BJRS

BRAZILIAN JOURNAL
OF
RADIATION SCIENCES
08-03 (2020) 01-16

SBPR
Sociedade Brasileira de Proteção Radiológica

COMPUTATIONAL SIMULATION OF A TEST FACILITY IN REDUCED SCALE FOR ANALYSIS OF BORON DISPERSION IN A PRESSURIZER OF AN INTEGRAL COMPACT AND MODULAR REACTOR

Rojas Mazaira [ab] L.Y., García [c] C.R., Dominguez [d] D.S., Gámez [a] A., González [ab] D., Rosales [c] J., Lima [ab] F.R.A., Bezerra [a] J.L., Silva [a] M.A.B., Lira [ab] C.A.B.O.

[a] *Departamento de Energia Nuclear – Universidade Federal de Pernambuco, Ave. Prof. Luiz Freire, 1000. Cid. Universitária, CEP 50740-540, Recife, PE, Brasil*

[b] *Centro Regional de Ciências Nucleares do Nordeste, Comissão Nacional de Energia Nuclear, Av. Prof. Luiz Freire, 200, CEP 50740-540 Recife, PE, Brazil*

[c] *Instituto Superior de Tecnologías y Ciencias Aplicadas (InSTEC), Universidad de La Habana. Ave. Salvador Allende No. 1110, Quinta de los Molinos, La Habana 10400, Cuba*

[d] *Universidade Estadual de Santa Cruz, Campus Soane Nazaré de Andrade, Rodovia Jorge Amado, Km 16, Bairro Salobrinho CEP 45662-900, Ilhéus, BA, Brasil*

leored1984@gmail.com

ABSTRACT

In an iPWR (integral pressurized water reactor), the pressurizer is located at the top of the reactor vessel; this configuration requires the investigation of the mechanisms adopted for the homogenization of boron. In previous work, three representative experiments were conducted in an experimental installation made of stainless steel with a volumetric scale equal to 1:200, representing one-fourth of the pressurizer of an iPWR. The test facility layout was mounted so that the test section was fed with water or saline solutions with different concentrations, representing scenarios of boration or deboration. To determine the concentration at the inlet and outlet of the test section, samples have been collected every 10 min during the experiments representing some scenarios. The main goal in this work was to investigate how well mixing during boron dilution transients in the pressurizer of a

ISSN: 2319-0612
Accepted: 2020-08-18

small modular reactor can be modelled accurately by CFD (Computational Fluid Dynamics) codes. Two CFD codes were used, the commercial ANSYS CFX and the open source CFD code OpenFOAM. The use of open source software such as OpenFOAM offers a way to CFD gain acceptance in the licensing. The results of the comparison of simulations with an experiment at the test section were presented and showed a very good agreement. It was verified that deviations are less than 4%, both codes can be used to accurately represent this phenomenon. In order to improve future experiments, the dispersion of the solution inside the test section is studied with the aim to propose positions for sensors. A first simple configuration is proposed, but some further simulations will be done to find an optimum configuration.

Keywords: **iPWR (integral pressurized water reactor), CFD (Computational Fluid Dynamics), boron dilution.**

1. INTRODUCTION

Small Modular Reactors (SMRs) are projected with modular technology, reducing expenditures on series production and allowing the fabrication in a short time. SMRs designs include a range of technologies, some of them are variants of the Generation III systems [1]. Many small modular reactor designs with distinct characteristics have been proposed or are being developed. These designs vary in their power output, physical size, fuel type, refueling frequency, siting options, and status of development [2].

Some of the advanced SMRs, as NuScale, mPower and W-SMR, share a common set of design principles like the incorporation of the primary system components in only one single vessel, the consequent increase of the primary reactor vessel, the promotion of more effective heat removal, the increase of the pressurizer volume, the relocation of components in the vessel, that facilitate the core cooling by natural convection. These features, also adopted by IRIS reactor, use the passive safety systems, guaranteeing the optimization of the installations' safety and its better operation and on the global economy aspects [3].

Generally, the pressurizer is located at the reactor vessel top in an iPWR. This configuration involves changes in the techniques such as the fabrication of a larger system, without any additional costs [4]. Coolant mixing inside the nuclear reactor is the most important inherent safety mechanism against power peaks or overcooling transients [5]. Therefore, it is necessary to study the mixing and homogenization of boron in the pressurizer liquid volume as a function of movement mechanisms, which is being considered in the development of an integral modular nuclear reactor.

For the analysis of the mixing process in the pressurizer of a small modular reactor with an integrated primary system, an experimental facility was built at the Centro Regional de Ciências Nucleares do Nordeste (CRCN-NE). This facility should allow variation of the various parameters of design and operation so that they can optimize the mechanisms of homogenization.

In a previous work [6], three experiments were conducted at the experimental setup, all at room temperature and using only an input and an output of the test section (TS). The characteristics that defined the execution of experiments and the core values are presented in Table 1.

Table 1: Parameters used in the experiments [6].

	Experiment 1	**Experiment 2**	**Experiment 3**
Scenario	Boration	Deboration	Boration
Initial concentration in TS (ppm)	0	1000	0
Concentration in the external tank (ppm)	1000	0	1500
TS Volume (m^3)	0.0405	0.0405	0.0736
Volumetric flow rate (m^3/s)	2.89×10^{-6}	2.33×10^{-6}	3.89×10^{-6}
Duration of the experiment	10800	10800	18000

The competiveness of CFD is continuously growing due the rapid developments in computer technology. CFD modeling has emerged as a useful tool in simulating the transient mixing phenomena.

To gain acceptance in the licensing world, investigations need to be underpinned by a comprehensive validation programme to demonstrate the capability of the technology and to provide results reliable enough to be used in licensing procedures. One issue that needs to be resolved is that generally the major commercial CFD vendors do not allow unrestricted access to their source code, a situation which could be unacceptable from a regulatory standpoint. The use of open source software such as OpenFOAM offers a way to circumvent this difficulty [7].

The main objective in this work was to investigate how well mixing during boron dilution transients in the pressurizer of a small modular reactor can be modelled accurately by CFD codes. Two different CFD codes, namely ANSYS CFX (commercial software) and OpenFOAM (open source software) were assessed by comparing the results in deboration process in the experiment 2 by [6].

2. THE TEST FACILITY

The test facility built for experimental investigations of boron dispersion in the pressurizer of the iPWR is located at the Centro Regional de Ciências Nucleares do Nordeste (CRCN-NE). The main parameters of test facility were determined through the combination of Fractional Scaling Analysis and local scaling [8]. The construction in a reduced scale guarantees the similarity of phenomena with a reduction in time and costs.

This facility comprehends a boration tank (BT), a dilution tank (DT), a storage tank (TA), a heat exchanger (HE), two pumps (P-1 and P-2), five flow meters and a test section (TS). About 95% of the connecting lines are made of stainless steel pipes with a nominal diameter of 3/800. For the other remaining 5%, natural polyethylene pipes were installed with the same diameter.

All connections used for the experimental set are built by HOKE GYROLOK, made of stainless steel 316, adapted to accommodate the piping. The general arrangement of the items making up the experimental setup is shown in Figure 1.

The test section (TS) is made of stainless steel with 0.092 m^3 capacity, corresponding to one-fourth of the small modular reactor pressurizer in a reduced scale of 1:200. It has two inlet orifices and two outlet orifices simulating mass flow inlets and outlets, respectively. The test section is shown the Figure 2. The boron concentration is modeled by the concentration field of a tracer solution. In this case, using sodium chloride (NaCl) as the tracer element [9].

Three experiments were conducted to evaluate the potential of the test facility to determine the concentration of the injected tracer element at the inlet/outlet of the test section. The experiments were executed without heating. In this work, the experiment 2 is used to compare with the CFD results.

In experiment 2 the test section was analyzed with an equivalent volume equal to 0.0405 m^3 by using a solution having a NaCl concentration of 1000 ppm. After establishing an equivalence between the volumetric flow in and out of the test section, a steady flow of distilled water in ST was injected. Thus, the concentration of the test section gradually decreased. The volumetric flow rate was stabilized at 2.33×10^{-6} m^3/s and the test section was fed with only distilled water. To quantify the concentration at the inlet and outlet sections, 100 ml samples were taken every 10 min interval for 180 min.

Figure 1: *The test facility.*

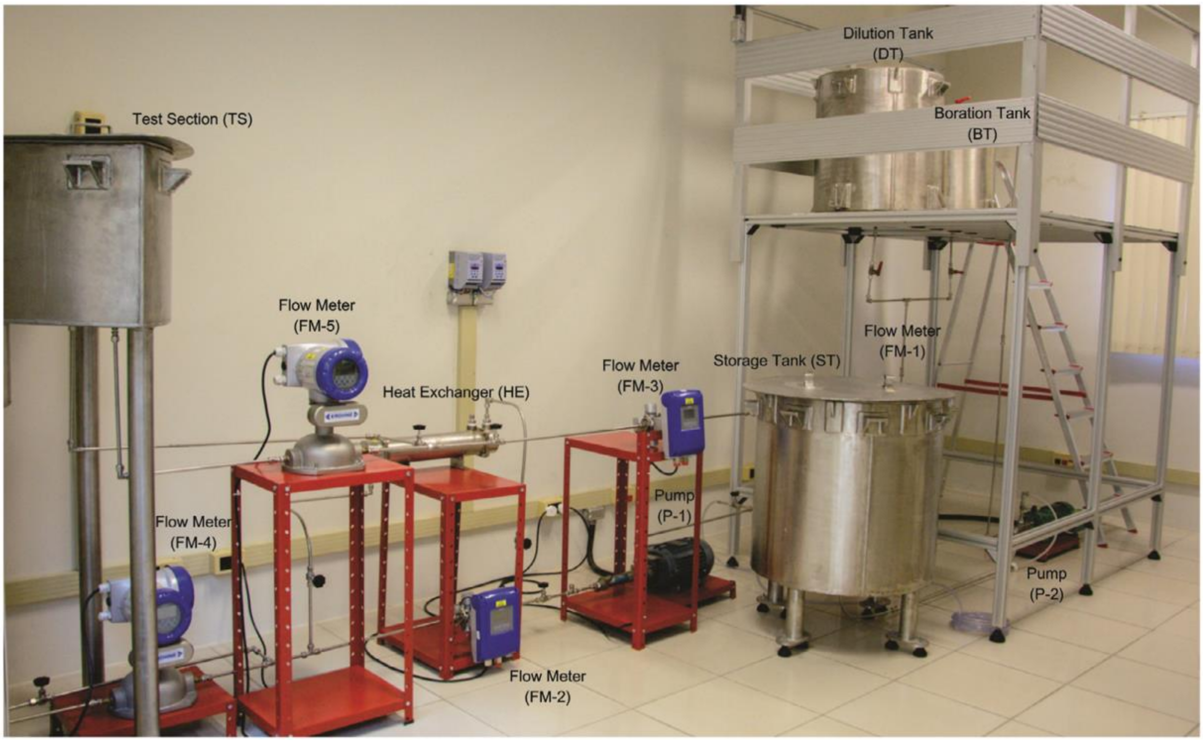


Figure 2: *The test section (TS).*

3. CFD MODEL

3.1. Main Fluid Dynamics Model

In this work, we applied the computational fluid dynamics (CFD) codes ANSYS CFX Version 19 [10] and OpenFOAM Version V5 [11]. These codes solve the conservation equations for mass and momentum (Navier–Stokes) for adiabatic simulations and, in addition, the conservation equation for energy for non-adiabatic simulations with heat transport or the conservation equations of chemical species. Turbulent flows are typically modeled by Reynolds averaged Navier–Stokes equations (RANS), which require – depending on the chosen turbulence model – up to seven further conservation equations.

In the boron dilution phenomenon, the transient behavior of the boron concentration is mainly governed by the turbulence mixing effect. Thus, the selection of the most appropriate turbulence model and grid quality relevant to the selected model are key aspects for accurate prediction. The standard k-ε turbulence model was used. It performs very well for the boron dilution phenomenon [9]. The flow was considered as incompressible. The effect of buoyancy due to gravity and density differences is taken into account.

There are situations when you want to simulate the transport of some specie in the flow. In general, one would have to employ the species transport model. This model is rather complex and only applies to compressible flows, which means that one would not be able to use it to for example simulate contamination in the water. In such cases, it might be appropriate to use the Passive Scalars model.

The Passive Scalar Transport analysis type allows to simulate the transport of a scalar quantity within an incompressible fluid flow. The core assumption of this analysis is that the species that is transported within the flow does not affect the physical properties of the flow like density or viscosity (therefore passive). This condition is usually met when the scalar is used to model relatively small concentrations of some species, for example for the transport of oxygen within a water flow. It is important to note that, scalar transport does not assume any physical dimensions for passive quantities.

For passive scalar transport simulations, diffusion coefficient of passive tracers in the fluid must be defined. In this work was used 1.5×10^{-9} m²/s, which is characteristic of NaCl in water used in the experiments.

In ANSYS CFX, passive scalar transport simulations are carried out using the transport of Additional Variables. This describes how rapidly the scalar quantity would move through the fluid in the absence of convection. In OpenFOAM, a scalar was added using the Scalar Transport Model.

In ANSYS CFX the physical time-step was chosen as 1 second. In OpenFOAM, the physical time-step was defined adaptative in such a way that the maximum Courant–Friedrichs–Lewy (CFL) number in the domain is around 10. The calculations take a similar duration in each simulation of about 12 hours, in a computer with CPU i7 3770 and with 16GB de RAM.

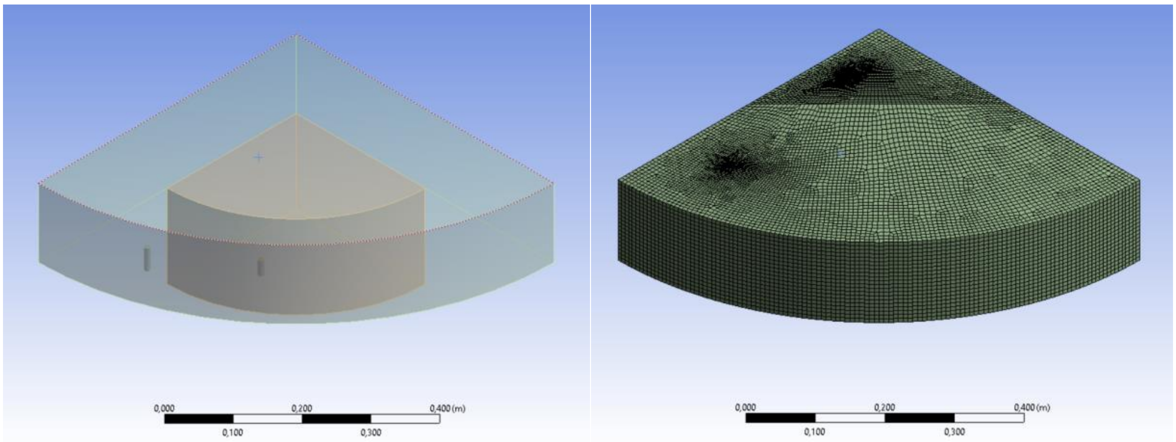
3.2. CAD Geometry and Computational Mesh. Initial and Boundary Conditions

In the CAD geometry of the test section, there are two simplifications. Firstly, it was represented just an inlet and an outlet, due to in the experiments the second inlet and outlet were no used. Secondly, only the volume occupied by the water is represented, because the air was not considered in the simulations inside the test section.

Unstructured grids consisting of hexahedral cells were generated using the software ANSYS MESHING. Two grids were generated, each with a different resolution for the cells. The total number of cells in the coarse grid and fine grid are 2.3 M and 4.8 M, respectively. The minimum orthogonal quality was 0.45, which is a good mesh quality. The comparison of the results with these two grids showed a difference of less than 1%. This means that the results are mesh independent. Figure 3 shows the CAD Geometry and mesh of the volume of water in the test section.

The initial and boundary conditions are settled as were described in Table 1[6] and in more detail in [12] for experiment 2. At the top of the domain was used a symmetry boundary condition. On that surface there is not a wall, there is the interface with air that can be modelled with the symmetry condition.

Figure 3: *CAD Geometry and mesh of the volume of water of the test section (TS).*



4. RESULTS AND DISCUSSION

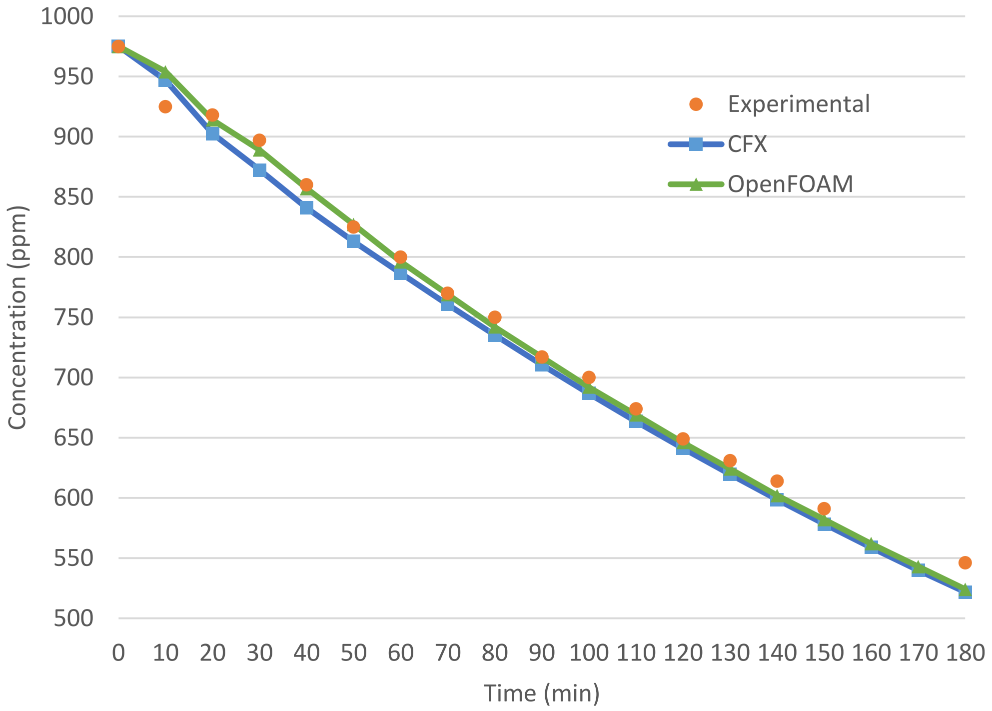
4.1. Validation Analyses

For the representation of a deboration scenario, the concentration at the outlet of the test section, the fit of an exponential function of first order and the theoretical values of concentration were presented in [6]. By comparing the experimental results with a theoretical model, it was verified that deviations are less than 5% [6].

In the simulations carried out with ANSYS CFX and OpenFOAM were monitored the concentration of the tracer at outlet every 10 min, like in the experiments. In Figure 4 are shown the comparison of the CFD simulations with experiment 2. In the experiment, the measurements at 10 minutes and 180 minutes in the experiment are a bit separated from the curve behavior. It was verified that deviations with ANSYS CFX are less than 4%, but if not considered the measurements at 10 minutes and 180 minutes it is less than 3%.

With OpenFOAM simulation it was verified that deviations are less than 3%, but if not considered the measurements at 10 minutes and 180 minutes less than 1%. This comparison shows that the results with OpenFOAM, a free CFD code, can be similar or in some cases like this, a little more accurate than with an established commercial code as ANSYS CFX.

Figure 4: *Concentration versus time at the outlet.*

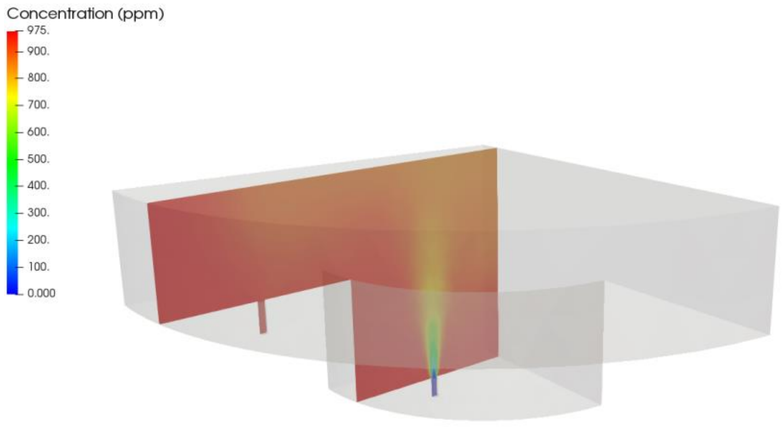


4.2. General flow behavior

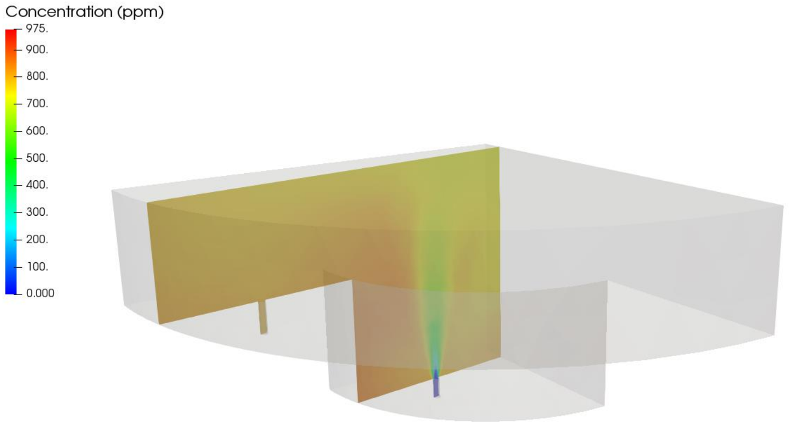
The comparison results showed that the grid effects and the predicted concentration distribution are similar with the two codes. For simplicity of discussion, only the results with OpenFOAM are presented.

Figure 5 shows the overall behavior of tracer mixing and how the dilution phenomenon occurs inside the test section. The outside walls of the test section are represented on transparent surfaces. A vertical plane that cuts the geometry such that it passes through the center of the inlet and the exit is shown. In the first image at 10 minutes, most of the test section is still occupied by water with the maximum concentration of the tracer. At 60 minutes it is observed that the concentration decreases by about 200 ppm (20%) and at 180 minutes by 450 ppm (46%).

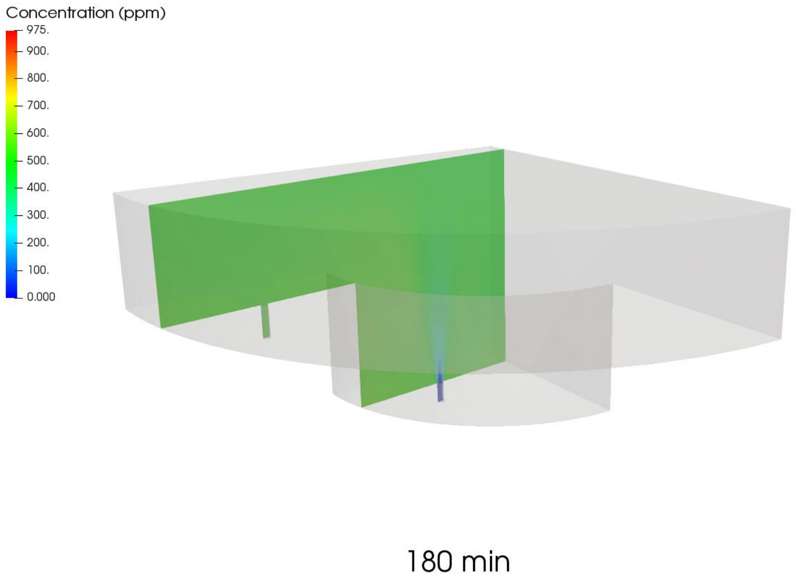
Figure 5: *Concentration distribution vs time in a vertical plane with inlet and outlet.*



10 min



60 min



The scale was set at the minimum and maximum of the whole process (Figure 5). It allows observing the evolution but does not allow to see how homogenized the inside of the test section is. In Figure 6 the scale was set at the minimum and maximum at that time. At 180 minutes, the concentration at the outlet is 530 ppm. In Figure 6 is observed that far enough from the inlet jet, the concentration is in the range from more than 450 up to 572 ppm. With this, good homogenization is achieved.

Figure 6: *Concentration distribution vs time in a vertical plane with a local scale.*

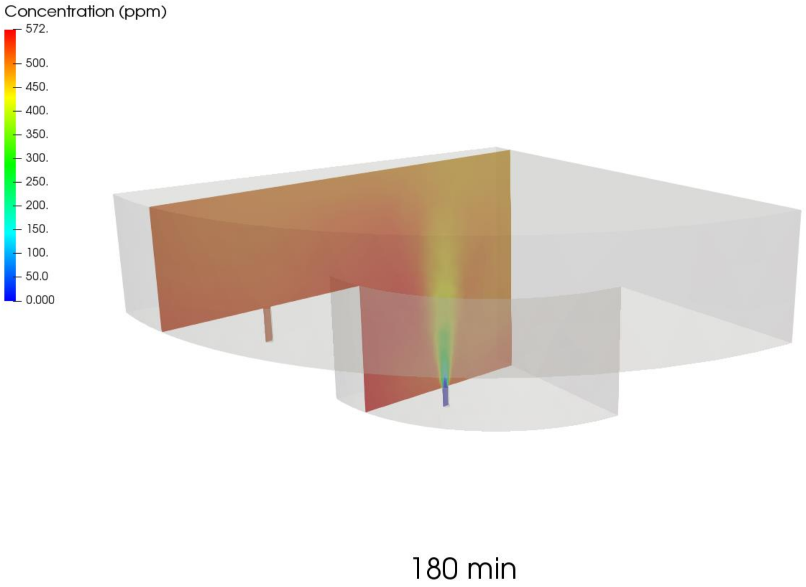
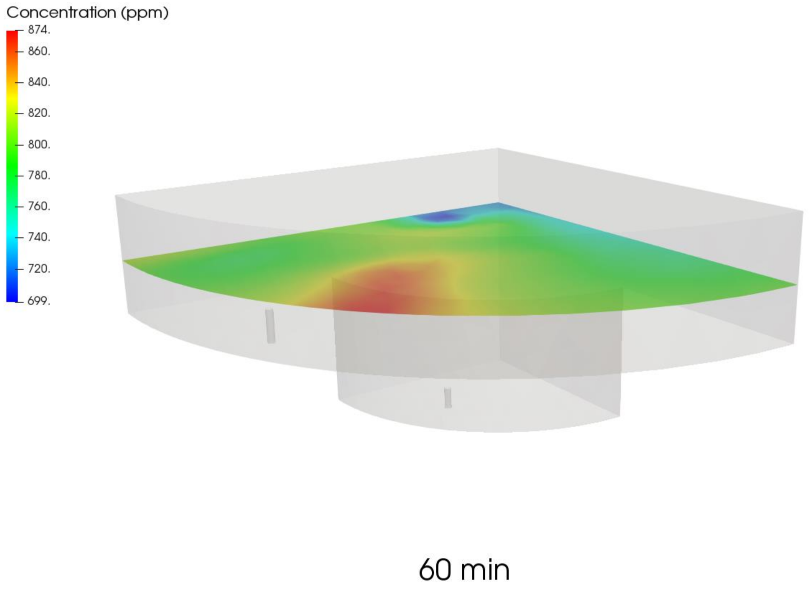
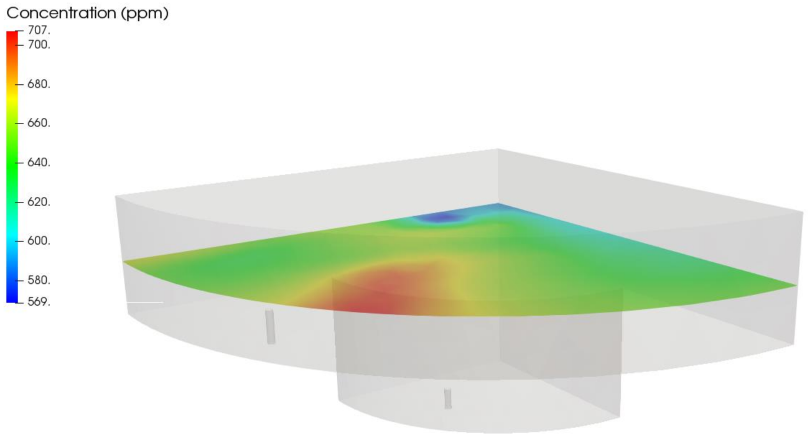


Figure 7 is represented a horizontal plane at half of the height the volume occupied by water at the up region of the test section. The scale was set at the minimum and maximum in each time. It is observed that the concentration variation decreases, but the behavior of the distribution is the same. The variation of the concentration at 180 minutes is in the range of 463 to 575 ppm. The same is shown in Figure 6. These images confirm the circulation pattern of the flow asymmetric due to the use of only one inlet and one outlet, that are not in the center. The highest tracer concentration will be close to the outer walls in both the lower and upper regions.

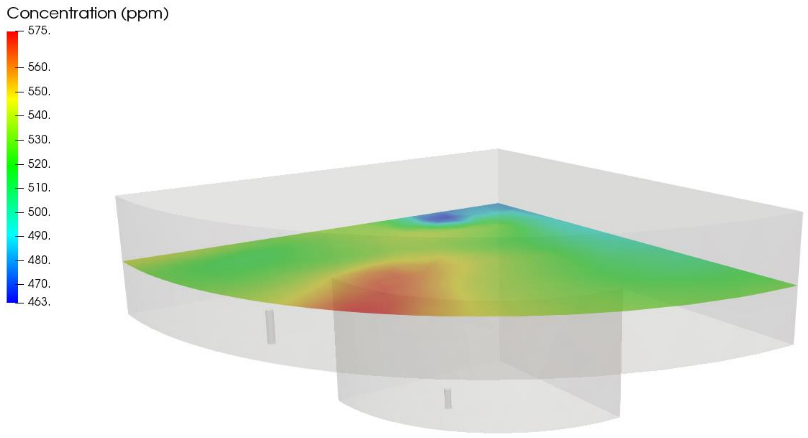
From these preliminary results, it is recommended to place at least three internal sensors to measure the concentration of the tracer. Considering the plane presented in Figure 7, a sensor is recommended in the center of the quarter of circumference and another two located with 15 degrees between them and the plane lateral walls. This configuration can be carried out more experiments using the two inlets and outlets. To choose the best height where the sensors could be placed is needed the simulations of the other two experiments already done. The best alternatives are if these sensors can be repositioned for different experiments or use nine in three different heights. Some further works will be done to study the optimum configuration.

Figure 7: *Concentration distribution versus time in a horizontal plane.*





120 min



180 min

5. CONCLUSION

The conclusion should include the main points of the article, reflecting the problem statements and the objectives. The conclusion may also summarize the implications of your findings and suggest further researches.

The project for the test facility aimed to provide relevant data for boron homogenization phenomena in the pressurizer of a compact modular reactor. In this work, the results of the simulation with two CFD codes are presented. To investigate how well mixing during boron dilution transients an experimental facility was built at the Centro Regional de Ciências Nucleares do Nordeste (CRCN-NE), representing the pressurizer of a small modular reactor. The comparison shows that the results with OpenFOAM and ANSYS CFX are in agreement with the experiment of deboration. They are more accurate than the theoretical model used in [6] to compare with experiments. Especially the OpenFOAM results, which is an open source software, with some great advantages for use in nuclear thermohydraulic.

From the results with OpenFOAM, an analysis of the distribution of the concentration inside the test section was carried out. The aim was to observe the mixing behavior and propose positions to place sensors for future experiments. An asymmetric circulation pattern of the flow due to the use of only one inlet and one outlet was shown. The recommendation is to use at least three sensors placed in the same plane. One at the center and the others in the circumference equally spaced between them and with the plane lateral walls. To best select, the height of the locations is needed for the simulations of the other two experiments already done.

Several further experimental and simulation works will be done to study the boron dilution phenomenon in the pressurizer of a compact modular nuclear reactor.

ACKNOWLEDGMENT

The authors thank the Brazilian research funding agencies, CNPq (Conselho Nacional de Desenvolvimento e Pesquisa), CAPES (Coordenação de Aperfeicoamento de Pessoal de Nível Superior), FACEPE (Fundação do Amparo a Ciência e Tecnologia do Estado de Pernambuco) and CNEN (Comissão Nacional de Energia Nuclear) for the support to this project.

REFERENCES

[1] Carelli, M. D.; Garrone, P. ; Locatelli, G.; Mancini, M. ; Mycoff, C. ; Trucco, P. ; Ricotti, M. E. Economic features of integral, modular, small-to-medium size reactors. **Prog. Nucl. Energy**, v. 52, no. 4, p. 403–414, 2010.

[2] Carelli, M. D.; Ingersoll, D. T. **Handbook of Small Modular Nuclear Reactors**. Woodhead Publishing, Cambridge, UK, 2015.

[3] Ferri, R. ; Achilli, A. ; Cattadori, G. ; Bianchi, F. ; Luce, A. ; Monti, S. ; Meloni, P.; Ricotti, M. E. SPES-3: the integral facility for safety experiments on small and medium sized reactors. **European National Conference**, Manchester, United Kingdom, 2012.

[4] Barroso, A. C. O. ; Baptista, B. D. F. ; Arone, I. D.; Macedo, L. A.; Sampaio, P. A. B.; Moraes, M. IRIS Pressurizer Design Studies. **ICAPP '03**, Córdoba, Spain, 2003.

[5] Rohde, U.; Höhne, T.; Kliem, S.; Hemström, B.; Scheuerer, M.; Toppila, T.; Aszodi, A.; Boros, I.; Farkas, I.; Mühlbauer, P.; Vyskocil, L.; Klepac, J.; Remis, J.; T. Dury. Fluid mixing and flow distribution in a primary circuit of a nuclear pressurized water reactor-Validation of CFD codes. **Nucl. Eng. Des.** 237, no. 15-17 SPEC. ISS., p. 1639–1655, 2007.

[6] Nascimento, S. R. V.; Lira, C. A. B. O.; Lapa, C. M. F.; Lima, F. R. A.; Bezerra, J. L.; Silva, M. A. B.; Otero, M. E. M.; Hernández, C. R. G. Implementing a test facility in reduced scale for analysis of boron dispersion in a pressurizer of an integral compact and modular reactor. **Ann. Nucl. Energy**. 105, pp. 259–265, 2017.

[7] Smith, B. L. Assessment of CFD Codes Used in Nuclear Reactor Safety Simulations. **Nucl. Eng. Technol**. 42, no. 4, p. 339–364, 2010.

[8] Da Silva, M. A. B.; Brayner de Oliveira Lira, C. A.; Carlos de Oliveira Barroso, A. Determination of a test section parameters for IRIS nuclear reactor pressurizer**. Prog. Nucl. Energy**, 53, no. 8, pp. 1181–1184 (Nov. 2011).

[9] Cho, Y. J.; H. Y. Yoon. Numerical analysis of the ROCOM boron dilution benchmark experiment using the CUPID code. **Nucl. Eng. Des**. 341, no. October 2018, p. 167–175, 2019.

[10] ANSYS Inc., **CFX R19 User's Guide.** ANSYS Inc., Canonsburg, 2018.

[11] **OpenFOAM Foundation**, http://www.openfoam.org, 2018.

[12] Nascimento, S. R. V. Implementação de uma montagem experimental em escala reduzida para análise da dispersão de boro em um pressurizador de um reator modular compacto e integral. UFPE, 2016.