Development of the tools of high speed photography since the first photo of the horse in 1878 is ongoing till now. The first photo was taken with a frame rate 69 fps. A classification divided the high speed imaging into four categories according to the frame rate, a) High speed imaging in the range of 50-500 frame/second (fps), the system uses mechanical shuttering and discontinues film motion; b) Very high speed imaging with rate between 500-100,000 (fps), using a continuous moving film and a digital video system; c) Ultra high speed imaging, 100,000–10 million fps (Mfps),using stationary film with moving image systems and electronically with image converter cameras; and (d) super high-speed, more than 10 Mfps, where film has been largely superseded by electronic imaging and recording [1].

Nowadays recent technology achieved many types of cameras capable of photographing with a very high frame rate, for example we have "Shimadzu HPV X2" camera recorded photos with 10 Mfps also "Kirana" camera achieved 5 Mfps [2].

REFERENCES

1. Zhang H.Z.X.Q.B. High-Speed Photography and Digital Optical Measurement Techniques for Geomaterials : Fundamentals and Applications // Rock Mechanics and Rock Engineering. Springer Vienna, 2017.
2. Slangen P. et al. Recent developments in high speed imaging and applications in speckle light To cite this version : HAL Id : hal-02011901. 2021.

Adam Mustapha Umar (Nigeria)
Tomsk Polytechnic University, Tomsk
Scientific supervisor: Goryunov Alexey Germanovich

CFD APPLICATIONS IN PWR THERMAL-HYDRAULICS

Introduction

In the last decade, three-dimensional CFD codes have increasingly been used to predict single-phase and multiphase flows in nuclear reactor applications under stationary or unsteady conditions. CFD is used to process mathematical modelling of a physical phenomenon involving fluid flow and solve it numerically using computational prowess. CFD modelling and analysis became a popular online simulation solution as the difficulty grew in applying the laws of physics directly to real-life scenarios to make analytical predictions. This circumstance became prevalent especially for heat transfer and fluid flow engineering problems.

Detailed heat transfer and fluid flow features can be predicted accurately using the CFD technology in 3-D physical phenomena in the operation of nuclear reactor, which is beneficial for a deep understanding of thermal-hydraulic mechanisms. Application of CFD technology range could cover the whole nuclear reactor systems, including primary circuit system, secondary circuit system and containment. In general, the CFD community within the nuclear reactor safety circle is formed and becoming more and more critical for the nuclear engineering development in the world [1]. The CFD method, utilizing powerful computers and applied mathematics to model heat transfer and fluid flow in industrial process, is developing rapidly. Worldwide there is a significant attention in applying three-dimensional CFD codes in estimating single-phase and multiphase flows under steady-state or transient conditions in nuclear reactors [2]. SimScale's CFD software can analyze various problems related to laminar and turbulent flows. CFD acts as a virtual fluid dynamics simulator [3]. Current studies proved simulations were performed using the commercial three-dimensional (3-D) Computational Fluid Dynamics (CFD) code. The estimation capabilities of different bubble departure diameter models and departure frequency models for subcooled boiling flow are compared by ANSYS CFX [4].

CFD has increased in importance and accuracy; however, its estimations are never entirely exact due to many potential sources of error that may be involved, hence the results from CFD techniques needs to be carefully interpreted [5].

Deterministic safety analyses for nuclear reactors are conducted through mathematical modelling and computer simulations. Thermal-hydraulic analyses of potential accidents play a significant role in overall safety assessment. The nuclear industry has developed two computer codes for such analyses: the system codes, also called the lumped parameter codes, based on one-dimensional models of physical phenomena, and the field codes, based on three-dimensional models and CFD technique [6]. CFD In open medium also contributes to investigations when 3D geometrical aspects play an essential role [7].

Coolant Mixing

CFD methods are an operational tool to compute 3-D mixing phenomena. It is now practicable to simulate transient, 3-D flows and transport of de- borated water in PWR owing to the rapid advance of computer hardware and software. Throughout small break loss of coolant accidents borated water can collect in the steam generator tubes and the pump suction lines which may instigate an excursion of reactivity and destruction to the reactor core. Numerical investigations and turbulence models require experimental validation based on detailed flow. This investigations on coolant mixing in PWRs have been performed by other institutes and at the FZD for more than a decade [8] [9].

Boron Dilution

Mechanisms are set with the potential to accumulate boron free volumes, and such mechanisms include, for instance, Inadvertent dilution during maintenance and accumulation of condensate under specific conditions during small break loss of coolant accident (SBLOCA). CFD technology is used to assess this mechanism and even possibly eliminate the safety concerns by developing a three-dimensional, neutron kinetics thermal-hydraulics code enabling the modelling of coolant flow in the reactor, transport and mixing boron to prevent reactivity excursion.

There is a possibility of accidental supply in the reactor core coolant accumulated in one of the cold legs. The creation of boron dilute slug may be caused due to the internal and external reasons in the primary coolant system. The CFD technology uses mechanism set for the boron dilution process to prevent reactivity excursion. The Western PWR and Russian WWER-440, 1000 reactors analogues apply the boron-based reactivity control system [10].

Reactor core

In PWR, the coolant flows through the rod bundles to subtract the heat produced by the fission reaction in the fuel element. The thermal power fluctuates spatially in the fuel assembly due to the non-uniform neutron flux density in each fuel rod producing transverse temperature peaks in subchannel. The temperature peaks result in thermal stress in fuel cladding as well as trends to cause DNB suffering the safety of the nuclear reactor. Several designs strategies as a result are performed to enhance the flow mixing and flat the unevenness of temperature through subchannels. Using CFD methods has remained one of the most vital ways to understand the design structure optimization, accompanying with the necessary experiments [1].

CFD requirement for simulation of passive systems

Even though passive systems are simple to use, their incorporation in the reactor concepts needs to be tested effectively due to various technical issues such as quantification of their functional capability. The main difficulties in the assessment of functional capability are as a result of lack of plant data and operational experiences; lack of sufficient experimental data from necessary facilities or even from separate effect tests in order to understand their performance characteristics not only at regular operation but also during transients and accidents; difficulty in modelling the physical behaviour of such systems. However, these "Best Estimate Codes" might not be capable to quantify the functional capability of these passive systems. This is because passive systems operate on the principle of natural circulation, which has a low driving force and, as a result, the flow may not be fully developed. Thus, for understanding the characteristics of these systems and predicting their functional behaviour, rigorous CFD analysis is a must. [11].

Conclusion

CFD is increasingly being used in the nuclear society to develop safety-releted practices occuring in the reactor coolant system. For this determination, numerous research and development programs are dedicated to develop new theoretical models for the basic experiences of transient, single and 3 – D systems. Such models are essential in nuclear facilities for complex fluid dynamics codes to modelling flow relate phenomena.

REFERENCES

1. M. Wang, Y. Wang, W. Tian, S. Qiu, and G. H. Su, "Recent progress of CFD applications in PWR thermal hydraulics study and future directions," Annals of Nuclear Energy, vol. 150, Jan. 2021, doi: 10.1016/j.anucene.2020.107836.
2. C. H. Song, B. L. Smith, D. Bestion, and Y. A. Hassan, "Special Issue of the 4th Workshop on the CFD for Nuclear Reactor Safety (CFD4NRS-4)," Nuclear Engineering and Design, vol. 279, pp. 1–2, 2014, doi: 10.1016/J.NUCENGDES.2014.03.004.
3. "Computational Fluid Dynamics (CFD) Simulation Software | SimScale." https://www.simscale.com/product/cfd/ (accessed Mar. 17, 2022).
4. Y. C. Lin, Y. Zhao, M. Ishii, J. P. Schlegel, K. J. Hogan, and J. R. Buchanan, "Assessment of Nucleation Boiling Models and Improvement by the Chen Correlation for Two-fluid Model CFD," International Journal of Heat and Mass Transfer, vol. 175, Aug. 2021, doi: 10.1016/j.ijheatmasstransfer.2021.121363.
5. H. H. Hu, "Computational Fluid Dynamics," Fluid Mechanics, pp. 421–472, Jan. 2012, doi: 10.1016/B978-0-12-382100-3.10010-1.
6. "SOAR on Containment Thermalhydraulics and Hydrogen Distribution."
7. D. Bestion, "System thermalhydraulics for design basis accident analysis and simulation: Status of tools and methods and direction for future R&D," Nuclear Engineering and Design, vol. 312, pp. 12–29, Feb. 2017, doi: 10.1016/j.nucengdes.2016.11.010.

8. T. Höhne, E. Krepper, and U. Rohde, “Application of CFD codes in nuclear reactor safety analysis,” Science and Technology of Nuclear Installations, vol. 2010, 2010, doi: 10.1155/2010/198758.
9. S. Kliem, B. Hemström, Y. Bezrukov, T. Höhne, and U. Rohde, “Comparative Evaluation of Coolant Mixing Experiments at the ROCOM, Vattenfall, and Gidropress Test Facilities,” Science and Technology of Nuclear Installations, vol. 2007, pp. 1–17, 2007, doi: 10.1155/2007/25950.
10. “Boron Dilution in WWER Reactors.” https://www.istc.int/en/project/4F07A1883DFB4045C3256A0D002444F0 (accessed Mar. 17, 2022).
11. J. B. Joshi et al., “Design of passive safety systems for advanced reactors using CFD,” Advances of Computational Fluid Dynamics in Nuclear Reactor Design and Safety Assessment, pp. 387–485, Jan. 2019, doi: 10.1016/B978-0-08-102337-2.00005-5.

Aljasar Shojaa Ayed Ali (Jordan)
Tomsk Polytechnic University, Tomsk.

THE DIFFERENCE BETWEEN THE POND TYPE RESEARCH REACTOR IN TERMS OF DESIGN AND CONSTRUCTION

The research reactor is widely used for many purposes such as education and training, neutron activation analysis, radioisotope production, conversion effects, neutron radiography, material structure studies, neutron capture therapy [1]. Generally, the fission heat from fuel assembly is not used in a research reactor while electrical energy is produced in a commercial nuclear power plant using the fission heat of nuclear fuel. Numerous research reactors (RR) are designed as pond- type reactors. The paper is substantially demonstrated with RR of pool type and the difference between them in terms of design and construction.

Confinement of radioactive materials

Search reactors generally apply the conception of the three confinement walls. The first confinement bulkhead is assured by the reactor energy cap, -In the case of an open heart design, the alternate confinement hedge is the pool water, the pool liner and the primary circuit outside the pool. This description may be subject to review especially for people familiar with PWRs.

The third confinement hedge is assured by constraint. NS. Containment Means The structure and associated ventilation as defined by the International