Comparative study of CFD and 3D thermal-hydraulic system codes in predicting natural circulation phenomena in a small-scale pool test facility

Audrius Graževičius[1,*], Anis Bousbia-Salah[2]

[1]Lithuanian Energy Institute, Breslaujos st. 3, LT-44403, Kaunas, Lithuania
[2]Bel V, Walcourtstraat 148, 1070 Brussels, Belgium

Abstract

Natural circulation phenomena have been nowadays largely revisited nowadays. The objectives were to investigate the possibilities to incorporate and rely on passive safety systems for heat removal under accidental conditions. For this purpose, assessment studies using CFD (Computational Fluid Dynamics) and also 3D thermal-hydraulic system codes are used. However, due to the lack of extended validation the predictions of such tools may not be accurate especially for natural circulation flow regimes where complex phenomena like flow mixing, temperature stratification, flow recirculation and instabilities, could take place.

In the present study, an experimental test case based on a small-scale pool test rig experiment performed by Korea Atomic Energy Research Institute [1], is considered for code-to-code and code-to-data comparison. The assessment is carried out using the ANSYS Fluent and the 3D thermal-hydraulic system CATHARE codes. The objective is to evaluate and compare their prediction capabilities with respect to the experimental measurement. It was observed that, notwithstanding the numerical and modelling differences in the used approaches, similar prediction results are obtained. Nevertheless, additional investigations efforts are still needed for a better representation of the considered phenomena.

Keywords: Natural convection; Thermal stratification; Experimental test; ANSYS Fluent; CFD; CATHARE; 3D

---

1. Introduction

Natural circulation phenomena have been increasingly revisited after the Fukushima-Daiichi accident. The objectives were to assess the reliabilities and capabilities of passive safety systems for heat removal under accidental conditions. Generally, systems consisting in large pools of water are used to provide heat sink for removing heat from the reactor or containment buildings. Such process provide reliable and efficient emergency cooling specially for design extension conditions including multiple failure accidents. Therefore detailed flow information, such as multi-dimensional local temperature and flow velocity, turbulence intensity, and turbulent energy, is required to evaluate the multi-dimensional flows and their associated heat transfers mechanisms. Assessment studies of such phenomena rely generally on CFD codes having the capabilities to describe the 3D microscopic phenomena occurring at small-scale geometries. However, recent studies emphasized the possibilities to use the capabilities of 3D thermal-hydraulic system codes for some specific cases [2]. Generally, acceptable results are obtained using these tools. However, due to reduced experimental database under natural circulation flow regimes, the predictions of such tools may not be accurate especially where complex phenomena like flow mixing, temperature stratification, flow recirculation and instabilities, could take place.

In the present study, the small-scale pool test rig experiment performed by KAERI [1], is considered. Indeed, several phenomena are that may occur under the passive condensate cooling tank operating are emphasized. These phenomena are challenging for the current computational tools, especially when local thermal mixing, turbulence and boiling are taking place. Therefore, in the current framework an assessment is carried out using the ANSYS Fluent and the 3D thermal-hydraulic system CATHARE codes. The objective is to evaluate and compare their prediction capabilities in predicting the main phenomena occurring under heating pool conditions. It was observed that, notwithstanding the numerical and modelling differences in the used approaches, similar prediction results are obtained.

* Corresponding author.
E-mail address: audrius.grazevicius@lei.lt

2. Description of experimental facility

The rectangular enclosure consists of the single heater rod, solid walls, and 2D particle image velocimetry measurement technique. The horizontal cylindrical heater rod inside the rectangular enclosure was designed to produce natural convection and thermal stratification phenomena. The working fluid in this facility is the de-ionised water. The length of the rectangular enclosure is 300 mm, width – 60 mm, and height – 650 mm. The heater rod, which is 19.05 mm diameter, is placed in a horizontal position, 85 mm above the bottom. The total length of the heater rod is 160 mm: 150 mm is heating part and 10 mm at the end of the heater rod is a non-heating part. The back wall is made from 15 mm thick Polycarbonate, the front and right walls are made from 3 mm thick Pyrex-glass, and for the bottom and left walls the 20 mm thick Stainless steel “304” is used. Five thermocouples (TF-01…TF-05) in the rectangular enclosure were installed to monitor and record thermal changes during the experiments. The TF-01…TF-05 thermocouples were placed in the centre of the rectangular enclosure, except the TF-02 thermocouple, which was placed 2 mm away from the heater rod surface [1]. A 3D view and drawing of rectangular enclosure construction is shown in Fig. 1.

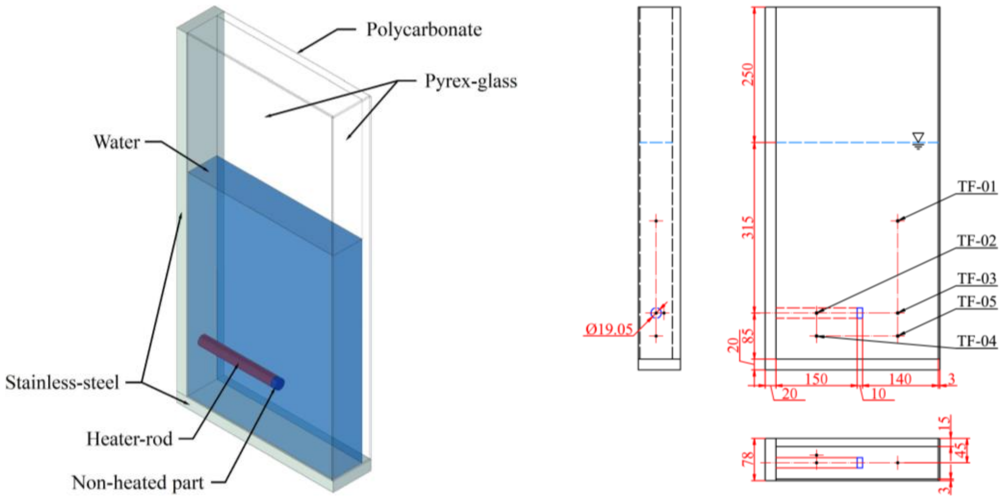

Fig. 1. A 3D view and drawing (dimensions in mm) of the rectangular enclosure.

3. Experimental conditions

The experiment takes five hours and can be divided into two stages. The first short stage is water cooling (0-600 s) and the second stage is water heating (600-14400 s). The initial ambient and rectangular enclosure walls temperature is approximately 14 °C. The rectangular enclosure is filled by water up to 400 mm level. The initial water temperature is approximately 32 °C. Within the first stage, due to differences in ambient, enclosure walls and water temperatures, the water temperature is decreasing from 32 °C down to approximately 31 °C. In the beginning of the second stage of experiment the heater rod with 600 W of thermal power is switched on. The insulation could not be used on the experimental facility walls due to the employment of the particle image velocimetry measurement technique, heat losses through the walls had to be evaluated. These losses were evaluated by comparing the amount of heat supplied to the water (given electrical power) and the real increase of the water enthalpy and the time spent to reach the saturation temperature. The estimated average heat loss during the time interval, when the water temperature increased from 20 °C to 90 °C, was approximately 310 W [1].

4. Computational tools

In order to compare CFD code and 3D thermal-hydraulic system code in modelling the natural convection and thermal stratification phenomena, the experimental facility of Korea Atomic Energy Research Institute and the ANSYS Fluent 17.2, and the CATHARE 2/V2.5_3/mod4.1 codes were chosen.

ANSYS Fluent is a CFD code and has powerful modelling capabilities: to model fluid flow, heat transfer, turbulence, reactions, airflow over an aircraft wing, combustion, bubble columns, oil platforms, blood flow, etc. This software is written in the C computer language, therefore, it can be automated using journal and scripting files and widely used in commercial and academic organisations. Modelling capabilities can be enhanced by incorporating user-defined functions. ANSYS Fluent offers highly scalable, high performance computing helps to solve complex and large CFD problems quickly and cost-effectively. The governing equations are solved using a finite volume method [3].

* Corresponding author.
E-mail address: audrius.grazevicius@lei.lt

CATHARE 2/V2.5_3/mod4.1 developed by CEA, EDF, Framatome, and IRSN is a thermal-hydraulic system code that solves the conservation laws for water and steam for a wide variety of single and two-phase flow conditions. The code has 3D vessel component capabilities aimed at representing large-scale 3D effects. The equations are space averaged over the 3D mesh and solved using the semi-implicit finite differences numerical scheme.

4.1. ANSYS Fluent model

The ANSYS Fluent 17.2 version [4,5] and high performance computing cluster SGI Altix ICE 8400 were used for numerical investigations. The high performance computing cluster consists of the 20 nodes, where each node has 48 GB of random access memory and 12 cores (3.33 GHz). A full-scale geometry of rectangular enclosure and hexahedral mesh were created using ICEM CFD, taking into account OECD/NEA Best Practice Guidelines [6,7], ECORA [8], ERCOFTAC [9] and ANSYS [10] recommendations. There are no symmetry planes or simplifications in the computational domain. Volume of Fluids multiphase model for two-phase flow simulations and time-depended explicit formulation for volume fraction discretization were chosen. We considered that natural convection is turbulent and $k - e$ Realizable turbulence model with Enhanced Wall Treatment was chosen. The discretization equations were solved using Pressure-Implicit with Splitting of Operators (PISO) segregated algorithm. The convergence criteria applied for energy 10^{-6} and 10^{-5} for continuity, k, omega, X, Z and Y velocities. The Lee model for water evaporation and boiling phenomena was used. The user-defined function has been developed by Lithuanian Energy Institute and incorporated in the CFD model in order to describe and regulate the thermal power rate of the heater rod. The mesh of computational domain is presented in Fig. 2.

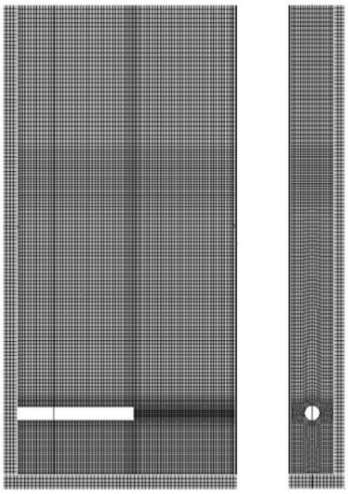

Fig. 2. The computational domain hexahedral mesh (313964 cells).

4.2. CATHARE model

The adopted CATHARE V2.5_3 nodalization for the current study is a full 3 D Cartesian model. It is mainly based upon the geometrical data provided in Fig. 1. The built-up nodalization (see Fig. 3) consists of an upper volume (VOLAIR) with a boundary condition (ATMOS1) representing the atmospheric conditions. It should be noticed that this volume do not simulates the upper part of the free surface of the pool. The pool represented by a THREED component SFP3D represents the volume occupied by the water. The latter (see Fig. 3) consists of:

1. 30 meshes of 10 mm length in the X direction. The mesh size was chosen to represent the 300 mm length of the pool.
2. In the Y direction, 3 identical meshes are considered to represent the 60 mm width of the pool. Each mesh is 20 mm long. This corresponds approximately to the heated rod diameter.
3. The axial meshing is important for the current test since the natural circulation flow is governed by the buoyancy forces, acting in the upward direction. The size of the axial meshes was chosen equal to the one in the X direction i.e. 10 mm.

In total, 3600 3D nodes are considered to represent the experimental facility. This number is considered sufficient to provide adequate results for the current simulation case [2].

* Corresponding author.
E-mail address: audrius.grazevicius@lei.lt

In CATHARE2 the 3D nodes are represented using the porous media approach. In this model, the governing equations are space averaged over the 3D mesh using the porosity P defined as the following:

$$P_v = \frac{V_{mesh} - V_{structures}}{V_{mesh}} = \frac{V_{fluid}}{V_{mesh}} \tag{1}$$

where: V_{mesh} represents the 3D mesh volume; $V_{structures}$ represents the volume occupied by the RPV structure inside the 3D mesh volume; V_{fluid} represents the volume occupied by fluid inside the 3D mesh volume.

In the current case, the porosity of all the 3D nodes is equal to the unity except the volume occupied by the heated rod. In this case, the porosity is evaluated as:

$$P_{orosity} = 1 - \frac{\pi \cdot D^2}{4 \cdot (2 \cdot \Delta Z \cdot DY)} = 1 - \frac{0.00113}{0.016} = 0.29 \tag{2}$$

where: D is the rod diameter.

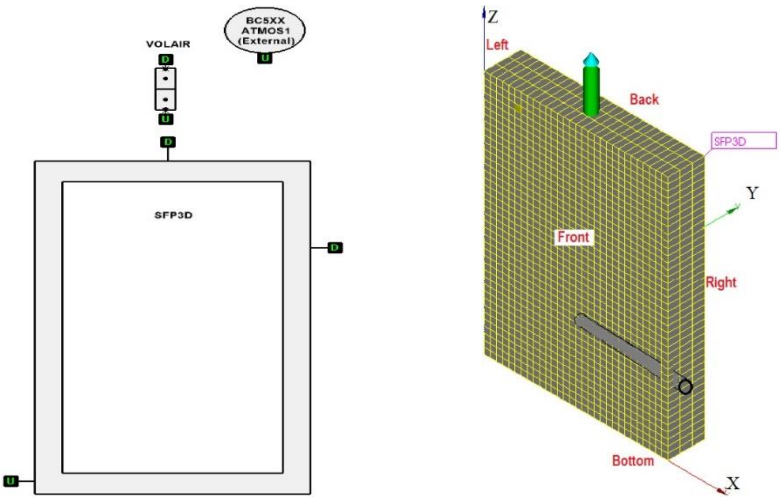

Fig. 3. CATHARE nodalization.

CATHARE 2 allows using the k-epsilon turbulence model in order to represent the physical diffusion in which turbulent phenomena are significant. However, this model was up to now not validated and in the current study, it could not be considered due to the presence of 3D nodes having a porosity less than the unity [11].

5. Results and discussion

The calculations results of selected key parameters are compared with the measurement data provided in reference [1]. Qualitative and quantitative comparison are outlined hereafter.

Fig. 4 shows the simulated temperatures of the thermocouples vs. the measured ones. In this set of curves one can observe that the temperatures above or at the heated rod level (TF-01…TF-03 see Fig. 1) are well simulated by both computational tools during the transient. Actually, better results are obtained by ANSYS Fluent code in predicting the equilibrium temperature. Indeed, CATHARE under predicts the equilibrium temperature by 4 °C. The origin of this discrepancy is supposed to be related to the heat losses mechanism at the free surface. In CATHARE, this heat transfer coefficient remains constant on during the whole test time since the volume above the free surface is not simulated. This is not the case in the ANSYS Fluent model.

On the other hand, larger discrepancy is observed for the temperatures of the thermocouples located beneath the heated rod level (TF-04 and TF-05). This deviation is mainly due to the codes ability in predicting the stratification phenomenon at the bottom of the pool. Indeed, as shown in Fig. 4, both ANSYS Fluent and CATHARE codes seem to predict narrower stratification zone with respect to the experimental one. In fact, better matching with the experimental trend is obtained by both codes when temperatures of lower axial level are considered.

On the other hand, one can notice that the break-up of the bottom stratification, which occurs at around 9000 s, is better predicted by ANSYS Fluent code with respect to the CATHARE results. As shown in Table 1, the break-up occurs earlier in CATHARE prediction around 5000 s while it is around 7000 s for ANSYS Fluent.

* Corresponding author.
E-mail address: audrius.grazevicius@lei.lt

[Fig. 5](#) shows the mean axial velocity distribution at an axial level 150 mm and the x-velocity distribution along the x-direction at 150 mm. As could be observed, qualitatively good agreement with the experimental measurements is obtained. This means that the main phenomena (pool recirculation zones and stratification) are predicted by the codes. However, from the quantitative point of view, ANSYS Fluent code has a tendency to over predict the velocities while CATHARE code under predicts the velocities values. Such discrepancies could be related to the absence in the CATHARE model heat and mass transfer at the upper pool free surface and also to the absence in the CATHARE calculations of the turbulence model.

This might be due to the occurrence of subcooled boiling at the heater rod at that temperature. This break-up phenomenon is well predicted by the CATHARE code even though it occurs earlier in time. On the other hand, in the upper part of the pool (above the heater rod level) a larger central recirculation is predicted as the measured one. Furthermore, local recirculation zone in the top left side of the pool is not predicted by the code for pool temperatures below 90 °C. Beyond this temperature, a local recirculation zone at the top left side appears in the CATHARE calculation results.

Table 1. Phenomena occurring during the test.

Parameter	Experiment	ANSYS Fluent	CATHARE
Presence of bottom stratification	Yes	Yes	Yes
Recirculation zone(s)	Yes	Yes	Yes
Stratification break-up	9000 s	7000 s	5000 s
Number of nodes	-	313964	3600
Turbulence model	-	Yes	No
Calculation time CPU	-	504 hr	52.7 hr

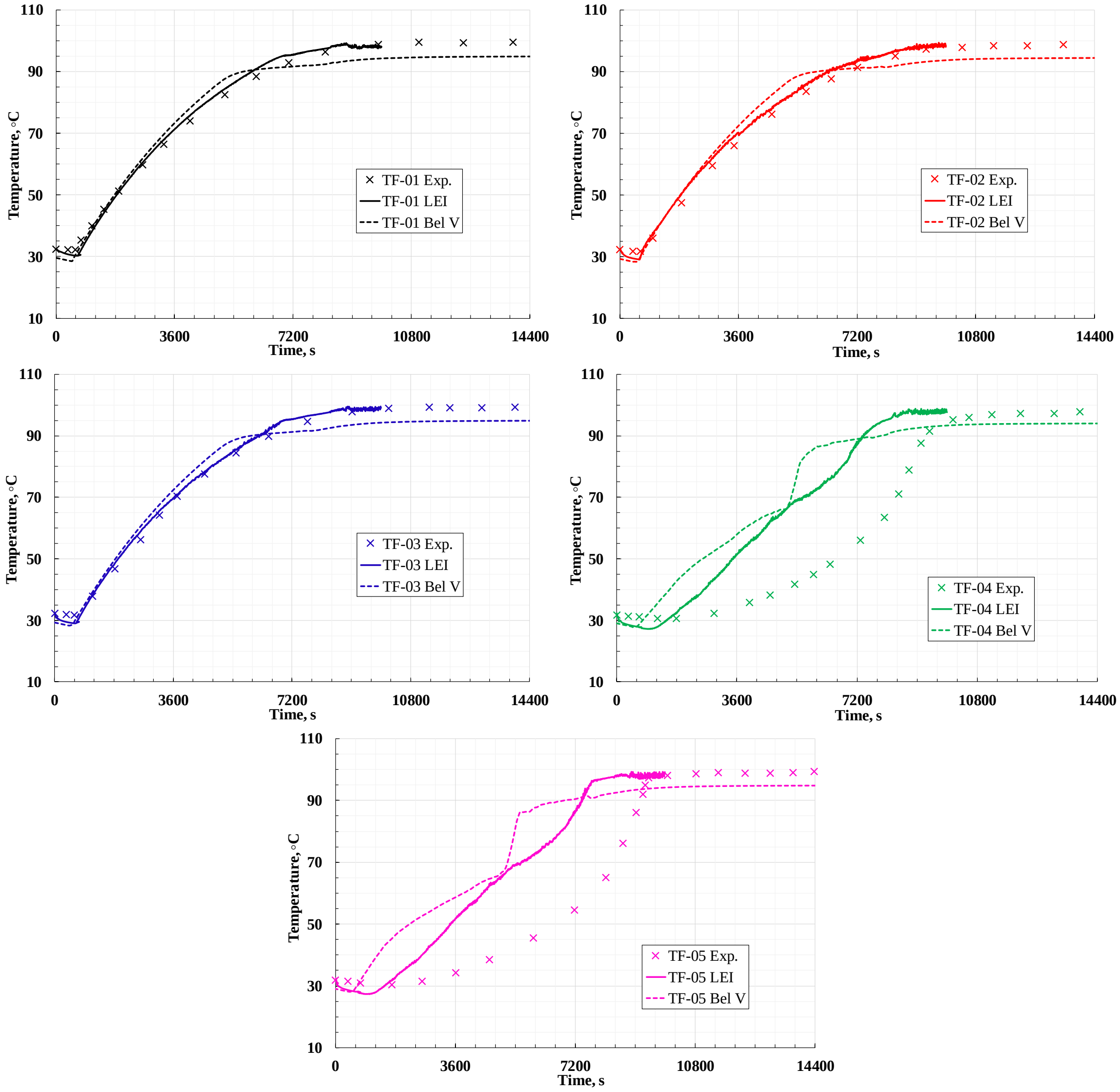

Fig. 4. Comparison of TF-01…TF-05 water temperatures.

* Corresponding author.
E-mail address: audrius.grazevicius@lei.lt

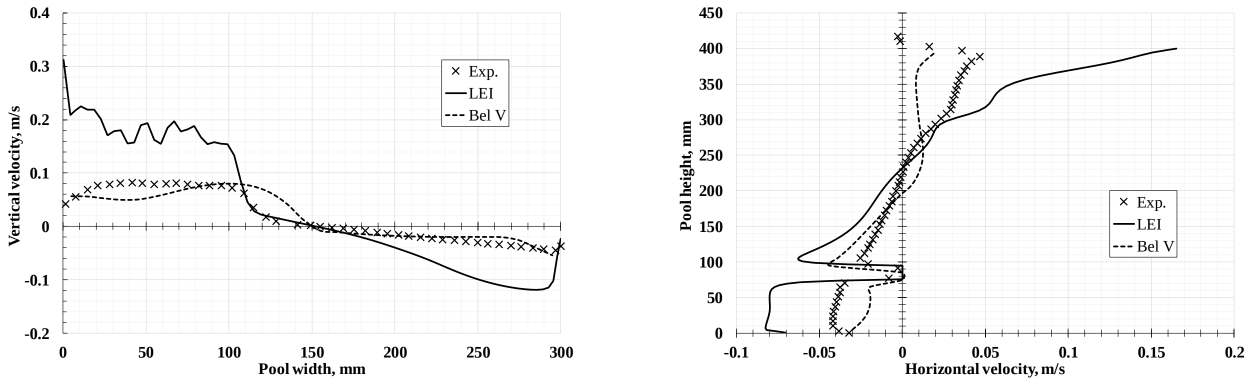

Fig. 5. Comparison of vertical and horizontal velocity at equilibrium temperature.

6. Summary and conclusions

In the current study, ANSYS Fluent CFD and CATHARE2 3D thermal-hydraulic system codes features are assessed against the KAERI pool test rig experimental test. The objective was to evaluate and compare different numerical approaches in predicting the natural circulation phenomenon under heating pool conditions. In general, it was observed that the main phenomena occurring during the experiment are well reproduced by both codes. On the other hand, similar temperature and velocity profiles are predicted notwithstanding the fact that some discrepancies are observed. The main deviations between the two codes could be explained by the absence of a model that allow the 3D system code to simulate the heat and mass transfer at free surface as well as the non consideration of the turbulence model.

Finally, the outcomes of the current study confirm the capabilities of CFD codes in simulating phenomena occurring under natural circulation flow regimes in pool with free surface configuration. It should be also mentioned that acceptable results are obtained using 3D porous media thermal-hydraulic system code.

References

[1] S. Kim, D. E. Kim, S. U. Ryu, S. T. Lee, D. J. Euh, Experimental investigation on the natural convection flow in pool boiling, Nuclear Engineering and Design 280 (2014) 349–361.

[2] A. Bousbia Salah, J. Vlassenbroeck, Assessment of the CATHARE 3D capabilities in predicting the temperature mixing under asymmetric buoyant driven flow conditions, Nuclear Engineering and Design 265 (2013) 469–483.

[3] ANSYS. ANSYS Fluent. [viewed 1 August 2020]. Accessed via internet: https://www.ansys.com/Products/Fluids/ANSYS-Fluent

[4] ANSYS. ANSYS Fluent User's Guide, 17.2 Release. August 2016. [viewed 1 August 2020]. Accessed via ANSYS Customer Portal.

[5] ANSYS. ANSYS Fluent Theory Guide, 17.2 Release. August 2016, [viewed 1 August 2020]. Accessed via ANSYS Customer Portal.

[6] J. Mahaffy et al., Best Practice Guidelines for the use of CFD in Nuclear Reactor Safety Applications, NEA/CSNI/R(2007)5.

[7] J. Mahaffy et al., Best Practice Guidelines for the use of CFD in Nuclear Reactor Safety Applications – Revision, NEA/CSNI/R(2014)11.

[8] F. Menter et al., CFD Best Practice Guidelines for CFD Code Validation for Reactor - Safety Applications, EU project Evaluation of Computational Fluid Dynamic Methods for Reactor Safety Analysis (ECORA) under Contract No: FIKS-CT-2001-00154.

[9] M. Casey, T. Wintergerste, ERCOFTAC Special Interest Group on Quality and Trust in Industrial CFD, Fluid Dynamics Laboratory, Sulzer Innotec, Version 1.0, January 2000.

[10] ANSYS, 12 April 2016. Module 09: Best Practice Guidelines. Introduction to ANSYS Fluent, 17.0 Release.

[11] J. Darona, CATHARE 2 v25_3mod8.1 code: General description, DEN/DANS/DM2S/STMF/LMES/ NT / 2018-63808/A.

* Corresponding author.
E-mail address: audrius.grazevicius@lei.lt