Experimental study of coolant flow mixing processes in a pressure chamber model of nuclear reactor

S M Dmitriev, A E Khrobostov, A V Ryazanov, A A Dobrov, D V Doronkov, M A Legchanov, A N Pronin, E V Rubtsova and D N Solntsev

Nizhniy Novgorod State Technical University n.a. R.E. Alekseev, Minin St., 24, Nizhniy Novgorod, 603950, Russia

A_V_Ryazanov@rambler.ru

Abstract. The paper presents results of experimental research, the purpose of which was to simulate the phenomenon of mixing of loop coolant flows inside the model of the pressure chamber of the water-cooled reactor. The study was conducted at the high-pressure aerodynamic test bench of NNSTU n.a. R.E. Alekseev. The scale model consisted of elements characteristic of water-cooled nuclear reactors (lowering annular channel, lower pressure chamber). Experimental studies were carried out in the range of Reynolds numbers from 20,000 to 50,000. The axial velocity field was studied at the entrance to the reactor core simulator, using a pneumometric research probe. The temperature field was simulated using the impurity diffusion method by introducing a contrast tracer into one of the loops of the model. Propane was used as a passive tracer. Tracer concentration at the inlet core simulator allowed to evaluate the degree of flow mixing and thereby studied the direction of coolant flow. As a result of the studies, the spatial distribution of the tracer concentration in the coolant flow in the lower annular channel, as well as in the lower pressure chamber, was determined. It was shown that in the lower pressure chamber, an axial central vortex influences the mixing intensity. The estimation of the parameters of mixing of the passive tracer in the model of reactor pressure was given.

1. Introduction

For regions with decentralized energy supply, it is advisable to place nuclear reactors as part of small sized nuclear power plants. Promising projects of such stations are based on nuclear marine propulsion technologies. Nuclear reactors of such stations should have maximum maneuverability and reliability characteristics, both in terms of safety and reducing the cost of electricity supplied to the consumer, therefore, these plants are subject to the highest requirements for the above factors [1].

One of the consequences of operation of a power unit with partially switched off heat exchange equipment is the occurrence of non-isothermal flows of a heat carrier with various physical properties and their mixing in the pressure chamber of the reactor, which can lead to failure. Moreover, a conservative approach to safety justification does not imply taking into account the influence of such phenomena, which significantly limits the choice of nuclear reactor operating modes.

An alternative safety justification method is the calculation analysis of both new structural elements of a nuclear reactor and operating modes of a reactor facility. In this case, the substantiation of the thermotechnical reliability of the core is impossible without accurate input data of the velocity, temperature or impurity concentration fields in the reactor pressure chambers [2].
Nowadays level of technical development, as well as safety requirements, do not make it possible to conduct full-scale experiments at a nuclear reactor to justify its thermal engineering reliability. A comprehensive study of this physical phenomenon in water-cooled nuclear reactors will create a unified database of precision experimental data. This database is one of the necessary conditions for the validation of computational fluid dynamics codes, with the help of which it is possible to justify the thermotechnical reliability of the small sized nuclear power plant in new operating conditions in order to increase the economic attractiveness of the nuclear power plant project.

Research in this field of science presented in open sources has a number of drawbacks, among which the lack of experimental studies of the influence of individual physical quantities on turbulent mixing processes can be noted. Also, studies have not been conducted to study the scalability of the results of CFD calculations for full-scale facilities.

In view of the structural complexity of nuclear facility equipment, a number of research carried out by Russian scientists were devoted to the study of turbulent mixing of flows in individual nodes of a reactor unit [3-4].

Carrying out an aerodynamic experiment using an isothermal model of pressure chamber of a water-cooled nuclear reactor makes it possible to study the features of the flow and to simulate the temperature of the coolant or impurity concentration at the entrance to the active zone during mixing of loop flows.

The aforementioned determines the high degree of relevance of studies of mixing processes of loop coolant flows in the lower chamber of a nuclear reactor.

2. Experimental test bench

The study of the hydrodynamic features of the mixing processes of the loop coolant flows in the lowering chamber of a nuclear reactor was carried out at the high-pressure aerodynamic test bench of NNSTU n.a. R.E. Alekseev [5]. The bench (Fig. 1) is an open-loop aerodynamic circuit which includes: high-pressure fan, buffer tank, distribution system of pipelines, experimental model, and measuring complex.

![Figure 1. General view of the test bench](image)

The test model is a simplified scale model that has geometry characteristic of a water-cooled nuclear marine propulsion (such as icebreaker). Movement of the coolant is organized as follows: radial coolant supply through the nozzles, movement along the lowering channel to the lower pressure chamber formed by the elliptical bottom and the lower part of the core simulator, ascent through the core simulator channels, followed by axial exit from the upper drain chamber. Structurally, the test model has four inlet nozzles, separated by 90° from each other. The outer diameter of the lowering chamber is 400 mm. The core simulator is a set of throttled vertical channels that simulate the pressure drop in the core (Fig. 2).
The circulation of the coolant is provided by one high-pressure fan, therefore, at the outlet of the buffer tank, a distribution system of circles is organized. All loops have the same geometry, which allows you to maintain and control the equality of costs in each loop at the entrance to the test model. The “instability” of experimental mode parameters does not exceed 3%.

To study the mixing process of loop coolant flows in pressure chamber model of a nuclear reactor, two regions of the model were selected as characteristic regions: lowering annular channel, lower pressure chamber. The measurements were carried out in 3 control planes of the lowering chamber (evenly located in the region from the inlet pipes of the experimental model to the flow inlet to the lower pressure chamber), as well as inside the channels of the core simulator (Fig. 3).
3. Measuring system

To measure the speed of the coolant inside the channels of the core simulator, a Pitot-Prandtl tube was used. Control measurements of the flow velocity in each loop at the entrance to the experimental model were carried out using an E+E Omniport Logprobe 60 air velocity probe (accuracy ± (0.2 m/s+2%Q)).

During experiment, the maximum air flow through the test model reached 0.4 m³/s (Re=50,000 mode), with an error in determining the flow not exceeding 3%. The pressure in the test model was up to 4 kPa.

To study the characteristics of the coolant flow inside the pressure chamber model of a nuclear reactor, the impurity diffusion method was used. This method is based on the registration of the mass flow for some portable substance. Propane was chosen as a contrasting tracer. During the experimental study, a gas supply was arranged in one of the loops of the test model. The values of the volume concentration of propane in the air flow did not exceed 1200 ppm (tracer injection pipe), which is much lower than the concentration of the formation of an explosive mixture (minimum 17000 ppm). Moreover, such values can be measured with satisfactory accuracy (± 15 ppm) using the ADK-03R measuring complex based on the infrared method for measuring the concentration of hydrocarbons. The gas supply was controlled by an El-Flow F-201CV flowmeter (the error in maintaining the flow rate is 0.5%).

The use of the impurity diffusion method in experimental studies allows us to study the redistribution of the temperature field or the concentration of the tracer in the coolant flow using isothermal aerodynamic models. Adding a contrast tracer to the coolant flow is widely used in studies of thermohydraulic and mass transfer processes [6-9]. The mathematical basis for the application of this method is the same form of the differential equations of heat and mass transfer [10]. In a moving highly turbulent fluid flow, the temperature field ranged from the walls, like the impurity concentration field, depends on the velocity field, which is described by the equations of hydrodynamics, and large vortices, the generation of which is due to the shape of the channel.

Comparing the processes of turbulent and molecular transfer, it should be noted that the pulsation component of the velocity is much less than the speed of molecular motion, but the length of the mixing path is much greater than the mean free path of the molecules. In addition, mixing in a turbulent flow is due to the pulsating motion of particles having a significant mass; therefore, in a turbulent flow, the phenomena of turbulent transfer occur much more intensively than the phenomena of molecular transfer. Turbulent diffusion is much more intense than molecular diffusion, turbulent thermal conductivity is much greater than molecular thermal conductivity.

Based on these judgments and the theory of approximate modelling, the assumption was made that in the phenomenon under study the bulk of the substance and energy is transferred due to the turbulent properties of the flow, while the molecular transfer is negligible.

4. Experimental research

During experiment, data of coolant velocity distribution and impurity concentration in the characteristic regions of the test model were obtained. Results were presented in dimensionless form:

1) dimensionless relative tracer concentration \( \phi = \frac{c_i - c_{\text{min}}}{c_{\text{max}} - c_{\text{min}}} \) (maximum and minimum concentration of the tracer, \( c_i \) – measured current concentration of the tracer);

2) dimensionless velocity \( \langle w \rangle = \frac{W_{ch}}{W_{av.ch}} \) (the ratio of the axial velocity in the channel to the average flow rate through the core simulator).

At the first stage of research, experiment was carried out of the nature of the motion of coolant flow when it passing through the lowering chamber from inlet pipes to the core simulator.
In all experimental modes, a spot of a contrast tracer was found in the region opposite to its input, which is a consequence of the presence of a swirl of the coolant flow. In this case, the lowering chamber makes the largest contribution to the spiral motion of coolant; when moving inside, flow deviates by an angle of about 120° (Re = 20,000, Fig. 4a) and an angle of 170-180° (other experimental modes Fig. 4b,c).

Figure 4. Concentration map in the lowering chamber of the test model (left), in the lower pressure chamber (right) a) Re=20 000; b) Re=30 000; c) Re=50 000
This cartograms make it possible to determine that the coolant flow from the circulation loop with the tracer immediately at the entrance to the lowering annular channel turns out to be shifted by an angle of ~ 30°, which is probably due to an impact on the inner wall of the annular chamber. Further downward movement takes place in a spiral, while the intensity of the twist remains constant throughout the lower part of the model. In addition, we note that for all experimental modes, maximum width of the contrasting jet is in the range of 20°.

It also should be noted that distribution of the tracer concentration in the flow the angle of rotation of the contrast spot in the lower pressure chamber of the reactor at all Reynolds numbers is in the range of 50-60°.

More over complete mixing of the contrasting tracer in the test model did not occur, as evidenced by a significant difference in the relative concentration at the entrance to the core simulator (Fig. 5). For the Re=30,000 mode, the minimum relative concentration was found in channel No.8 and amounted to 0.05, the maximum relative concentration was 0.44 and was locally located in channel No.12.

**Figure 5.** Map of relative concentration at the entrance to the core simulator (Re=30,000 mode)

A comparison of the concentration distribution in the lower part of the lowering chamber and immediately before entering the core simulator indicates that convective concentration transfer in the lowering chamber of the test model has a more significant effect on the equalization of the studied field characteristics than turbulent transport in the lower pressure chamber.

An analysis of the obtained experimental data made it possible to construct a detailed picture of the flow through the channels of the core simulator. In addition to the torus vortex, which is caused by a sharp turn of the flow when leaving the lowering chamber to a large volume of the pressure chamber, a central vortex with a vertical axis can form at the entrance to the core [11].

In the distribution of the coolant through the channels of the core simulator, two flow regions are clearly observed (Fig. 6):

- flow rate below the average flow rate \( w_{ch} < w_{av,ch} \) - central 7 channels;
- flow rate above the average flow rate \( w_{ch} > w_{av,ch} \) - peripheral row of channels.
Figure 6. The distribution of axial velocities along the channels of the core simulator Re = 50 000 mode (map of relative axial velocities (left), absolute axial velocity diagram (right))

This distribution of coolant flow through the channels of the core simulator of the test model is a consequence of the redistribution of the pressure field in the lower part of pressure chamber. A significant underestimation of the total pressure at the entrance to the core simulator could be caused by the presence of a large central vortex.

The generalized results of experimental studies are summarized in the table and are presented below in Table 1.

Table 1. The results of experimental studies

| Mode number | Flow rate through the model, Q, m³/s | Tracer rate through the model, Q, l/min | Re number 10³ | The angle of rotation of the flow in the lower pressure chamber, ° | The angle of rotation of the flow in the lower pressure chamber, ° | Maximum relative concentration in the pressure chamber (Channel number) | Minimum relative concentration in the pressure chamber (Channel number) |
|-------------|--------------------------------------|------------------------------------------|--------------|-------------------------------------------------|-------------------------------------------------|-------------------------------------------------|-------------------------------------------------|
| 1           | 0.16                                 | 2                                        | 20           | 120                                             | 60                                              | 0.44 (14)                                       | 0.08 (8)                                        |
| 2           | 0.24                                 | 3                                        | 30           | 180                                             | 60                                              | 0.44 (12)                                       | 0.05 (8)                                        |
| 3           | 0.40                                 | 5                                        | 50           | 180                                             | 60                                              | 0.47 (12)                                       | 0.09 (8)                                        |

5. Conclusion

Analysis of the results of turbulent mixing coolant flows in the downcomer of the test model showed that the swirling of the flow occurs mainly in the downcomer and for different experimental modes was an angle of 120-180°. Moreover, in the lower pressure chamber, regardless of the value of the Reynolds criterion, the flow is additionally twisted by 60°.
Due to the fact that the main mixing of the loop flows is caused by convective turbulent transfer in the lower chamber of the test model, it is advisable in reactor unit to place additional mixing devices in the zone of the lower annular channel, and not in lower pressure chamber.

Experimental studies have shown that, with an increase in the Reynolds number from 20,000 to 50,000, the phenomena of coolant turbulent mixing does not change significantly (maximum and minimum relative concentrations, the width of the jet, the rotation of the flow in the lower part of pressure chamber), however, the swirl angle of the coolant flow changes the lowering chamber (especially pronounced during the transition from the Re=20,000 to 30,000 mode). Therefore, it is expedient to conduct studies with higher Reynolds numbers to identify the scalability of the results of studying loop mixing processes. In this regard, NNSTU n.a. R.E. Alekseev created a large-scale experimental test bench with a water coolant [12], on which similar studies will be carried out with the same and large Reynolds numbers, as well as on an enlarged pressure chamber model.

References
[1] Mitenkov F M [e.a.] 1993 Cost-effectiveness of small-sized nuclear power plants vol.5 (Moscow: Atomic Energy) pp 333-336
[2] Dmitriev S M [e.a.] 2012 Experimental studies of the hydrodynamic and mass transfer characteristics of the coolant flow in the VVER TVSA vol 5 (Moscow: Atomic Energy) pp 252-256
[3] Bolshuhin M A [e.a.] 2014 Thermographic study of turbulent pulsations of water during non-isothermal mixing vol 5 (Novosibirsk: Autometia) pp 75-83
[4] Vasilev A Y [e.a.] Turbulent convective heat transfer in an inclined tube filled with sodium vol 9 (St.Pereburg: Technical Physics) pp 45-49
[5] Barinov A A [e.a.] 2018 Description of the experimental studies of coolant mixing flows in the reactor pressure vessel vol 1128 (IOPScience: Journal of physics: conference series)
[6] Gabrianovich B N, Delnov V N 1994 Investigation of coolant mixing in distribution systems of fast reactors and VVER vol 5 (Moscow: Atomic Energy) pp 340-344
[7] Gabrianovich B N, Delnov V N, Rukhadze V K 1995 Methodology for studying the processes of mixing and the time of movement of the coolant in heat exchange equipment using a halide tracer vol 7 (Moscow: Thermal energy) pp 60-63
[8] Dmitriev S M [e.a.] 2009 Experimental studies of the efficiency of mixing coolant to justify the choice of the optimal design of TVSA for implementation in VVER-type reactors vol 1 (Obninsk: Izvestiya vuzov. Yadernaya Energetika pp 88-97.
[9] Dmitriev S M [e.a.] 2020 Investigation of coolant local hydrodynamics in the mixed core of the VVER reactor vol 6 (Minsk: Energetika. Proceedings of CIS higher education institutions and power engineering associations) pp 120-126
[10] Snegirev A Y 2009 High performance computing in technical physics. Numerical modeling of turbulent flows (St.Peterburg: Politechnic)
[11] Mitrofanova O V 2010 Hydrodynamics and heat and mass transfer of swirling flows in the channels of nuclear power plants (Moscow: Fizmatlit)
[12] Dmitriev S M [e.a.] 2018 Experimental researches of turbulent mixing processes in the main equipment of NPP vol 3 (Obninsk: Problems of atomic science and technology. Series: nuclear and reactor constants) pp 120-126