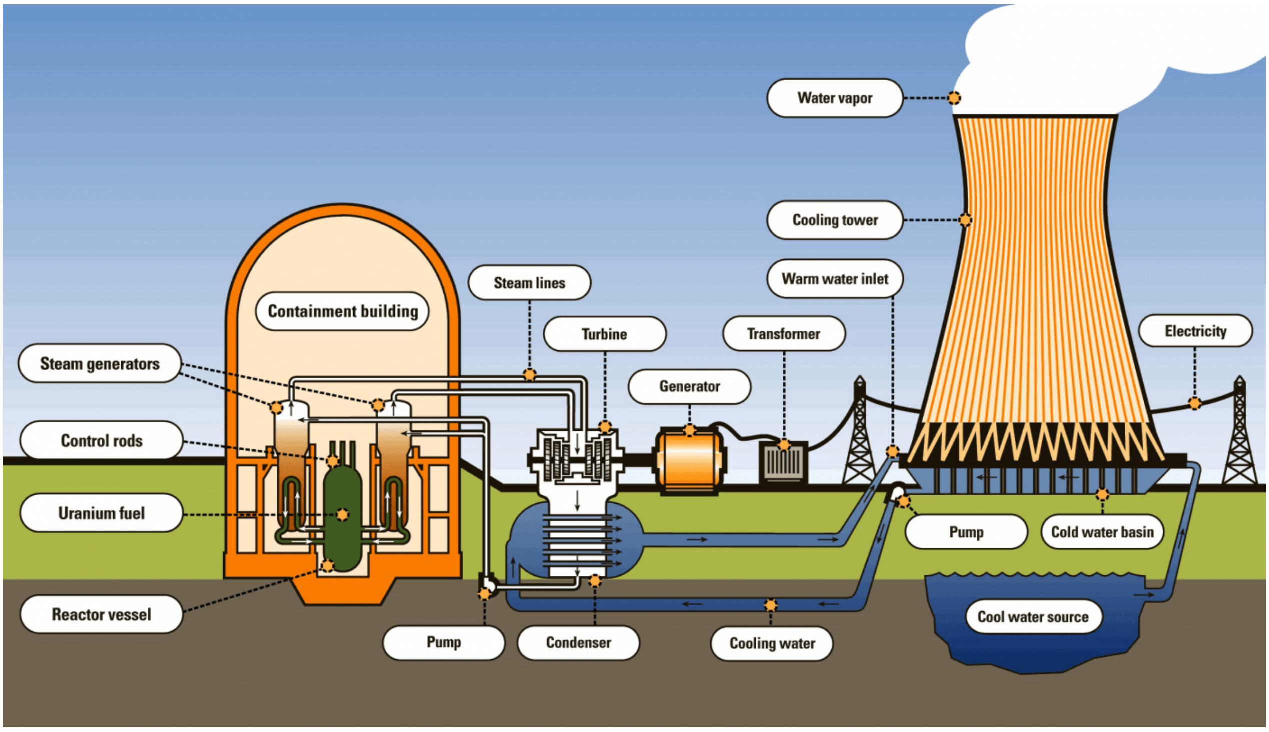

Figure 1. *Basic Components and layout of a Nuclear Power Plant [2].*

Heavy elements such as Uranium (U235) or Thorium (Th232) are subjected to nuclear fission reaction in a nuclear reactor. Due to fission, a large amount of heat energy is produced which is transferred to the reactor coolant. The coolant may be water, gas or a liquid metal. The heated coolant is made to flow through a heat exchanger where water is converted into high-temperature steam. The generated steam is then allowed to drive a steam turbine. The steam, after doing its work, is converted back into the water and recycled to the heat exchanger. The steam turbine is coupled to an alternator which generates electricity. The generated electrical voltage is then stepped up using a transformer for the purpose of long distance transmission [3]. Nuclear power plants are advanced facilities designed to harness the incredible energy released during nuclear reactions for the generation of electricity. These plants play a crucial role in meeting the growing global demand for energy while minimizing greenhouse gas emissions. Unlike conventional power plants that rely on burning fossil fuels, nuclear power plants generate electricity through controlled nuclear fission reactions.

2.1. Reactor Pressure Vessel

The reactor pressure vessel is the pressure vessel containing the reactor core and other key reactor internals. Most commercial power reactors are light water reactors (PWRs and

BWRs), which are cooled and moderated by high-pressure liquid water (e.g. 16MPa in case of PWRs), and therefore the reactor vessel must withstand high pressures.

It is a cylindrical tank featuring a hemispherical base and an upper head equipped with a flange and gasket for sealing purposes. The bottom head is welded to the cylindrical shell while the top head is bolted to the cylindrical shell via the flanges. The top head is removable to allow for the refueling of the reactor during planned outages. One inlet (or cold leg) nozzle and one outlet (or hot leg) nozzle for each reactor coolant system loop. The reactor coolant enters the reactor vessel at the inlet nozzle. It exits the reactor at the upper internals region, where it is routed out the outlet nozzle into the hot leg of the primary circuit and goes on to the steam generator. The primary circuit of a typical PWR is divided into 4 independent loops (piping diameter ~ 800mm). Each loop comprises a steam generator and one main coolant pump but can differ according to certain reactor designs. Therefore, numerous inlet and outlet nozzles, control rod drive tubes (in case of BWRs), instrumentation, and safety injection nozzles penetrate the cylindrical shell. This number of inlet and outlet nozzles is a function of the number of loops.

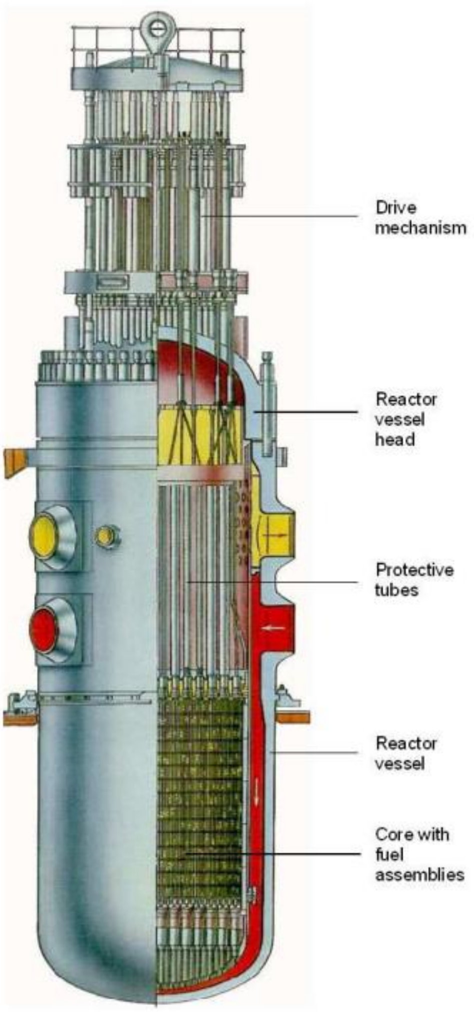

Figure 2. *Reactor Pressure Vessel [4].*

The reactor pressure vessels are the highest priority key components in nuclear power plants. The reactor pressure vessel houses the reactor core, and because of its function, it has direct safety significance. During the operation of a nuclear power plant, the material of the reactor pressure vessel is exposed to neutron radiation (especially to fast neutrons), which results in localized embrittlement of the steel and welds in the area of the reactor core. Radial neutron reflectors are installed around the reactor core to minimize such material degradation. There are two basic types of neutron reflectors, the core baffle, and the heavy reflector. Due to higher atomic number density, heavy reflectors reduce neutron leakage (especially of fast neutrons) from the core more efficiently than the core baffle. Since the reactor pressure vessel is considered irreplaceable, these aging effects of the RPV have the potential to be life-limiting conditions for a nuclear power plant [5].

2.2. Coolant Flow in Reactor Pressure Vessel

A nuclear reactor coolant is a coolant in a nuclear reactor used to remove heat from the nuclear reactor core and transfer it to electrical generators and the environment. Frequently, a chain of two coolant loops are used because the primary coolant loop takes on short-term radioactivity from the reactor [6].

The flow path for the reactor coolant through the reactor vessel would be:

The coolant enters the reactor vessel at the inlet nozzle and hits against the core barrel. The core barrel forces the water to flow downward in the space between the reactor vessel wall and the core barrel, and this space is usually known as the downcomer. The flow is reversed up through the core from the bottom of the pressure vessel to pass through the fuel assemblies, where the coolant temperature increases as it passes through the fuel rods. Finally, the hotter reactor coolant enters the upper internals region, where it is routed out the outlet nozzle into the hot legs of the primary circuit and goes on to the steam generators. The body of the reactor vessel is constructed of high-quality low-alloy carbon steel, and all surfaces that come into contact with reactor coolant are clad with a minimum of about 3 to 10 mm of austenitic stainless steel to minimize corrosion [5].

3. CFD Analysis

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows. Computers are used to perform the calculations required to simulate the free-stream flow of the fluid, and the interaction of the fluid (liquids and gases) with surfaces defined by boundary conditions [7]. When an engineer is tasked with designing a new product, e.g. a winning race car for the next race season, aerodynamics plays an important role in the overall performance of the design. That said, aerodynamic performance is not easily quantifiable during the concept phase. Traditionally, the only way for an engineer to optimize his/her design is to conduct physical tests on product prototypes. With the rise of computers and ever-growing computational power (thanks to Moore's law), the field of CFD has become a commonly applied tool for predicting real-world physics. In a CFD software analysis, fluid flow and its associated physical properties, such as velocity, pressure, viscosity, density, and temperature, are calculated based on defined operating conditions. In order to arrive at an accurate, physical solution, these quantities are calculated simultaneously [8].

Computational Fluid Dynamics (CFD) analyses play a crucial role in the design, operation, and safety assessment of reactor pressure vessels (RPVs) in nuclear power plants. RPVs are pivotal components that house the nuclear fuel and coolant, and understanding the flow behavior within them is essential for ensuring optimal performance and safety. CFD offers a powerful toolset for simulating fluid flow, heat transfer, and other relevant phenomena within RPVs, enabling engineers to evaluate various operating conditions, predict potential hazards, and optimize design parameters. In this section, we explore the diverse applications of CFD analyses in RPVs, ranging from thermal-hydraulic performance assessments to safety evaluations and beyond.

4. CFD Analysis in Coolant Flow in Reactor Pressure Vessel

Computational Fluid Dynamics (CFD) Analysis in coolant flow is a method used to simulate and analyze the behavior of coolant fluids within various engineering systems, such as heat exchangers, engines, and electronic components. It employs mathematical equations to model fluid flow, heat transfer, and other related phenomena, providing insights into factors like velocity distribution, pressure gradients, and temperature variations within the coolant system. CFD allows engineers to optimize designs, improve efficiency, and ensure proper cooling in various applications, contributing to the development of more reliable and efficient cooling systems.

4.1. Coolant Flow in PWR Reactor

Coolant flow in Pressurized Water Reactors (PWRs) plays a critical role in transferring heat from the nuclear fuel to generate steam for power production, ensuring safe and efficient operation of the reactor. Various researches have been performed in this field.

H. Farajollahi et al. presented a mathematical model and computational fluid dynamics (CFD) analysis of fluid flow and heat transfer in a Pressurized Water Reactor (PWR), focusing on the downcomer and lower plenum regions. It discussed the importance of accurate modeling for safety analysis and reactor performance assessment. The study utilized ANSYS CFX software for numerical simulations, aiming to understand fluid dynamics, velocity distribution, and heat transfer within the reactor pressure vessel. Challenges such as computational limits and simplifications in modeling are addressed. The paper included steady-state and transient simulations, with discussions on velocity distribution, outlet temperature, and flow fields. While the model is validated against experimental data, future work is suggested to improve reactor geometry and study transient mixing behavior in more detail. Overall, the paper highlights the significance of CFD in enhancing understanding and safety in PWR reactors [9].

In conclusion, understanding and optimizing coolant flow in Pressurized Water Reactors (PWRs) is essential for maintaining reactor safety and efficiency, ensuring effective heat transfer and power generation while minimizing the risk of accidents such as Departure from Nucleate Boiling (DNB). Continued research and analysis of turbulent flow properties within PWR coolant channels are vital for advancing reactor design and operation.

4.2. Coolant Flow in IPWR Reactor

An Integral PWR refers to a specific type of reactor design where key reactor components, such as the reactor vessel, steam generators, and primary coolant loops, are integrated into a single pressure vessel. This integration simplifies the reactor design, reduces the number of external piping and connections, and can enhance safety by minimizing potential leak points. Various researchers have performed in this field.

Lin Sun et al. investigated nonuniform coolant flow and temperature distribution in an Integral Pressurized Water Reactor (IPWR) during low power operation, particularly when using a group operation strategy for once-through steam generators (OTSGs). To mitigate this issue, a flow mixing chamber (FMC) was designed to enhance coolant mixing. Computational fluid dynamics (CFD) simulations model the downcomer and lower plenum, revealing that the FMC effectively reduces temperature differences at the core inlet. Sensitivity analysis of turbulence models confirms their suitability for the study, while optimization of FMC discharge hole sizes improved coolant mixing but increases flow resistance. Overall, the study emphasized the importance of addressing coolant distribution for reactor safety and operation optimization in IPWRs [10].

In conclusion, understanding coolant flow dynamics in Integral Pressurized Water Reactors (IPWRs) is crucial for ensuring their safe and efficient operation, impacting heat transfer, reactor performance, and overall system integrity. Ongoing research and analysis of coolant flow properties will advance IPWR technology, enhancing its suitability for future nuclear power generation.

4.3. Coolant Flow in VVER-1000

Coolant flow in VVER-1000 reactors is fundamental for transferring heat from the nuclear fuel to produce steam, ensuring efficient power generation. Understanding coolant flow dynamics is crucial for maintaining reactor safety and performance in VVER-1000 reactors. Various researches have been performed in this field.

Khanbabaei et al. presented a mathematical model which was associated with the VVER-1000 coolant flow in its RPV using CFX assuming some simplifications which include some reduction in the total number of fuel assemblies. However, the model was validated by the outlet temperature [11].

Shojja Ayed Aljasar et al. explored the thermal-hydraulic behavior of coolant flow in the sub-channels of fuel assemblies within a VVER-1000 reactor using Computational Fluid Dynamics (CFD) simulations conducted with ANSYS Fluent. It investigated the accuracy of different mesh densities and turbulence models to predict water flow properties, velocity distribution, pressure changes, and hydraulic resistance. The study aimed to validate and design the VVER-1000 reactor considering turbulent flow heat transfer. Results indicated that appropriate mesh density and turbulence models are crucial for accurately predicting disturbances in sub-channels. The Reynolds BSL stress model was chosen for further investigation, showed promising results in simulating coolant flow behavior. Future research aimed to explore full-dimensional fuel bundle models to enhance safety analysis and operation of nuclear power plants [12].

M. Abbasi et al. investigated the three-dimensional flow distributions in the downcomer and lower plenum of the VVER-1000, V446 reactor at the Bushehr nuclear power plant, Iran, using computational fluid dynamics (CFD) simulations. Despite computational limitations, the study showed reasonable agreement between numerical results and measured data, emphasizing the importance of accurate heat transfer knowledge for proper reactor design and operation. CFD methods offer insights into fluid distribution, crucial for preventing and predicting accidents. The study highlighted the growing interest in using CFD for analyzing thermal-hydraulic behavior in nuclear reactors, although a consistent picture regarding predictive accuracy and efficiency of CFD methods has not yet emerged. The paper outlined the computational setup and discusses the results, providing valuable insights into reactor performance and safety considerations [13].

In conclusion, investigating coolant flow in VVER-1000 reactors is essential for optimizing reactor safety, efficiency, and performance, with implications for heat transfer and power generation. Continued research and analysis of coolant flow dynamics will contribute to further enhancing the operation and safety of VVER-1000 reactors.

In conclusion, understanding coolant flow dynamics in Integral Pressurized Water Reactors (IPWRs) is crucial for ensuring their safe and efficient operation, impacting heat transfer, reactor performance, and overall system integrity. Ongoing research and analysis of coolant flow properties will advance IPWR technology, enhancing its suitability for future nuclear power generation.

4.4. Coolant Flow in VVER-1200 Reactor

Coolant flow in VVER-1200 reactors is critical for transferring heat from the nuclear fuel to generate steam for power production, ensuring safe and efficient operation of the reactor. Understanding and optimizing coolant flow dynamics are essential for maintaining reactor safety and performance. Various researches have been performed in this field.

Huseyin Ayhan & Sule Ergun performed a study on computational fluid dynamics (CFD) simulations to investigate the thermal-hydraulic behavior of the Reactor Pressure Vessel (RPV) in the VVER-1200 (AES-2006) nuclear reactor. Utilizing a 3D modeling approach, the outer sections of the RPV are detailed while the inner sections are simplified using a porous media method to accommodate core complexities. Adopting the OECD VVER-1000 coolant transient benchmark with adjustments for the VVER-1200 reactor core, the research conducted ANSYS Fluent analyses to compute temperature distribution and pressure drop, which align well with literature values. Through the porous media modeling technique, computational efficiency was enhanced, providing reasonable results for temperature and pressure behaviors, particularly in the intricate reactor core section. The study further examined the effects of variations in cold leg temperature on reactor safety, highlighting their significant impact. In conclusion, the porous media approach emerged as a valuable tool for capturing essential thermal-hydraulic phenomena while minimizing computational demands and time [14].

Begüm Kütük and Ibrahim Halil Güzelbey presented a comprehensive computational fluid dynamics (CFD) simulation for a VVER-1200 nuclear reactor vessel under various operating conditions, including symmetric and asymmetric inlet boundary conditions, as well as loss-of-coolant accident scenarios. Through CFD analyses, the temperature, velocity, pressure, and mass flow rate distributions within the reactor vessel were investigated. The study utilized sophisticated CFD methods to simulate the complex three-dimensional fluid flow and heat transfer phenomena, providing valuable insights into reactor safety and performance. The results demonstrated good agreement with analytical values and benchmark data, validating the effectiveness of the CFD simulation approach. The paper also discussed the importance of possessing numerical knowledge for reactor safety and suggested future directions for improving reactor vessel models [15].

Taosif Alam and M. A. R. Sarkar conducted a three-dimensional Computational Fluid Dynamics (CFD) analysis to investigate turbulent flow characteristics within the interior, edge, and corner sub-channels of a hexagonal fuel sub-assembly, resembling those in the Rooppur Nuclear Power Reactor in Bangladesh. The study focused on understanding the turbulent properties' impact on velocity, temperature, convective heat transfer, pressure drop, friction factor, Nusselt number, and the probability of Departure from Nucleate Boiling (DNB). By utilizing a k-ε turbulence model, the analysis explored how turbulent kinetic energy, turbulence dissipation rate, and turbulent eddy viscosity vary along the axial length of the sub-channels. Key findings included the significant influence of turbulent properties on flow behavior, with corner sub-channels exhibiting higher turbulence levels and lower Nusselt numbers, potentially leading to DNB and hot spot occurrence. The study suggested optimal fuel sub-assembly lengths to achieve fully developed Nusselt numbers and highlights differences in friction factors among sub-channels. Overall, the analysis provides insights into thermal-hydraulic aspects crucial for reactor safety and operational efficiency [16].

In conclusion, studying coolant flow in VVER-1200 reactors is crucial for ensuring safe and efficient reactor operation, with implications for heat transfer, power generation, and overall system integrity. Continued research and analysis of coolant flow dynamics will contribute to further enhancing the safety and performance of VVER-1200 reactors.

5. Summary

Overview of CFD Analysis of coolant flow in different reactor pressure vessels is discussed in Table 1.

Table 1. *CFD Analysis of coolant flow in different reactor pressure vessel.*

Field of Review	Observation	Reference
CFD Analysis in PWR	The study utilized ANSYS CFX software for numerical simulations, aiming to understand fluid dynamics, velocity distribution, and heat transfer within the reactor pressure vessel	[9]
CFD Analysis in IPWR	Computational fluid dynamics (CFD) simulations model the downcomer and lower plenum, revealing that the FMC effectively reduces temperature differences at the core inlet	[10]
CFD Analysis in VVER-1000	It investigated the accuracy of different mesh densities and turbulence models to predict water flow properties, velocity distribution, pressure changes, and hydraulic resistance.	[12]
CFD Analysis in VVER-1000	The study highlighted the growing interest in using CFD for analysing thermal-hydraulic behaviour in nuclear reactors, although a consistent picture regarding predictive accuracy and efficiency of CFD methods has not yet emerged	[13]
CFD Analysis in VVER-1200	A study performed on computational fluid dynamics (CFD) simulations to investigate the thermal-hydraulic behaviour of the Reactor Pressure Vessel (RPV) in the VVER-1200 (AES-2006) nuclear reactor.	[14]
CFD Analysis in VVER-1200	The findings underscore the significance of advanced CFD methods in enhancing understanding of reactor safety and performance, with implications for future model improvements.	[15]
CFD Analysis in VVER-1200	The study highlighted the importance of optimal fuel sub-assembly lengths for achieving fully developed Nusselt numbers and identified differences in friction factors among sub-channels, providing insights crucial for reactor safety and efficiency.	[16]

6. Results & Discussion

The studies presented encompass a comprehensive exploration of Computational Fluid Dynamics (CFD) applications in various nuclear reactor systems, ranging from Pressurized Water Reactors (PWRs) to Integral Pressurized Water Reactors (IPWRs) and VVER-1000 reactors. Farajollahi et al. delved into the fluid flow and heat transfer dynamics within a PWR, emphasizing the criticality of accurate modeling for safety analysis and reactor performance assessment. Utilizing ANSYS CFX software, their simulations provided insights into velocity distribution, temperature profiles, and flow fields in the downcomer and lower plenum regions. Similarly, Sun et al. investigated nonuniform coolant flow in an IPWR during low power operation, demonstrating the effectiveness of a flow mixing chamber in enhancing coolant mixing and reducing temperature differences at the core inlet.

Aljasar et al.'s study focused on the thermal-hydraulic behavior of coolant flow in VVER-1000 reactor sub-channels, highlighting the importance of mesh density and turbulence models for predicting water flow properties accurately. Their findings underscored the necessity of appropriate modeling techniques for ensuring the safety and efficiency of nuclear power plants. Abbasi et al. further investigated flow distributions in the downcomer and lower plenum of the VVER-1000 reactor at the Bushehr nuclear power plant, showcasing reasonable agreement between numerical results and measured data. Their study emphasized the growing interest in using CFD for analyzing thermal-hydraulic behavior in nuclear reactors, despite challenges associated with predictive accuracy and computational limitations.

Lastly, Ayhan & Ergun's research on the VVER-1200 reactor highlighted the significance of the porous media approach in capturing essential thermal-hydraulic phenomena while minimizing computational demands. Their study provided valuable insights into temperature distribution, pressure behaviors, and the impact of variations in cold leg temperature on reactor safety.

Overall, these investigations underscore the pivotal role of CFD in enhancing understanding, safety analysis, and operation optimization in nuclear reactors, paving the way for further advancements in reactor design and operation.

7. Conclusion

In conclusion, the collective body of research presented in this paper demonstrates the significant advancements made in utilizing Computational Fluid Dynamics (CFD) for analyzing fluid flow, heat transfer, and thermal-hydraulic behavior within various nuclear reactor systems. From Pressurized Water Reactors (PWRs) to Integral Pressurized Water Reactors (IPWRs) and VVER-1000 reactors, researchers have employed sophisticated modeling techniques and simulation software to gain insights into critical aspects of reactor performance and safety.

The studies have elucidated the importance of accurate modeling for safety assessments, operation optimization, and design improvements in nuclear reactors. Despite challenges such as computational limitations and the need for appropriate turbulence models, the results underscore the growing interest and effectiveness of CFD in analyzing thermal-hydraulic behavior. Furthermore, the development and validation of novel modeling approaches, such as the porous media method, offer promising avenues for enhancing computational efficiency while capturing essential phenomena accurately.

Overall, these findings highlight the continued evolution and importance of CFD in advancing our understanding of nuclear reactor systems, ultimately contributing to enhanced safety, efficiency, and reliability in nuclear power generation.

Abbreviations

CFD: Computational Fluid Dynamics
RPV: Reactor Pressure Vessel
PWR: Pressurized Water Reactor
IPWR: Integral Pressurized Water Reactor
BWR: Boiling Water Reactor
OTSGs: Once-Through Steam Generators
FMC: Flow Mixing Chamber
DNB: Departure from Nucleate Boiling

Author Contributions

Md. Wazihur Rahman: Conceptualization, Data curation, Formal Analysis, Funding acquisition, Methodology, Resources, Visualization, Writing - original draft, Writing - review & editing

Md. Saadbin Chowdhury: Conceptualization, Data curation, Formal Analysis, Funding acquisition, Methodology, Resources, Visualization, Writing - original draft, Writing - review & editing

Mohammad Zoynal Abedin: Conceptualization, Data curation, Formal Analysis, Funding acquisition, Methodology, Resources, Visualization, Writing - original draft, Writing - review & editing

Conflicts of Interest

The authors declare no conflicts of interest.

References

[1] International Atomic Energy Agency. What is Nuclear Energy? The Science of Nuclear Power. Available from: https://www.iaea.org/newscenter/news/what-is-nuclear-energy-the-science-of-nuclear-power [Accessed 15 November 2022].

[2] Foro Nuclear. What are the different components of a nuclear power plant? Available from: https://www.foronuclear.org/en/updates/in-depth/what-are-the-different-components-of-a-nuclear-power-plant/ [Accessed 19 December 2014].

[3] Electrical Easy. Basic Layout and Working of a Nuclear Power Plant. Available from: https://www.electricaleasy.com/2015/09/nuclear-power-plant.html [Accessed 2021].

[4] Pavel Polach and Michal Hajžman, ECCOMAS Congress 2016, VII European Congress on Computational Methods in Applied Sciences and Engineering, M. Papadrakakis, V. Papadopoulos, G. Stefanou, V. Plevris (eds.), Crete Island, Greece, 5–10 June 2016.

[5] Nuclear Power. Reactor Pressure Vessel. Available from: https://www.nuclear-power.com/nuclear-power-plant/nuclear-reactor/reactor-pressure-vessel/ [Accessed 2024].

[6] Wikipedia. Nuclear Reactor Coolant. Available from: https://en.wikipedia.org/wiki/Nuclear_reactor_coolant [Accessed 19 January 2024].

[7] Wikipedia. Computational Fluid Dynamics. Available from: https://en.wikipedia.org/wiki/Computational_fluid_dynamics [Accessed 20 January 2024].

[8] Simscale. What is CFD | Computational Fluid Dynamics? Available from: https://www.simscale.com/docs/simwiki/cfd-computational-fluid-dynamics/what-is-cfd-computational-fluid-dynamics/ [Accessed 7 December 2023].

[9] H. Farajollahi, A. Ghasemizad, and B. Khanbabaei, "Journal of Sciences, 2008, 19(3), 273-281.

[10] Lin Sun, Minjun Peng, Genglei Xia, Xing Lv, and Ren Li, Nuclear Engineering and Technology. 2017, 49, 71-81.

[11] B. Khanbabaei, A. Ghasemizad, H. Farajollahi, CFD-calculation of fluid flow in VVER-1000 reactors, J. Appl. Sci. 2008, 8(5), 780–787.

[12] Shojja Ayed Aljasar, Oleg. Yu Dolmatov, Muhammad Saqib & Yubin Xu, International Journal of Mechanical and Production Engineering Research and Development (IJMPERD), 2020, Vol. 10, Issue 2, 1419–1428.

[13] M. Abbasi, A. Ghasemizad, B. Khanbabaei, K. Keshtkar, Journal of Theoretical and Applied Physics, 2011, 5-3, 135-141.

[14] Huseyin Ayhan, Sule Ergun, Ayhan and Ergün/ BgNS conference 18-21 September 2013.

[15] B. Kütük and I. H. Güzelbey, International Journal of Pressure Vessels and Piping. 2020, 187(2020) 104165.

[16] Taosif Alam and M. A. R. Sarkar, International Conference on Mechanical, Industrial and Energy Engineering 2018, 23-24 December, 2018, Khulna, Bangladesh.