ISSN 1392–1207. MECHANIKA. 2014 Volume 20(5): 471–475

Numerical simulation of foam flow through RBMK-1500 control rod channel

M. Trepulis*, **J. Gylys****
**Kaunas University of Technology, K. Donelaičio g. 20, LT-44239 Kaunas, Lithuania, E-mail: marius.trepulis@ktu.edu*
***Kaunas University of Technology, K. Donelaičio g. 20-225, LT-44239 Kaunas, Lithuania, E-mail: jonas.gylys@ktu.lt*

crossref http://dx.doi.org/10.5755/j01.mech.20.5.7165

1. Introduction

The foam flow phenomen on the forced convection, even with laminar flow, is complex and hard to develop analytically. Only key to the problem is experimental results and numerical solution. Heat transfer coefficient is very important parameter because it determines the rate of heat transfer. This paper uses two methods to determine this parameter: experiment and CFD (computational fluid dynamics) simulation. Experiment was made for transient-turbulent convective heat transfer of macro foam in vertical cylindrical tube. CFD simulation in CFX was made and the experimental results have proven to be very useful for the validation of CFD calculations of foam flow through a heated cylindrical tube

There are two main approaches for the simulation of multiphase flow, namely the Euler–Lagrange method which considers the bubbles as individual entities tracked using trajectory equations, and the Euler–Euler method which is based on two-fluid model which assumes the gas and liquid phases to be interpenetrating continua. From computational considerations, the Euler–Euler approach is more economical and commonly used.

After the numerical simulation proved the experimental results, RBMK-1500 control and protection channel cooling with macro foam was modeled. ANSYS CFX software is a general purpose fluid dynamics simulation program has been developed over more than 20 years and is widely used in solving complex two-phase flow situations. In this work the foam flow channel geometry mesh was created using ANSYS MESHING TOOLS. The foam flow and heat transfer were simulated by using ANSYS CFX 14.5 software package.

2. RBMK-1500 reactor

RBMK-1500 reactor, which previously operated on Ignalina NPP site, is a multichannel boiling water and graphite-moderated reactor. The graphite blocks are assembled within the inner cavity of the reactor on a supporting metal structure. The stack can be visualized as a vertical cylinder, made up of 2488 graphite columns, constructed from various types of graphite blocks with openings inside. These openings are used for fuel channels, control rod channels, and a few instruments associated with the reactor core [1]. During nuclear fission 95% of the generated energy is released in the fuel element and an additional 5% is released in the graphite during neutron moderation and gamma absorption. One of the principal distinguishing characteristics of the RBMK-type reactor is that each core fuel assembly is housed in an individual pressure tube (Fig. 1). Pressure in channels with control rods is always close to atmospheric.

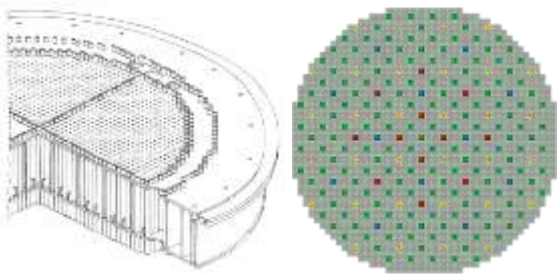


Fig. 1 RBMK-1500 reactor core and control rod channels (CPS) distribution

The CPS (Control and Protection System) channels are cooled by an independent water circuit provided with its own pumps and heat exchangers. The cooling water is supplied to the channels from above, and flows over the exterior and interior casings of the absorber rods. In this process, the water is heated from 40°C to a temperature of 70°C. During reactor operation, regardless of the position of the control rod, the inside of the channel is filled with water. However, after the Chernobyl accident, filling water was changed into the cooling with water film. This allows, in the event of an emergency, the control rods to move faster through the channel. Another alternative to water film could be a control channel cooling with foam. Easily disrupt foam structure could ensure the free movement for the control rod across the channel.

In the case of total electrical energy disruption in nuclear power plant, it would be possible to generate foam from emergency air tanks. On the assumption that it is necessary to maintain CPS channels emergency cooling for one hour without electrical energy, the necessary air consumption would be 70 m^3/h for one CPS channel cooling, in case the foam flow speed is 1m/s. For all 211 CPS channels it would be 14770 m^3 of air for one hour. The volume can be reduced by compressing air. The storage volume for a compressed gas can be calculated using Boyle's law:

$$p_a \times V_a = p_c \times V_c,$$

where p_a is atmospheric pressure, V_a is volume of gas at the atmospheric pressure, p_c is compressed pressure, V_c is volume of the gas at the compressed pressure. According the formula, if the air is compressed to 2 MPa, it should be enough around 740 m^3 volume tanks for all 211 CPS emergency cooling for one hour. Detergent solution flow doesn't need electrical power, it flows downward because of gravity. It is necessary 60 m^3 volume tank of solution in generating foam for all 211 CPS cooling for one hour. It could be used natural gas flame nozzles to destruct the used foam.

3. Experiment

Fig. 2 shows a schematic of the cross-section of the test section. The calculation design is similar to that used by Warrier et al. 2002, [2] to study boiling heat transfer in mini-channels except that there is only one channel and it is heated straight with electrical current. Cylindrical tube stays in vertical position [3] and is heating with transformer. The wire is fitted to the tube by copper bended bus. The buses are soldered to the tube for better tube - bus contact. Cylindrical tube is made from stainless steel 0.5 mm thick with active heating height of 0.8 m. All outer side of the tube was covered with insulation. The flow channel has a hydraulic diameter D_h of 0.15 m defined based on the wetted perimeter. The inlet and outlet temperatures are measured by type-E thermocouples. Moreover, to measure the surface temperature along the stainless steel tube, twenty eight type E thermocouples were soldered to the surface. There are four thermocouples per axial location and the temperature readings at each axial location did not differ by more than ±0.2°C for all heat transfer experiments. The seven axial location measured from the entrance are 0.1, 0.2, 0.3, 0.4, 0.5, 0.6 and 0.7 m. The average of four temperatures is taken as the temperature of the heated surface at that axial location.

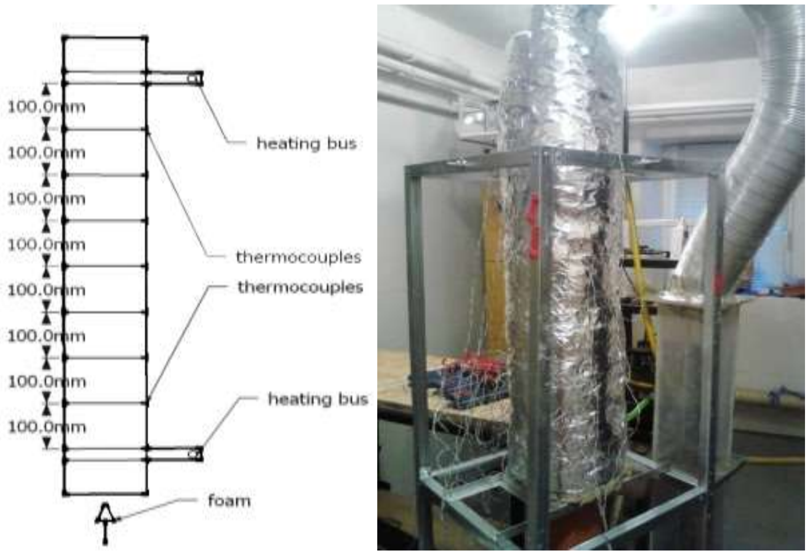


Fig. 2 Schematic cross-section of the test section and experimental setup

As mentioned by Warrier et al. (2002) [2], it was not possible to measure the fluid flow temperature inside the channels because the thermocouples were in contact with the channel walls leading to a higher temperature reading. Instead, the local fluid temperature is computed using the local energy balance equation. All the thermocouples readings are recorded every second by a Picotech TC 08 data acquisition system connected to a computer.

During the experiment it was observed that foam structure keeps statically stable if the air flow and detergent solution flow isn't interrupted.

4. Simulation of experiment using ANSYS CFX

The finite-volume method (FVM) is a method for representing and evaluating partial differential equations in the form of algebraic equations [4]. Similar to the finite difference method or finite element method, values are calculated at discrete places on a meshed geometry. "Finite volume" refers to the small volume surrounding each node point on a mesh. In the finite volume method, volume integrals in a partial differential equation that contain a divergence term are converted to surface integrals, using the divergence theorem.

These terms are then evaluated as fluxes at the surfaces of each finite volume. Because the flux entering a given volume is identical to that leaving the adjacent volume, these methods are conservative. Another advantage of the finite volume method is that it is easily formulated to allow for unstructured meshes as shown on Fig. 3. The method is used in many computational fluid dynamics packages.

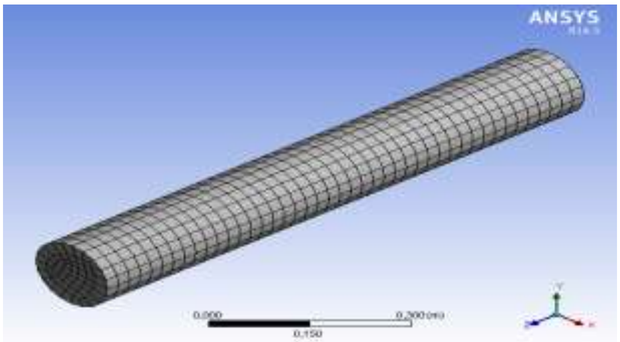


Fig. 3 Geometry of the grid formed for numerical analysis

In this paper, the solution method of approach is chosen when the two-phase flow is formed. The foam is described as a medium made up of the individual phases of the solution and air mixture. Obtained simulation results are shown in the Figs. 4 and 5. Since the experimental setup validation has been made with the air it was first developed numerical model when the heated channel is cooling with air flow. Simulation results are presented in

Fig. 4. It is easy to notice that the air temperature varies not only along the channel, but it is different in different cross-sectional areas.

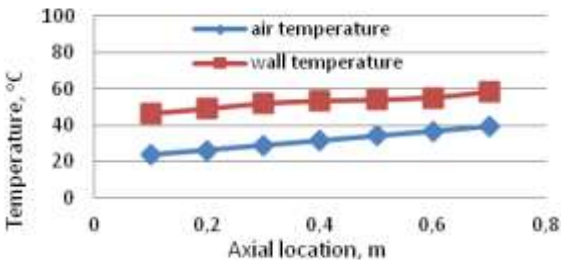
In order to calculate the average theoretical channel outlet air temperature, it is necessary to know the heat loss to the environment. Losses are found from the energy balance equation:

$$q_{channel}=m_{air}\times c_{p,air}\,(T_{air,outlet}-T_{air,inlet}), \quad (1)$$

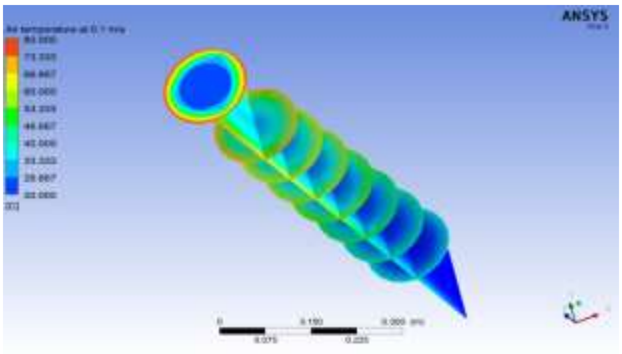
$$q_{loss}=q_{total}-q_{channel}, \quad (2)$$

where q_{total} is the total power input expressed by Joule's law $q_{total}=UI$. Voltage U is measured in volts and I current is measured in ampere. The actual heat input into the test section is $q_{channel}$. The mass flow rate is m_{air} and $c_{p,air}$ is the specific heat of air.

Experiment and simulation (Fig. 4) airflow output of the channel average temperature value differs by no more than 3°C (46°C and 43°C), which corresponds to less than 7% deviation, so it can be concluded that the simulation of foam flow results reliability should also be good. Since the wall temperature varies along the channel, for initial conditions of the simulation was used not the wall temperature, but the heat flow for channel surface area.



a



b

Fig. 4 Air temperature variation along the channel during: a - experiment; b - simulation

After agreement with the numerical simulation results reliability in the case of air-cooled channel, the numerical model of 0.97 void fraction foam was made. Simulation results are presented in Fig. 5. Graphics show how foam temperature profile varies along the heated channel.

In addition, the temperature profile depends on the foam flow rate. Since the experimental setup length to diameter ratio is relatively low (80/15 = 5.33), it can be argued that a well-established treatment temperature is reached only at the end of the experimental channel in the case of foam flow speed of 0.2 m/s.

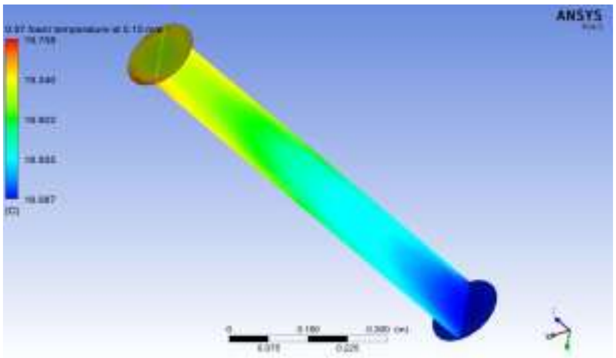


Fig. 5 Foam temperature variation along the channel (foam flow velocity of 0.15 m/s)

5. The experiment and simulation results comparison

Heat transfer coefficient of axial cross-sections increase (Fig. 6), moving away from the channel inlet to the channel top can be explained by the fact that the foam cool down the heated surfaces and increase in temperature. Decrease in the temperature difference between the channel wall and the foam directly influences the heat transfer coefficient increase.

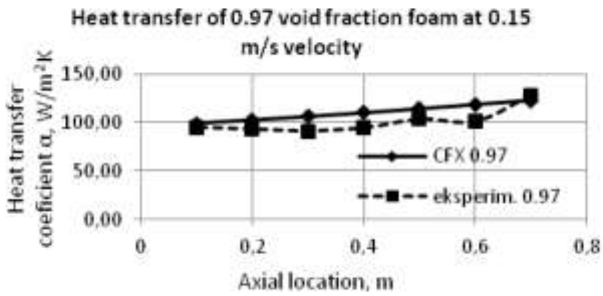


Fig. 6 The average heat transfer coefficient variation along the axial channel locations of foam flow

The difference between experimental and simulation results can be explained by the fact that the modeling assumed that the heated cylindrical surface has the same constant temperature. Meanwhile, the experiments showed that the temperature depends on the flow rate, in addition changes along the channel (for example, at 0.15 m/s velocity, temperature deviation of the individual axial locations was 0.6°C) It also should be noted that the experimentally determined axial location average heat transfer coefficient, which is found using the arithmetic average of the four thermocouples, is about 10W/m^2 K smaller than that obtained with numerical model, because the model has not seen a solution drainage from foam. As shown in previous work [5], the solution drainage from the foam increases actual foam void fraction. The solution drainage increases foam void fraction thus worsening the cooling conditions, the heat transfer coefficient goes down, the temperature difference between the wall and the foam increases. The assessment of the drainage process in numerical model could lead to smaller deviation with the experimental results.

6. CFX simulation of foam flow heat transfer in control rod channel (CPS)

For the simulation data it was assumed that the RBMK-1500 reactor emitted power is 4000 MW. The reactor core is composed of 2488 graphite columns, of which 211 are the control rods. On average, around 5% of heat is generated in graphite columns. In the calculations it was assumed that one graphite column then emits about 80 kW of heat. Control channel graphite column is 7 m long and 114 mm in diameter. The total channel wall area is 2.50572 m^2.

Fig. 7 shows the temperature profile in the channel obtained with CFX software package. At first, simulation has been done with the same foam void fraction and speed parameters as used in the experiment [3]. At the initial conditions foam inlet temperature was 20°C and 0.3 m/s speed. As can be seen from the picture, the average output foam temperature reaches 923.9°C, what means that the foam structure breaks up at this temperature. For low foam speed and high heat flux for channel, the average temperature of the foam in the entrance point of the channel immediately reached 42.73°C, while the initial conditions of 20°C been asked. The results suggest that the experimentally studied foam speeds and porosities are not suitable for use in the RBMK control rod channel cooling.

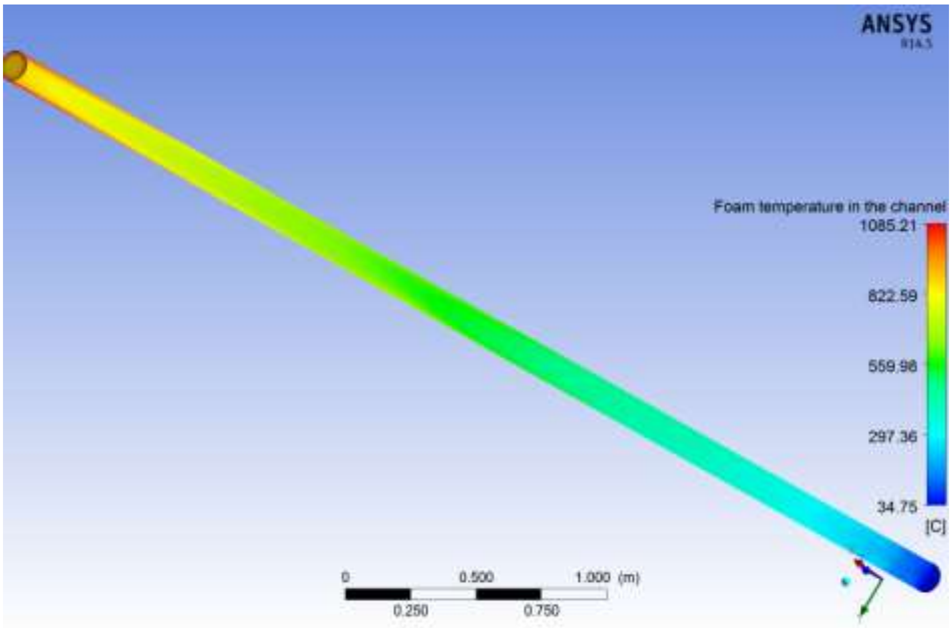


Fig. 7 Foam flow temperature profile of the RBMK control rod channel at 0.3m/s speed and 0.96 void fraction

Afterwards (Fig. 8) foam flow rate and inlet temperature was increased up to 1 m/s and up to 40°C. In the real-world conditions, control rod channel water inlet temperature is about 40°C and outlet temperature is around 70°C. Foam void fraction left the same - 0.96. The average channel entrance foam temperature obtained with software package is only slightly higher than the considered and was 40.75°C.

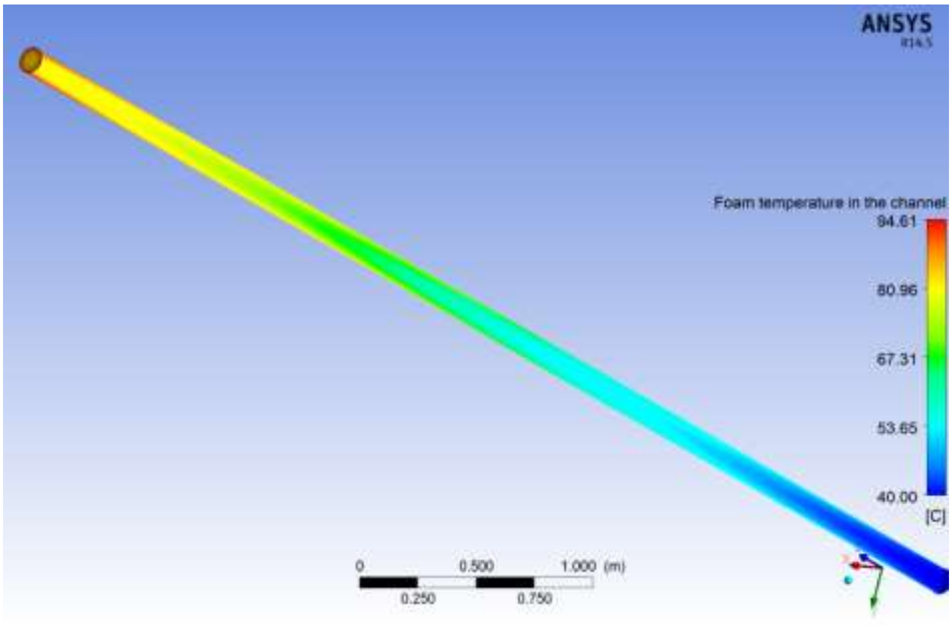


Fig. 8 Foam flow temperature profile of the RBMK control rod channel at 1 m/s speed and 0.96 void fraction

During experiment it was observed that 0.96 porosity foam has around 180 W/m^{2}K heat transfer coefficient. Water heat transfer coefficient varies from 500 up to 10 000 W/m^{2}K. In the current RBMK reactors construction, water flows downward because of gravity and reaches higher speed than that which was used (1m/s) with foam simulation. Due to higher water heat transfer coefficient and higher speed, the water film is capable to remove the excess heat. The purpose of this paper work was not to compare which cooling method is better, but to simulate if aqueous foam is capable to remove the excess heat from Control and Protection System channels.

As seen on Fig. 8, the output of the channel foam maximum temperature at the edges of the wall does not exceed 100°C, while the average total cross section reaches 87.38°C, suggesting that the foam structure remains non-degraded and such a control rod channel cooling is possible.

7. Conclusions

1. Using ANSYS CFX 14.5 software package it was composed numerical model of two-phase macro foam flow in a vertical cylindrical channel where inner channel wall transfers the heat to the upward foam flow. Simulation showed that foam temperature and heat transfer coefficient values corresponded well with the experimental results, it can be said that this is the right software package for modeling foam flow and heat transfer from cylindrical surface to the foam flow.
2. Foam application to the RBMK-1500 control rods channels cooling is possible, but it requires higher speeds or smaller void fraction foam than that was the subject of experimental studies.
3. During large void fraction foam numerical simulation, the best matching simulation results with experiment were obtained in the case when foam was described as Newtonian, Non-homogeneous, two-phase fluid flow.

References

1. **Almenas, K.; Kaliatka, A.; Ušpuras, E.** 1998. Ignalina RBMK-1500. A Source Book, Extended and Updated Version, Lithuanian Energy Institute, Kaunas, Lithuania.
2. **Warrier, G.R.; Dhir, V.K.; Momda, L.A.** 2002. Heat transfer and pressure drop in narrow rectangular channels, Experimental Thermal and Fluid Sciences 26: 53-64.
http://dx.doi.org/10.1016/S0894-1777(02)00107-3.
3. **Trepulis, M.** 2013. Macro foam convective heat transfer in vertical cylindrical tube, CYSENI, May 29-31, Kaunas, Lithuania ISSN 1822-7554.
4. **Leveque, R.J.** 1999. Finite Volume Methods for Hyperbolic Problems. Cambridge University, 2002. TORO, E.F. The Riemann Solvers and Numerical Methods for Fluid Dynamics.
5. **Gylys, M.** 2012. Inclined flat surface cooling by two-phase foam flow, Energetika 58(4): 219-230.

M. Trepulis, J. Gylys

RBMK - 1500 REAKTORIAUS VALDYMO STRYPO KANALO AUŠINIMO PUTŲ SRAUTU SUMODELIAVIMAS

R e z i u m ė

Tekančio putų srauto šilumos mainai priverstinės konvekcijos atveju, nors ir esant laminariniam tekėjimui, yra komplikuotas procesas, kurį sunku aprašyti analitiškai. Patikimiausias tokios problemos sprendimo būdas yra skaitmeninis modelis, patvirtintas eksperimentu. Eksperimentiškai tirtas vertikalaus cilindro formos vamzdžio vidinio paviršiaus šilumos atidavimas į viršų nukreiptam pereinamojo- turbulencinio režimo putų srautui. Skaitiniam modeliavimui naudotas ANSYS CFX programinis paketas. Eksperimentiškai nustatyta šilumos atidavimo koeficiento priklausomybė nuo putų srauto greičio bei dujingumo.

Šio darbo tikslas yra naudojantis gautu patvirtintu putų skaitmeniniu modeliu patikrinti galimybę RBMK-1500 branduolinio reaktoriaus valdymo strypų kanalus aušinti putomis. Nors visi tokio tipo reaktoriai (sumontuoti Ignalinos AE) jau uždaryti, tačiau yra dar 11veikiančių panašios konstrukcijos reaktorių.

M. Trepulis, J. Gylys

NUMERICAL SIMULATION OF FOAM FLOW THROUGH RBMK-1500 CONTROL AND PROTECTION CHANNEL

S u m m a r y

The foam flow phenomen on the forced convection, even with laminar flow, is complex and hard to develop analytically. Only key to the problem is experimental results and numerical solution. Heat transfer coefficient ‘h’ is very important parameter because it determines the rate of heat transfer. This paper uses two methods to determine this parameter, experiment and CFD simulation. Experiment was made for transient-turbulent convective heat transfer of macro foam in vertical cylindrical tube. CFD simulation in CFX was made and the experimental results have proven to be very useful for the validation of CFD calculations of foam flow through a heated cylindrical tube

After the numerical simulation proved the experimental results, RBMK-1500 control and protection channel cooling with macro foam was modeled. While all of the reactors (installed in Ignalina NPP) have been already closed, but are still 11 operational reactors with similar in design.

Keywords: foam flow, heat transfer coefficient, vertical tube, numerical simulation, experimental setup.

Received March 24, 2014
Accepted September 17, 2014