ANALYSIS OF PASSIVE HEAT REMOVAL SYSTEM TROUGH STEAM GENERATOR BY ANSOFT FLUENT

Taron Petrosyan

Doctoral Degree Programme (2), FEEC BUT

E-mail: taron.petrosyan@vut.cz

Abstract. This paper presents the simulation of the condensation of water vapor in one tube of the heat exchanger in the Passive Heat Removal System, which is a protective safety system of the Nuclear Power Plant. The simulation accounts for the turbulent flow of the gas along the tube with an assumption of constant wall temperature. The results from the CFD (computational fluid dynamics) simulations are compared with the experimental results from the literature for the condensation of water vapor and in general, agree well.

Keywords: PHRS, CFD, ANSYS, Fluent, thermal-hydraulics, NPP

1 INTRODUCTION

Comprehensive experimental and code development research activities have been conducted worldwide in the past decades to understand thermal-hydraulic phenomena and to establish code predictive capabilities for existing nuclear power reactors and related systems. Taking into account that one of the main reasons of Fukushima Daiichi NPP (Nuclear Power Plant) accident was the loss of the ultimate heat sink, the role of PHRS (passive heat removal system) becomes essential. Passive Heat Removal System is a protective safety system of Nuclear Power Plant based on the principle of a passive action, designed to provide long-term heat removal from the reactor core via secondary circuit. In the current study, the model of PHRS through SG (steam-generator) was developed for CFD analysis by ANSYS Fluent software [1]. A thorough description of the steam condensation process with applied appropriate parameters in the passive cooling system obtained as a result of the analysis. The rate of the steam condensation in heat exchanger pipeline was assessed and the results of those cases that corresponded with the input parameters in previously analyzed calculations by other software [2] ("ANSYS CFX"), are compared with each other. During work, there was done three main tasks: creating a geometrical form of one PHRS tube, an appropriate meshing of the model to achieve a better analysis of thermo-hydraulic phenomena and creating domains with physical properties for simulation of steam condensation in passive heat removal system's heat exchanger tube. Analysis through the "ANSYS Fluent" software, with numerous different cases, investigated. At the corresponding parameters, simulation showed more relevant results to experimental ones [3] in contrast to the first software package. As a result of calculation condensation rate assessed for the inlet pressure range of 1-8 MPa with appropriate temperature scale of 180-300 degree Celsius, which are fitting to the values during different types of failures and/or accidents in Nuclear Power Plant.

2 DATA COLLECTION FOR PHRS SG

One of the special means of controlling the beyond design basis accident, envisaged by the project "AES-2006" [4] is the Passive Heat Removal System, which is protective safety system of NPP based on the principle of passive action, designed to provide a long-term heat removal from the reactor core via secondary circuit (Fig. 1) [5]. It performs its functions in all abnormal modes and accidents that bring to passive heat removal from the reactor facility in order to maintain it in a safe

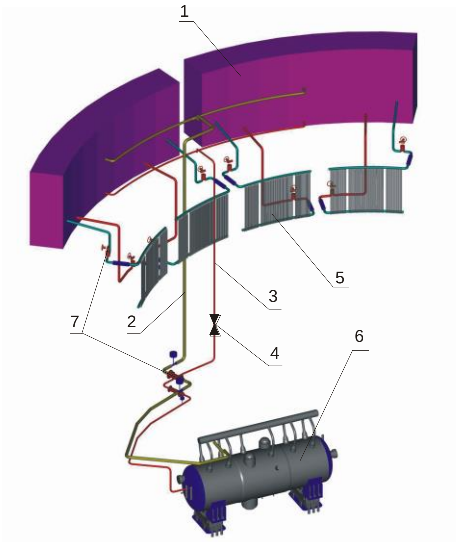

Figure 1: PHRS SG Components

1 - Emergency heat removal tanks
2 - Steam lines
3 - Condensate pipelines
4 - PHRS SG valves
5 - Heat exchangers of Containment PHRS
6 – Steam generator
7 - Cutoff valves

state, fc sis accident involving loss of all power sources. The system consists of four independent trains, one per each steam generator. Steam comes to the PHRS heat exchanger from a pipeline of each SG and condensed in the heat exchangers by heat removal water tanks. System components must withstand seismic impact loads, flooding. System channels are physically separated and totally independent from each other: process parts, control systems, supporting systems, locations of components, pipelines, cables, control elements are independent, so failure in one channel cannot bring to the failure in another one. This design eliminates dependent failures and common cause failures due to components locations as well as the impact on channel from any activities performed on another channel equipment (repair, maintenance). The design ensures automatic actuation of the system with passive principle (no need in power supply from external sources or operator interference) [3-5].

3 MODELING THE PHRS SG

The geometrical model was developed using ANSYS 16.0 Design Modeler User's Guide [6]. The detailed model was created as a composite of the following bodies: cylindrical heat exchanger tube with inner volume and surrounding heat exchanger tank. Heat exchanger tubes are submerged in a fluid tank with a water temperature of 20 °C. For simulation of heat transfer from tube to tank only part of the tank which surrounds the heat exchanger tube was modeled with size 32x200 mm, height 2100 mm. In accordance to design, it is supposed that tubes in two ends are curved approximately by 45 degrees to the side and bent by 90 degrees at the edges.

Parameters	Data
Distance between cold and hot collector	1816 mm
Length of heat exchange tube	2219 mm
Inner diameter of heat exchange tube	12 mm
Outer diameter of heat exchange tube	16 mm

Table 1: Dimensions of the PHRS SG heat exchanger tube

The geometrical model of the heat exchanger tube and surrounding heat exchanger tank is presented in Fig. 2. The partial differential equations that govern fluid flow and heat transfer are not usually amenable to analytical solutions, except for very simple cases. Therefore, in order to analyze fluid flows, flow domains are split into smaller subdomains (grids) that are called meshing. Meshing model was developed using ANSYS 16 Meshing User's Guide [7]. The grid size was refined iteratively until a reduction in grid size have no influence on the solution so the initial cell size of the finest grid was set to 0.5 mm. The final mesh appearance of PHRS heat exchanger tube's model is shown in Fig. 3.

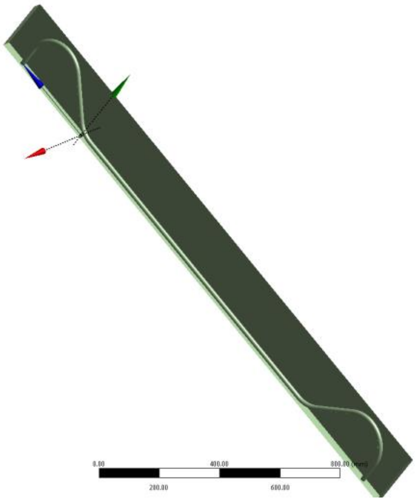

Figure 2: Geometrical model of PHRS SG

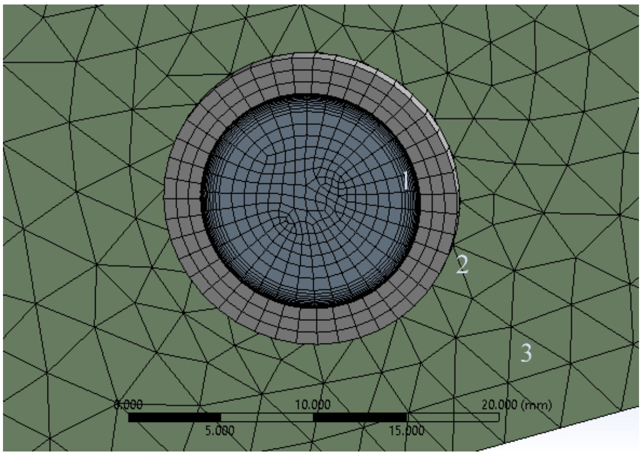

Figure 3: Meshing of PHRS

1 - Steam/fluid part
2 - Tube's wall
3 - Water tank

In a multiphase flow, the fluids are mixed on a macroscopic length scale, and separate velocity and temperature fields can be solved for each fluid. While flowing, phases interact with each other, as a result, heat and/or mass transfer occurs between the phases [8]. For the model mentioned above, three domains were created for simulation of condensation phenomena. First domain was created for simulation of steam/water part inside the heat exchanger tube. Material properties of the coolant were set according to the material library [9]. The turbulence model was set to the $k - \varepsilon$ model and for mass transfer model between the phases it is chosen Evaporation-Condensation model [8]. Second domain was created to model the wall of the heat exchanger tube. The material of the tube wall was selected as steel. Third domain was created to model the PHRS SG water tank. Material properties of the coolant were set according to the material library [9]. To assess possibility and rate of steam condensation in heat exchanger pipeline two variants with different inlet velocity were selected to analyze: 1 m/s and 3 m/s, which practically represent lower and upper possible operational boundaries of the system. Two different multiphase models are chosen for simulation. First one is the Eulerian model, which allows for the modeling of multiple separate, yet interacting phases [10]. An Eulerian treatment is used for each phase, in contrast to the Eulerian-Lagrangian treatment that is used for the discrete phase model. The ANSYS Fluent solution is based on the assumptions that a single pressure is shared by all phases, momentum and continuity equations are solved for each phase. Second model for multiphase flow modeling is Mixture model [10], which is a simplified multiphase model. It can be used to model homogeneous multiphase flows with very strong coupling and phases moving at the same velocity and lastly. The mixture model is a good substitute for the full Eulerian multiphase model in several cases. A simpler model like the Mixture model can perform as well as a full multiphase model while solving a smaller number of variables than the full multiphase model. Regarding time formulation, a steady state simulation is a much less time consuming than a transient simulation. However, multiphase flows often exhibit transient behavior and forcing a transient flow into a steady state might produce an unphysical solution. A transient simulation was, therefore, run in each code to investigate the transient behavior. The pressure-based coupled solver was used with gravity enabled. During the work, parameters were systematically changed in order to investigate a wide range of results accounting pressure range from 1 MPa up to 8 MPa. For most cases, inlet temperature set 300 degrees of Celsius, as it covers saturation temperature for all range of inlet pressures. However, several simulations also investigated with lower inlet temperature along with holding the condition to have it above saturation temperature, hence to observe condensation. Initial and boundary conditions, selected in accordance with PHRS SG system nominal operation conditions, are presented in Table 2.

Parameters	Boundary type	Option 1	Option 2
Velocity of steam (m/s)	Inlet	1	3
Mass fraction of steam at inlet		1	1
Temperature (°C)		180-300	180-300
Pressure (MPa)	Outlet	1-8	1-8
Temperature (°C)		170-285	170-285
Temperature of tube's outer wall (°C)	Wall	20	

Table 2: Initial and boundary conditions

4 CALCULATION RESULTS

As expected all calculations showed the same behavior but in different scales depending on initial conditions. Entering into the pipe, the temperature of steam starting to cool due to the low temperature of the wall and after reaching the saturation temperature, condensation occurs. For the case with the highest pressure (8 MPa) it takes place the earliest, starting after the second bending of tube (Fig. 4a) and vice versa for the case with lowest pressure (1 MPa), when condensation observed only after the long straight part of the tube (Fig. 4b). This difference is explained by the increase in the difference between the inlet temperature and saturation temperature at a particular pressure. The difference of the highest (300 °C) inlet temperature and saturation temperature for pressure range 1-8MPa is 121-5°C respectively. Thus, as saturation temperature is directly proportional to the pressure, for the case of 300 °C inlet temperature, the difference will be the smallest for 8 MPa pressure case (highest pressure), hence the earliest will be condensation. In other words, it takes the smallest time to reach saturation temperature for the high pressure case and the opposite for low pressure case.

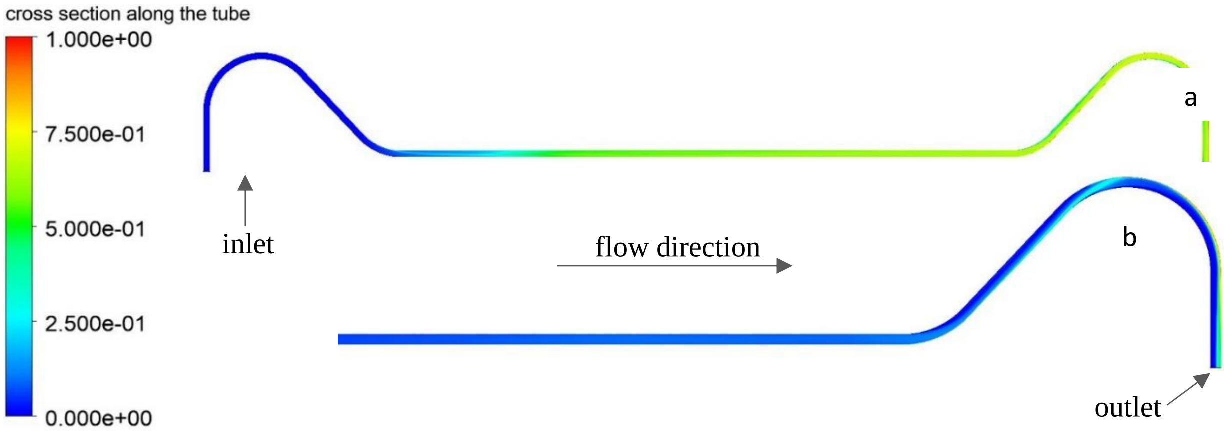

Figure 4: Liquid volume fraction a) for high pressure cases b) low pressure cases

According to figures (Fig. 5, 6) presenting the condensate mass flow rate at the outlet, the behavior of curve is the same for all of them, but located in different ranges. The amount of heat removed by one tube is calculated as a product of condensate liquid mass flow rate at the outlet and latent heat (heat of condensation). In contrast to Mixture model, Eulerian model showed relatively higher values for condensate mass flow rate. This could be explained by some simplifications and limitations of Mixture model, e.g. the Eulerian model is more flexible for phase's definitions. Compared to the case with inlet velocity 3 m/s, the mass flow rate of condensed liquid, hence heat removal capacity, is much higher for the 1 m/s inlet velocity case. Relatively small amount of condensate liquid in comparison with the first case is conditioned by the high speed of injected steam which passes through a tube in less than 1 second. Only the lowest and highest values of heat removal capacity (12-70 kW) are available from the experimental results for whole investigated pressure range, which in general agree well compared to the results of calculations. From figures also seen quite big differences of the result obtained by "ANSYS CFX" software for 7 MPa pressure case.

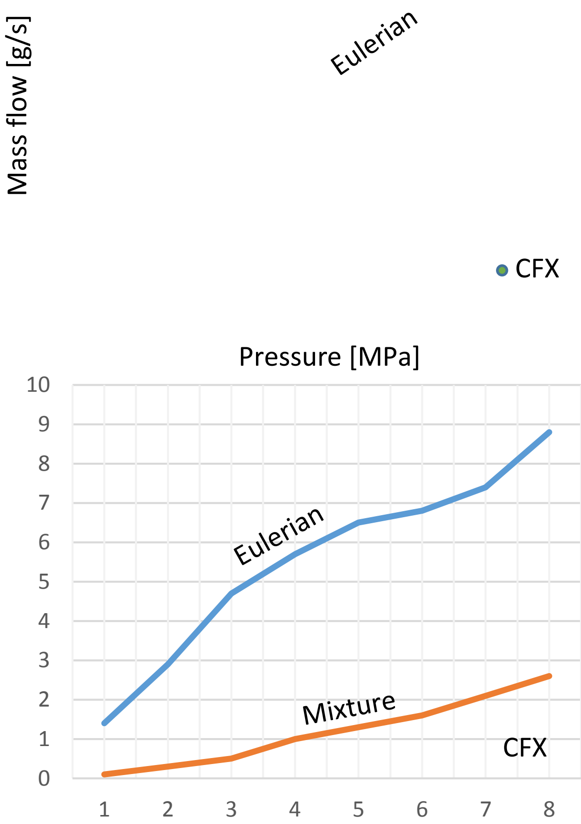

Figure 5: Condensate mass flow rate (1m/s)

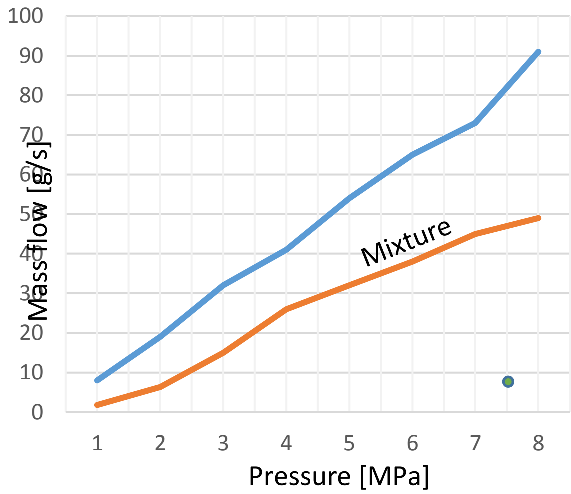

Figure 6: Condensate mass flow rate (3m/s)

5 CONCLUSION

The set of simulation results of condensation in PHRS heat exchanger tube showed more relevant results to experimental ones in contrast to the previous simulation by "ANSYS CFX" software, which indicates that ANSYS Fluent is more precise and capable for modeling condensation phenomenon. The assessment of two different velocity cases showed that in case of steam inlet velocity 1 m/s for pressures range of 1-8 MPa, total heat removal capacity will be equal to 12-136 kW for Eulerian model and 2-73 kW for Mixture model respectively. However, the results of the analysis for same pressure range with higher steam injection velocity (3 m/s) showed that condensation rate will decrease up to 2-13 kW for Eulerian model and 0.1-3.1 kW for Mixture model respectively.

REFERENCES

[1] ANSYS Fluent Reference Guide. ANSYS, Inc. Release 16.0, Southpointe, 2015

[2] Taron Petrosyan ; Karel Katovsky ; Tsolak Malakyan, Prceedings of 19th International Scientific Conference on Electric Power Engineering, 2018, 293-298

[3] V. Kukhtevich, Experimental study of thermal-hydraulic characteristics and stability of natural circulation, PhD dissertation, St. Petersburg Research and Design Institute “Atomenergoproekt”, St. Petersburg, Russia, 2010 (In Russian)

[4] Project “AES-2006”, Basic conceptual solutions on the example of leningrad NPP-2, "Atomenergoproekt", St. Petersburg, 2014, http://atomenergoprom.ru/u/file/npp_2006_eng.pdf

[5] V. Bezlepkin, S. Semashko, S. Alekseev, Investigation of the passive heat removal system through steam grenerator at the VVER-1200 reactor unit in the light of events at NPP “FUKUSIMA”, Prceedings of "Safety of NPPs with VVER", 28-31 May, St.Petersburg, Russia, 2013 (In Russian)

[6] ANSYS Design Modeler User’s Guide, ANSYS, Inc. Release 16.0, Southpointe, 2015

[7] ANSYS Meshing User’s Guide, ANSYS, Inc. Release 16.0, Southpointe, 2015

[8] ANSYS Fluent-Solver Theory Guide. ANSYS, Inc. Release 16.0, Southpointe, 2015

[9] The international association for the properties of water and steam, 2018, www.iapws.org

[10] ANSYS Fluent-Solver Modeling Guide, ANSYS, Inc. Release 16.0, Southpointe, 2015

[11] L. Landau, E. Lifshitz, Fluid Mechanics, Second Edition: Volume 6 (Course of TheoreticalPhysics S), 1987, ISBN-13: 978-0750627672

[12] Safety of nuclear power plants design, IAEA, Vienna, 2016, ISBN 978–92–0–109315–8