# A COMPUTATIONAL FLUID DYNAMICS VALIDATION STUDY OF STEAM CONDENSATION ON THE NUCLEAR REACTOR CONTAINMENT WALLS

B. Gera[1], Pavan K. Sharma[1], R. K. Singh[1], K. K. Vaze[1]
[1]Reactor Safety Division, Bhabha Atomic Research Centre, Mumbai, INDIA-400085
E-mail of corresponding author: bgera@barc.gov.in

## ABSTRACT

In water power cooled reactors, significant quantities of hydrogen could be produced following a severe accident (loss-of-coolant-accident along with non availability of Emergency Core Cooling System). A sound understanding of dispersion, stratification and diffusion of released hydrogen during severe accidents is, therefore, of practical importance and use to better understand the possibility of ignition, combustion and explosion of such releases within the context of containment safety. The presence of air and steam in the containment atmosphere also affects the hydrogen distribution as steam condensation takes place at containment walls in presence of non condensable and bulk of the mixture diffuses towards wall. The use of CFD codes for the analysis of the hydrogen behaviour within NPP containments during severe accidents has been increasing during last years. The commercial CFD codes do generally not have built-in steam condensations models. In the present work, the adaptation of a commercial multi-purpose code to this kind of problem is explained, i.e. by the implementation of models for several transport and physical phenomena like steam condensation onto walls in presence of non-condensable gases. Steam condensation was modeled using the Uchida correlation, which was originally developed to be used for "integral" (volume-averaged) modeling of steam condensation in the presence of non-condensable gases. The Uchida correlation is based on experiments on natural convection from relatively small vertical plates. The code has been validated against experimental data from the TOSQAN and COPAIN experimental facilities.

## INTRODUCTION

Steam condensation in the presence of non-condensable gases is a relevant phenomenon in many industrial applications, including nuclear reactors. Condensation on the containment structures during an accident is one of the thermal hydraulic phenomena that characterize the operation of passive emergency systems in the nuclear reactors of new generation. Rate of steam condensation at containment walls also affects the transient pressure in the inner containment after loss of coolant accident. Apart from this during a severe accident in a water cooled power reactor nuclear power plant (NPP), large amounts of hydrogen would presumably be generated due to core degradation and released into the containment. The integrity of the containment could be threatened due to hydrogen combustion. If composition of the hydrogen–steam–air mixture lay within a certain limits, the combustion would occur. The steam condensation phenomenon is important from hydrogen distribution point of view to locate the flammable region in the containment. The prediction of hydrogen behavior at severe accident conditions may help in devising adequate accident management procedures [1]. The main issue is to predict whether, due to atmosphere mixing and stratification, the local hydrogen concentration in certain parts of the containment is likely to exceed flammability limits.

Currently use of CFD techniques to model such scenario is popular, since CFD codes provide more detailed information in such scenario more correctly. These commercial available CFD codes do generally not have built-in steam condensations models. Consequently, it is necessary to implement steam condensation via user-defined subroutines. Two main approaches have been proposed by various authors to model wall condensation in CFD codes. In the first approach (two-phase flow approach) separate momentum equation are solved for vapour and liquid phase. A fine mesh is required and liquid film and diffusion of steam towards wall through boundary layer formed by non-condensable is modeled. This approach requires very large computational time and will probably take some time to be used for any practical applications in future. In the second approach a single fluid model is used where steam is modeled as a separate species via species conservation equation. For modeling steam condensation, mass sink and corresponding energy sink are modeled in the very first cell near to condensing wall. In this second approach there are two ways to calculate these sink terms. The one is based on diffusion theory which requires a very fine mesh near the wall and computes steam condensation rate that diffuses towards wall through species boundary layer. Houkema et al. [2] have used this approach without the use of engineering correlations for steam condensation on walls. Another way that is very popular currently is to include heat or mass transfer

correlations that were originally developed for "lumped" (volume-averaged) calculations and apply them in the layer of cells contiguous to the condensation surface. These correlations use the temperature and steam mass fraction value from bulk flow and calculate the condensing heat transfer coefficient, since the corresponding sink term is applied very first fluid cell near the condensing wall the bulk flow parameters and properties are evaluated at these cell centers. Thus in this approach rate of heat and mass transfer depends on the cell width near the condensing wall. CFD codes were developed to solve equations that are derived from first principles, using local instantaneous description, so that the inclusions of correlations, which are based on averaged physical quantities and provide average condensation rates, is somehow contrary to the basic "philosophy" of CFD. However, this optimized approach allows relatively fast calculations and may prove adequate for large industrial applications.

Babić et al. [3], Kljenak et al. [4] and Kljenak et al. [5] have used this approach with CFD code CFX to simulate experiments on containment atmosphere behavior at accident conditions, which were performed in the TOSQAN and ThAI experimental facilities. Steam condensation was modeled as a sink of mass and enthalpy by applying the correlation by Uchida et al. [6]. Valdepenas et al. [7] have also used second approach for modeling steam condensation on walls where condensation rate was based on correlation developed by Terasaka and Makita [8]. Mimouni et al. [9] have modeled steam condensation by a two phase flow approach that takes into account the physical phenomena of importance like effect of liquid film thickness. In the CFD code TONUS developed for hydrogen risk analysis [10] steam condensation was implemented as steam diffusion through a mass boundary layer based on heat and mass transfer analogy (Chilton-Colburn type). The present paper is about validation of the wall condensation model based on Uchida correlation in CFD-ACE+ [11]. The implemented condensation model has been described and results of validation calculations are compared to experimental data. A brief overview has been given on the comparison of results obtained with CFD-ACE+ versus experimental results from TOSQAN and COPAIN test facility. The simulation was performed with the knowledge of experimental results. In the considered TOSQAN experiment, which was also proposed for the OECD/NEA International Standard Problem No. 47, air was initially present in the vessel, and steam, air and helium were injected during different phases of the experiment at various mass flow rates. Intermediate steady states were obtained when the steam condensation rate became equal to the steam injection rate, with all boundary conditions remaining constant. Two such steady states were simulated in the present work.

## MATHEMATICAL MODEL

The general-purpose CFD code CFD-ACE+ solves the local instantaneous transport equations for mass, momentum, energy, species and turbulence quantities. The general transport equation for variable $\phi$ is

$$\frac{\partial(\rho\phi)}{\partial t}+\frac{\partial(\rho u_i\phi)}{\partial x_i}=\frac{\partial}{\partial x_i}(\Gamma_\phi\frac{\partial\phi}{\partial x_i})+S_\phi \tag{1}$$

Where $\phi$ is appropriate variable (mass, momentum, energy, species and turbulent quantities), $\Gamma_\phi$ is diffusion coefficient and $S_\phi$ is the sink or source term. To model the mixture behaviour of air and steam, a continuum approach was used, where only one velocity field was defined using the average density of gas mixture. The independent behaviour of steam was considered using species transport equation. The discretisation of the equations in the CFD-ACE+ code is based on a conservative finite-volume method. A non-staggered grid arrangement is employed, where all the variables (velocity components and scalars) are stored in the geometrical centers of control volumes (cells) that fill up the considered flow domain. In the present work, the second approach was used. Steam condensation was modeled as a sink of mass and enthalpy by applying the Uchida correlation based on experiments on natural convection and was basically developed for lumped approach. The condensate film on structures was not considered. Basically, the steam condensation rate $m^0$ was obtained from the expression.

$$m^0=C_u(\frac{\rho_{steam}}{\rho_{nc}})^{0.8}\frac{A(T-T_{wall})}{h_{fg}} \tag{2}$$

Where $C_u$ is adjustable coefficient in Uchida correlation (W/m$^2$k), $\rho_{steam}$ is density of steam (kg/m$^3$), $\rho_{nc}$ is density of non-condensable gases (kg/m$^3$), $A$ is area of condensation surface (m$^2$), $T$ is temperature (K), $T_{wall}$ is average temperature of condensation surface (K) and $h_{fg}$ is latent heat of steam (J/kg). In this correlation value of

all the physical variables was required from bulk flow except $T_{wall}$ which was the condensing wall temperature. This correlation will always predict the condensation as long as bulk temperature is more than wall temperatures even the steam partial pressure is lower than the saturation pressure at the condensing wall temperature. The steam partial pressure must be higher than the saturation pressure at the temperature of the wall for condensation to occur, this was made ensure by applying the appropriate condition in user defined subroutine. Thus, during the simulation the amount of steam condensation was calculated only if the steam partial pressure was above the saturation pressure. The corresponding enthalpy sink $H^0$ (sink term in energy flow equation) due to condensation was calculated as:

$$H^0 = m^0 (C_{p,steam} T_{cell} - C_{p,air} T_{ref}) \tag{3}$$

The specific heat of the air $C_{p,air}$ and reference temperature $T_{ref}$ was used for calculation of reference enthalpy in the CFD-ACE+ code. A user defined subroutine was added for modeling steam condensation as sinks of mass and enthalpy occurred in cells contiguous to the condensing wall in the CFD-ACE+ computational tool. The mass sink was calculated from Eq. (2) for each cell where the "bulk flow" physical quantities (temperature, steam density and non-condensable gas density) were evaluated at the cell centre. As the temperature of the gaseous mixture corresponding to the cell centre appears in Eq. (2) and (3), the calculated condensation rate and enthalpy sink necessarily depend on the width of cells contiguous to the condensation surface. The value of the coefficient $C_u$ was taken from the previous research done with the same approach by various authors. The saturation pressure at condensing wall temperature was calculated and it was ensured that steam partial pressure must be higher than the saturation pressure at the temperature of the wall for condensation to occur. It was assumed that the condensing wall immediately removes the condensation heat. The air steam mixture was considered compressible and density was modeled as ideal gas. Standard k-ε model with standard wall function was used for modeling turbulence in all the simulation.

## COPAIN FACILITY

The COPAIN facility Cheng et al. [12] was a simple facility designed to study the phenomenon of wall condensation in the presence of non-condensable gases. The facility was consisted a vertically placed rectangular channel of cross section 0.6 m X 0.5 m. The vertical height is around 2.5 m where one vertical side was served as a condensation plate upto 2 m height. The experiments were performed with pressure between 1 and 7 bar, temperature under 165 $^0$C, heat flux exchanged with the plate between 1 and 30 kW/m$^2$, maximum velocity of 3 m/s, with different fraction of air, steam and helium. Inside the channel forced or natural convection was observed depending on prevailing conditions. The database selected to validate the wall condensation model is shown in Table 1. The full computational domain was simulated. A fine mesh was done with 93750 cells so that average cell size near the wall is 0.02 m as suggested by different authors [3, 4]. The grids were uniform, including the nearest cells to the wall. In this simulation, the condensation acts as a sink of mass and energy. Thus, the liquid film and the influence of the non-condensable gas layer were reduced to a simple sink term. Besides, coefficient $C_u$ was adjusted to obtain a good agreement between measured and calculated condensation mass flow rate in the COPAIN experiment. The same values of coefficients were used for all four steady states simulated in this work. The Fig. 1 depicts the steam mass fraction on the condensation wall for the test P0444. This single-phase approach gives results which are acceptable but the condensation flux depends on the grid refinement and on the adjusted coefficients. Condensation rate is slightly over estimated in the case of free convective heat transfer. Fig. 2-5 show the condensation flux at wall along the height in the condensing section for all the steady states.

Table 1: Parameters of the COPAIN tests

| Test No. | Convective Heat Transfer | Air Velocity at Inlet (m/s) | Pressure (bar) | Air Temperature at Inlet (K) | Wall Temperature (K) | Mass Fraction of non-condensable |
|---|---|---|---|---|---|---|
| P0441 | Forced | 3 | 1.02 | 353.23 | 307.4 | 0.767 |
| P0443 | Free | 1 | 1.02 | 352.33 | 300.06 | 0.772 |
| P0444 | Natural | 0.5 | 1.02 | 351.53 | 299.7 | 0.773 |
| P0344 | Natural | 0.3 | 1.21 | 344.03 | 322 | 0.864 |

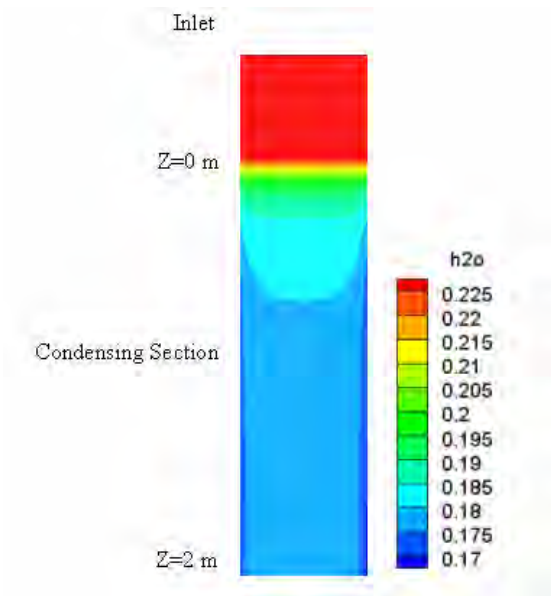


Fig. 1: Steam mass fraction on the condensing wall for test P0444

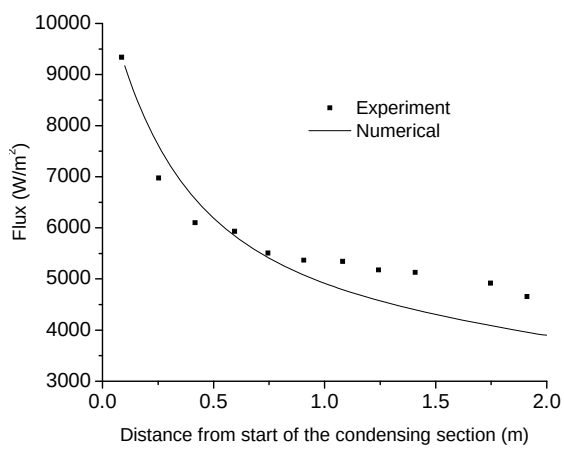


Fig. 2: Variation of condensation flux with height–test P0441

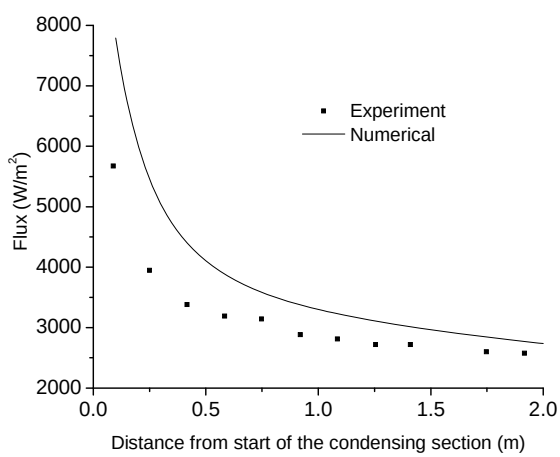


Fig. 3: Variation of condensation flux with height–test P0443

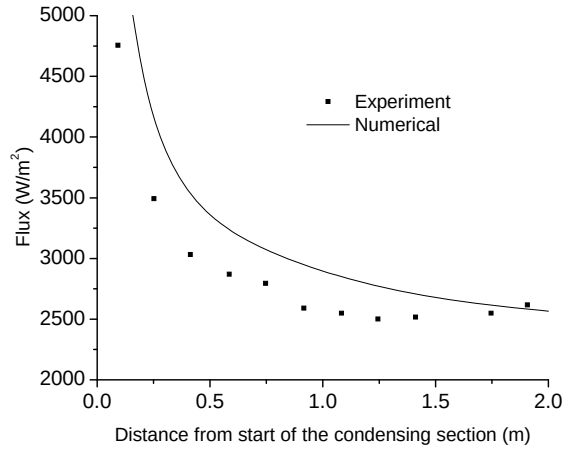


Fig. 4: Variation of condensation flux with height–test P0444

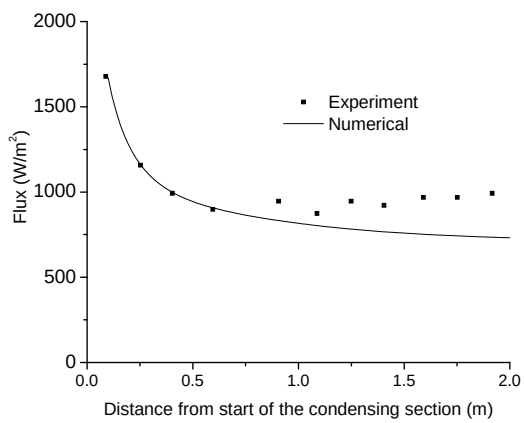


Fig. 5: Variation of condensation flux with height–test P0344

## TOSQAN FACILITY

The TOSQAN facility was made of a cylindrical vessel with an internal volume of 7.0 $m^3$. The vessel total height was 4.80 m, and the diameter of the main cylindrical part was 1.50 m. A vertical tube was located along the cylinder centre-line to facilitate injection of steam, air and helium. The fluids were injected though opening of injection tube located at the elevation of 2.10 m (all elevations refer to the sump floor). The vessel wall was divided in condensing and non-condensing region by means of appropriate temperature control of the vessel walls. During the experiment steam condensation was occurred only on walls with lower temperature, located between elevations 2.39 and 4.39 m [13, 14]. The vessel was initially filled with air to simulate the experiments. Air, steam and helium (which were used instead of hydrogen for safety reasons) were injected intermittently with various mass flow rates. The sump, where the liquid from the condensate film collected, was drained continuously. The thermal-hydraulic behavior was determined by the following dominant physical phenomena: gas injection, steam condensation, heat transfer and buoyant flow. The experiment was conducted for about 18,000 s. During the experiments certain steady states were achieved when the steam condensation rate became equal to the steam injection rate, while all boundary conditions were kept constant. The data during these steady states were recorded for validation of computer codes. The injected steam first flew upwards in the vessel center-line region then flow along the top wall and reaches to condensing section. As steam comes in contact of cold wall, condensation takes place and steam concentration at the wall decreases. This causes the movement of seam from the bulk of the mixture towards condensing wall. A local recirculation of steam takes place at the condensing wall. Two steady states were used to validate the present approach for condensation modeling. The parameters of the steady states are provided in Table 2. The temperatures

of the upper and lower non-condensing wall were 122.0 $^0$C and 123.5 $^0$C, respectively. The temperature of the condensing wall was 101.8 $^0$C during steady states 1 and 107.8 $^0$C during steady state 2.

Table 2: Parameters of the steady state during experiments in TOSQAN facility

| Parameter | Steady State 1 | Steady State 2 |
|---|---|---|
| Pressure, bar | 2.4 | 3.01 |
| Average Temperature, $^0$C | 113.7 | 119.7 |
| Air Mass, kg | 8.18 | 8.18 |
| Initial Steam Mass, kg | 4.37 | 6.60 |

A two-dimensional axisymmetric model (in cylindrical coordinates) was developed as shown in Fig. 6. The computational domain was divided with 480 cells in the axial (vertical) direction and 75 cells in the radial (horizontal) direction. Fig. 7-8 show the measured and calculated radial profiles of steam volume fraction during the two steady states at different vertical positions. The condensation takes place at the wall which is manifested by sharp decrease in steam mole fraction near wall. Although the computational domain was only one half of a vertical plane, symmetric results are shown over the entire vessel to facilitate the comparison. The simulation predicted a homogeneous atmosphere as resulted by the steam mole fraction profile in Fig. 7. For steady state 2, where the atmosphere was less homogeneous due to the higher steam injection rate, a good agreement was observed at the highest and lowest elevation. For the intermediate elevation, the code replicated the experimental pattern but with a shift towards higher values. The focus of the present work was to validate the condensation modeling approach, hence other phenomenon like mixing and stratification were not discussed in detail. In general, the calculated values are within the ranges of measured values. Discrepancies were observed for steam mole fraction during both the steady state. Steady state 2 is the less suitable for applying Uchida's correlation due to the high steam injection rate which induces forced convection and Uchida correlation is valid for natural convection. Nevertheless, general agreement between experimental and simulated results indicates that the proposed approach for modeling steam condensation may be considered as adequate. To illustrate the simulated patterns of steam distribution during second steady state, Fig. 9 show the fields of steam mass fraction. One of the problems of the present methodology is the use of first cell centre values for bulk flow parameters by the CFD code. If values from the cells contiguous to the condensation surface are applied, then the results are affected by cell size. Near the condensation surfaces, large variations in steam concentration are expected which are not reproduced by the proposed method.

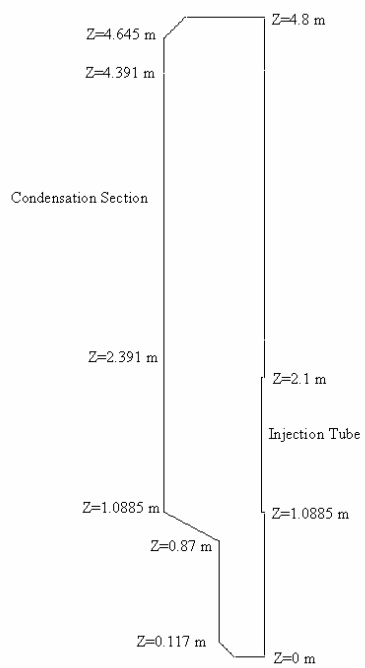


Fig. 6: Geometry of TOSQAN facility used for simulation

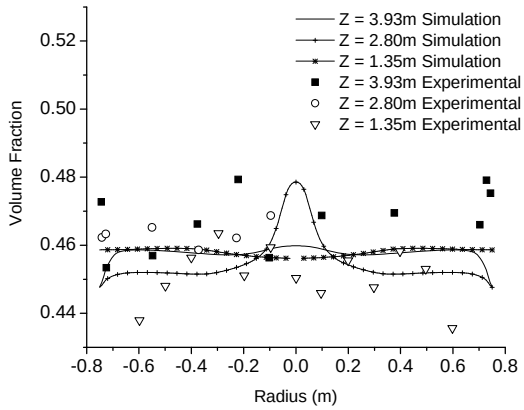


Fig. 7: Experimental and simulated steam mole fraction for first steady state in TOSQAN experiments

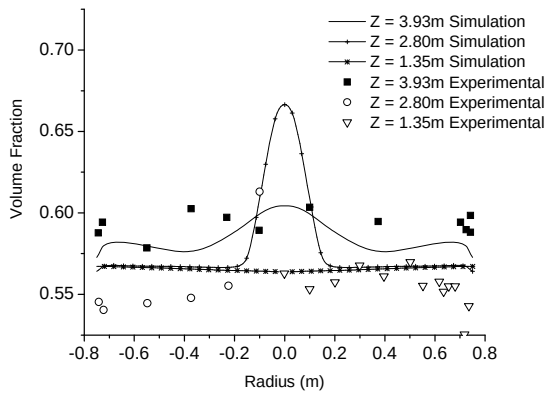


Fig. 8: Experimental and simulated steam mole fraction for second steady state in TOSQAN experiments

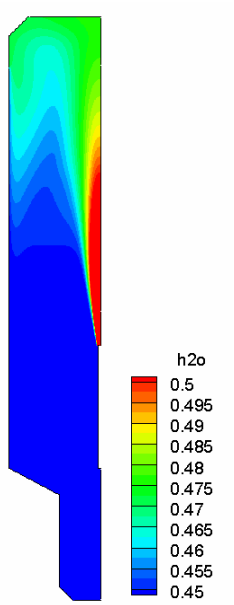


Fig. 9: Steam mass fraction for second steady state in TOSQAN experiments

## CONCLUSION

A model for steam condensation was implemented in the Computational Fluid Dynamics code CFD-ACE+ based on correlation developed for volume averaged approach. The CFD code CFD-ACE+ was used to simulate experiments that were performed in the TOSQAN and COPAIN facility. There was a reasonable agreement between simulated and experimental results which suggest that this simplified approach can be considered as adequate. This approach does not solve the phenomenon from the first principle, but this approach can be considered as effective for industrial problem where solution for large computational domain is required. The present work suggests that the approach could be used for hydrogen distribution calculation with steam condensation in nuclear reactor containment required for safety analysis.

## REFERENCES

[1] Royl, P., Rochholz, H., Breitung, W., Travis, J.R., Necker, G., "Analysis of steam and hydrogen distributions with PAR mitigation in NPP containments", *Nuclear Engineering and Design*, Vol. 202, 2000, pp. 231–248.

[2] Houkema, M., Siccama, N.B., Nijeholt, J.A.L., Komen, E.M.J., "Validation of the CFX4 CFD code for containment thermal-hydraulics", *Nuclear Engineering and Design*, Vol. 238, 2008, pp. 590–599.

[3] Babić, M., Kljenak, I., Mavko, B., "Prediction of light gas distribution in experimental containment facilities using the CFX4 code", *Nuclear Engineering and Design*, Vol. 238, 2008, pp. 538–550.

[4] Kljenak, I., Babić, M., Mavko, B., Bajsic, I. "Modeling of containment atmosphere mixing and stratification experiment using a CFD approach", *Nuclear Engineering and Design*, Vol. 236, 2006, pp. 1682–1692.

[5] Kljenak, I., Bajsić, I., Babić, M., "Modelling of steam condensation on the walls of a large enclosure using a Computational Fluid Dynamics code", Proceedings of the ASME-ZSIS *International Thermal Science Seminar II*, Slovenia, June 13-16, 2004.

[6] Uchida, H., Oyama, A., Togo, Y., "Evaluation of post-incident cooling systems of LWRs", *13th International Conference on Peaceful Uses of Atomic Energy*, International Atomic Energy Agency, Vienna, Austria, 1965.

[7] Valdepenas, J.M.M., Jimenez, M.A., Fuertes, F.M., Fernandez, J.A., "Improvements in a CFD code for analysis of hydrogen behaviour within containments", *Nuclear Engineering and Design*, Vol. 237, 2007, pp. 627–647.

[8] Terasaka, H., Makita, A., "Numerical analysis of the PHEBUS containment thermal hydraulics", J*ournal of Nuclear Science and Technology*, Vol. 34, 1997, pp. 666– 678.

[9] Mimouni, S., Foissac, A., Lavieville, J., "CFD modelling of wall steam condensation by a two-phase flow approach" Article in Press, *Nuclear Engineering and Design*, doi:10.1016/j.nucengdes.2010.09.020.

[10] Kudriakov, S., Dabbene, F., Studer, E., Beccantini, A., Magnauda, J.P., Paillere, H., Bentaib, A., Bleyer, A., Malet, J., Porcheron, E., Caroli, C., "The TONUS CFD code for hydrogen risk analysis: Physical models, numerical schemes and validation matrix" Nuclear Engineering and Design, Vol. 238,2008, pp. 551–565.

[11] CFD-ACE+ V2009.2, User Manual, ESI CFD Inc., Huntsville, AL 35806

[12] Cheng, X., Bazin, P., Cornet, P., Hittner, D., Jackson, J.D., Jimenez, J.L., Naviglio, A., Oriolo, F., Petzold, H., "Experimental data base for containment thermal hydraulic analysis", *Nuclear Engineering and Design*, Vol. 204, 2001, pp. 267–284.

[13] Brun, P., Cornet, P., Malet, J., Menet, B., Porcheron, E., Vendel, J., Caron Charles, M., Quilico, J.J., Paillere, H., Studer, E., "Specification of international standard problem on containment thermal-hydraulics ISP- 47" Step 1: TOSQAN–MISTRA, Saclay, France, 2002.

[14] Cornet, P., Malet, J., Porcheron, E., "TOSQAN experimental results of the air–steam phase", OECD International Standard Problem on Containment Thermal-Hydraulics ISP-47, Saclay, France, 2002.