Comparison and Analysis of the Condensation Benchmark Results

W. Ambrosini[1], M. Bucci [1,2], N. Forgione[1], F. Oriolo[1], S. Paci[1],
J-P. Magnaud[2], E. Studer[2], E. Reinecke[3], St. Kelm[3], W. Jahn[3],
J. Travis[4], H. Wilkening[5], M. Heitsch[5], I. Kljenak[6], M. Babić[6],
M. Houkema[7], D.C. Visser[7], L. Vyskocil[8], P. Kostka[9], R. Huhtanen[10]

CONTRACT SARNET FI6O-CT-2004-509065

1) Università di Pisa, DIMNP, Pisa, Italy
2) CEA Saclay, LETMF, Paris, France
3) FzJ, IEF-6, Juelich, Germany
4) FzK, IKET, Karlsruhe, Germany
5) JRC, Petten, The Netherlands
6) JSI, Ljubljana, Slovenia
7) NRG S&P, Petten, The Netherlands
8) NRI Rez (UJV), Prague, Czech Republic
9) VEIKI IEPR, Budapest, Hungary
10) VTT, Espoo, Finland

Summary

In the frame of the SARnet Network of Excellence, the need was felt for assessing the status of condensation models adopted in CFD codes relevant for nuclear reactor containment applications. The motivation for this work was provided by the increasingly widespread use of CFD in the analysis of containment behavior consequent to postulated severe accidents, in which wall condensation can promote containment atmosphere mixing. Since standard models are seldom available in many CFD codes for dealing with condensation and in consideration of the different strategies envisaged for analyzing downscaled facilities or full scale containments, this aspect was considered worth of a specific attention.

In this aim, after performing a review of the models available to the Participants in the network, appropriate Benchmark Problems were proposed to assess and compare their behavior. The University of Pisa took the charge of coordinating these efforts, proposing an initial step of the Benchmark (identified as the 0th Step) aimed at comparing code responses among each other and with applicable correlations in the application to a classical problem of condensation on a flat plate; the reference geometrical and operating conditions for this step were selected as an idealization of those typical in the CONAN experimental facility, operated at the University of Pisa. Then, the 1st Step of the activity involved addressing experimental data from the CONAN facility at different steam mass fractions and velocities and the comparison of the measured condensation rates and of local heat fluxes with code predictions.

Both the steps in this activity were fruitful, since they constituted a gradual and relatively systematic approach to the actual experimental conditions, allowing for revising model details and discussing numerical and physical options. Though the comparison with experiments involved up to now only a limited number of data points, the activity is not considered to be completed and additional experimental data will be offered in the future to obtain a broader assessment of codes in conditions of interest for severe accidents in light water reactors.

A. INTRODUCTION

In order to determine the risk associated with the presence of hydrogen in nuclear power plant containments during a hypothetical severe accident, predictive codes are necessary. Computational Fluid Dynamics codes are very promising for this purpose, since they are capable to provide a detailed description of flow patterns and chemical component distributions even in complex geometries, as those involved in nuclear power plant applications.

Among the relevant phenomena to be predicted in this frame, wall condensation plays an important role; in fact, containment behaviour is strongly affected by condensation, also influencing the levels of pressurization and atmosphere mixing. This justifies the considerable efforts spent in the past decades to collect experimental data suitable for model development and assessment, aiming at closing the issue from the point of view of reactor safety applications.

On the side of engineering correlations, the heat and mass transfer analogy constitutes the primary modelling tool for predicting condensation or evaporation; though different formulations of the analogy are available in the literature, they all produce very similar results in most conditions, generally providing consistent estimates of mass transfer rates, whenever appropriate heat transfer correlations are adopted for the specific addressed geometry (see e.g., Refs. 1-4 for a presentation of basic theories and Ref. 5 for a synopsis on them). In order to take full profit of the greater detail obtained in simulating 3D flow and thermal patterns, the adoption of computational fluid-dynamics in analysing reactor containments calls for a more mechanistic perspective in predicting mass transfer. However, the present state-of-the-art in this field shows that a completely mechanistic approach to full scale plant or large experimental facilities requires huge computational resources, suggesting the need for introducing model simplifications.

Actually, the complications involved in the application of CFD to heat and mass transfer phenomena are bound to the usual difficulty in treating the near-wall region (for a general treatment of turbulence models and of near-wall region approaches see Refs. 6 and 7). This aspect, being a crucial one for many flows requiring an accurate representation of boundary layer phenomena, is even more relevant in the case of simultaneous heat and mass transfer, since fluid-dynamic, thermal and concentration boundary layers superimpose and affect each other; transversal motion due to suction or blowing adds a further phenomenon to be accurately represented. In this respect, though it would be desirable to adopt turbulence models having low-Reynolds number capabilities, their application requires so fine discretization of the near-wall region that it is often impractical to represent real life systems with a reasonable computational effort. On the other hand, using wall functions represents a more feasible choice, allowing for the use of coarser meshes; anyway, the problem is faced of the use of standardised variable profiles for conditions which could be far from those in which they were developed and tested, including developing, transient and buoyancy affected flows.

Despite the efforts presently spent to tackle challenging multiphase fluid-dynamic problems, everyday experience shows that even single-phase conditions are sometimes challenging for CFD, no matter if wall functions or low-Reynolds number models are used. This constitutes a motivation to perform in depth analyses of interesting phenomena by CFD codes in order to get experience on their capabilities for relevant applications.

In the frame of the Severe Accident Research Network (SARnet), Benchmark exercises were proposed to compare and validate the different available CFD models in predicting heat and mass transfer during condensation in the presence of noncondensable gases. The activities were performed in two steps, proposing a 0^{th} level problem and a 1^{st} level one [8-9], addressing respectively idealised conditions and real experimental data. In the first case, a geometry similar to the one of the CONAN facility [10], operated at the University of Pisa, was considered, assuming constant condensing surface temperature conditions that cannot be actually achieved in experiments; in this step, code results for heat and mass transfer were compared with engineering correlations credited to be applicable in the considered conditions. In the second case, boundary conditions from CONAN experiments were provided to Participants, requesting blind predictions of the overall condensation rate and of the local heat fluxes and surface temperatures, to be compared with available measurements. Actually, performing the 0^{th} step was very useful as a preparation to the 1^{st} one, since models could be firstly checked and assessed.

The paper summarises the lessons learned from both steps of the work, highlighting the relevant conclusions drawn from the activities performed up to now and suggesting the desirable future developments for severe accident evaluation.

B. STEP 0: CONDENSATION ON AN ISOTHERMAL FLAT PLATE

This initial step of the benchmarking activity was conceived in order to let the Participants have a first approach to the problem to be subsequently dealt with on the basis of experimental data, by considering an idealised version of it. The step was prepared by a work autonomously performed by the University of Pisa and later published in a journal paper [11], in which use was made of different turbulence models to address similar conditions. Actually, the focus of the paper was on the prediction of transpiration effects, comparing the responses obtained by CFD models with classical expressions adopted to account for suction and blowing in the application of the analogy between heat and mass transfer.

The objective of the 0^{th} Step was instead to compare code results with correlations considered applicable to the addressed problem. Reference was made to the 2D computational domain sketched in Figure 1, to be used in two different ways:

- by pure convective heat transfer calculations (no steam condensation), capable to highlight the adequacy of the adopted turbulence models and of the selected numerical grids in reproducing the heat transfer trends predicted at large values of the local Reynolds number by the correlation

$$Nu_x = 0.0296\, Re_x^{0.8} Pr^{0.33} \tag{1}$$

- by simultaneous heat and mass transfer calculations, whose results should be compared among each other and with the correlation drawn from the previous one by the application of the analogy between heat and mass transfer

$$Sh_{0,x} = 0.0296\, Re_x^{0.8} Sc^{0.33} \tag{2}$$

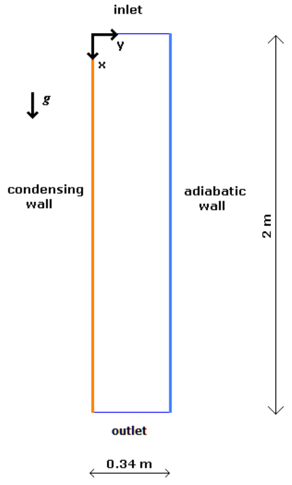


Figure 1. Geometrical configuration of the proposed computational domain for 0^{th} Step.

Eqs. (1) and (2) address external flow over a flat plate. These equations can be applied to the proposed internal flow problem on the basis of the previous work [11], considering the sufficiently large depth of the channel.

The rationale at the basis of the proposed route for code assessment in this 0^{th} Step can be summarised as follows:

- models used for predicting condensation over vertical surfaces should at least be capable to correctly predict pure convective heat transfer; in the lack of such a capability to adequately represent boundary layer phenomena, models should be reconsidered in their physical and/or numerical aspects before addressing the most challenging heat and mass transfer problem;
- condensation adds a further challenge to the capabilities in modelling boundary layer phenomena, since temperature, velocity and concentration distributions are distorted by the presence of transversal flow; the definition of mass and energy sinks owing to condensing steam and a consistent modelling of species diffusion represent further code features to be assessed by comparison with available correlations;
- actually, since the mass transfer correlation in Eq. (2) is obtained from Eq. (1) through the application of the analogy between heat and mass transfer, the validity of the analogy is an additional aspect to be discussed in the analysis of the resulting local mass transfer data.

Concerning the latter of the above items, Participants in the benchmark were asked to calculate the Sherwood numbers as a function of the calculated mass fluxes basing at least on the following two expressions:

$$Sh_{0,x,mass} = \frac{\dot{m}_i''}{\dfrac{\rho \mathcal{D}}{x} \log\left(\dfrac{Y_{n,bulk}}{Y_{n,wall}}\right)} \qquad Sh_{0,x,molar} = \frac{\dot{m}_i''}{M_v \dfrac{c\,\mathcal{D}}{x} \log\left(\dfrac{X_{n,bulk}}{X_{n,wall}}\right)} \tag{3}$$

In agreement with the theories at their basis and with information from practical use [5], these two formulations provided very similar results, coherently with the corrections for suction effects proposed on the basis of the solution of the Stefan's problem (see e.g., Refs. 1 and 4). On the other hand, classical definitions were proposed for the Nusselt number.

Table 1 summarises the values of the relevant independent parameters in the proposed eight cases involving both pure heat transfer (HT cases) and heat and mass transfer (HMT cases).

Test name	w_{inlet} **[m/s]**	T_{wall} **[K]**	T_{inlet} **[K]**	$Y_{v,inlet}$**[-]**	P **[Pa]**
HT-30-3	3	303.15	363.15	0	101325
HT-30-6	6	303.15	363.15	0	101325
HT-60-3	3	333.15	363.15	0	101325
HT-60-6	6	333.15	363.15	0	101325
HMT-30-3	3	303.15	363.15	saturation	101325
HMT-30-6	6	303.15	363.15	saturation	101325
HMT-60-3	3	333.15	363.15	saturation	101325
HMT-60-6	6	333.15	363.15	saturation	101325

Table 1. Boundary conditions for the 0^{th} Step cases.

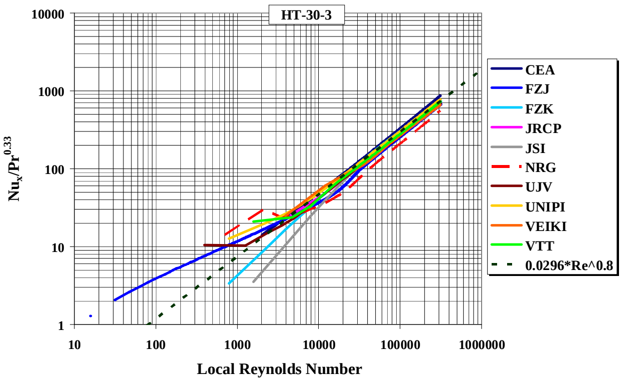
The values of inlet velocity were selected in order to allow the analysis of computed data at sufficiently large Reynolds numbers, to assure that the forced convection correlations are applicable at least in the last part of the channel. The heat and mass transfer cases involved a saturated air and steam mixture at the assigned inlet temperature. The two values of temperature of the condensing wall, assumed to be uniform, represent actually gas-liquid interfacial values, since no falling film is considered at the wall; this assumption was introduced to avoid complications due to different modelling of the falling film by the Participants.

Ten organisations participated in this step of the Benchmark adopting different codes and models, as reported in Table 2.

Organization	CODE	Main Model Characteristics
CEA Saclay, France	**CAST3M**	• Mixing length model with low-Re capabilities. • Finite elements, second order upwind, fully implicit.
Forschungszentrum Juelich (FzJ), Germany	**CFX 10**	• SST Model. Diffusive approach. • Finite volume discretization. High resolution advection scheme.
Forschungszentrum Karlsruhe (FzK), Germany	**GASFLOW-II**	• Modified standard k-ε model with a 3-layer wall treatment. HTC evaluated on the basis of the 3-layer treatment and the analogy. • Modified Arbitrary-Lagrangian-Eulerian finite volumes, 1st order.
JRC Petten, The Netherlands	**CFX 10**	• SST Model. Diffusive approach. • Finite volume discretization. High resolution advection scheme.
Jozef Stefan Institute, Slovenia	**CFX 4.4**	• k-ε model. Uchida correlation for condensation HTC. • Finite volume discretization.
NRG, The Netherlands	**FLUENT 6.3.26**	• Launder and Sharma low-Re. Diffusive approach • Finite volume discretization. 1st order advection terms.
UJV Rez, Czech Republic	**FLUENT 6.1.22**	• Realizable k-ε model with enhanced wall treatment. Diffusive approach, accounting for the presence of falling film. • Finite volume discretization. 2nd order advection terms.
University of Pisa, Italy	**FLUENT 6.1.16**	• RNG k-ε model with enhanced wall treatment. Diffusive approach. • Finite volume discretization. 1st order advection terms.
VEIKI Institute for Electric Power Research	**GASFLOW 2.4**	• Standard k-ε model with law-of-the-wall. HTC based on the analogy. • Van Leer's second order, slope-limiting advection scheme.
VTT, Finland	**FLUENT 6.2.16**	• k-ε turbulence model with enhanced wall functions • Wall condensation model implemented by GRS in the CFX-4 code. • Finite volume discretization. Second order upwind advection.

Table 2 – Participants and adopted models in the Condensation Benchmark 0th Step.

As it can be noted from Figure 2, all the codes were reasonably successful in predicting the asymptotic trend of the correlation in the heat transfer cases; this testifies for an adequate representation of boundary layer phenomena with the adopted models and discretizations. On the other hand, Figure 3 shows a greater degree of spread in the asymptotic power-law trends obtained by the different codes in the prediction of the Sherwood number; this spread is anyway limited, except in the case of predictions by JSI, which are out of range because of the use of a different definition of Sherwood number and the adoption of the Uchida correlation for a forced convection case.



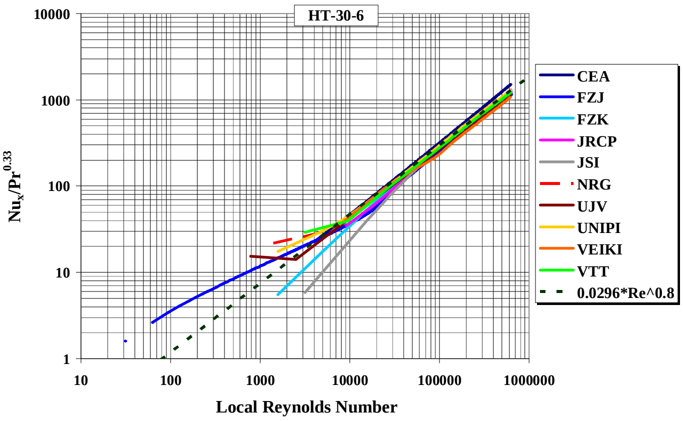
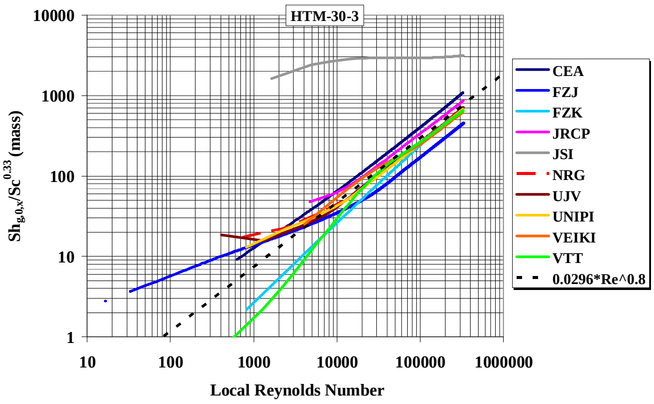


Figure 2. Results obtained for two heat transfer cases in the 0^{th} Step.



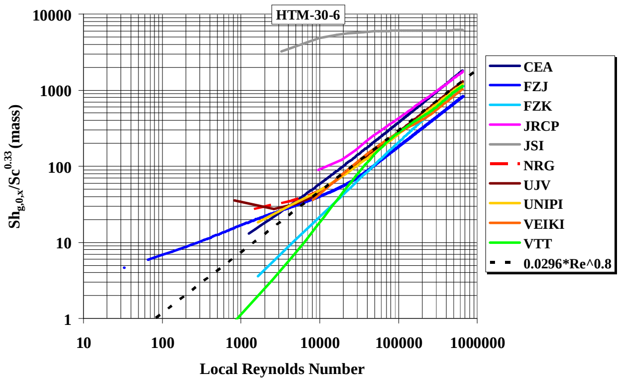


Figure 3. Results obtained for two heat and mass transfer cases in the 0^{th} Step.

C. STEP 1: CONDENSATION IN THE CONAN EXPERIMENTAL FACILITY

The next step in the benchmark addressed experiments purposely performed in the CONAN experimental facility, installed at the University of Pisa [10]. To the purposes of the benchmark exercise, the test section can be represented as shown in Figure 4.

The facility includes a primary loop containing the test section, being a 2 m long square cross section channel (0.34 × 0.34 m). One of the sides of the test channel is made of a 4.5 cm thick aluminium plate cooled on the rear surface by secondary water flowing in a 5 mm deep rectangular channel. The primary steam and air mixture enters nearly at atmospheric pressure at the top of the channel in downward direction, with measured velocity, temperature and relative humidity. The lateral surfaces of the channel can be assumed to be adiabatic, except for the thick aluminium plate. The cooling water flow enters the secondary channel in upward direction and its inlet and outlet temperatures are measured, together with its flow rate, in order to provide the relevant thermal boundary conditions to be adopted in code analyses. Upstream the inlet section of the channel, three 35 mm thick panels of 3 mm cell diameter honeycomb are placed in order to improve the uniformity of the inlet velocity distribution.

Along the aluminium plate, calibrated K-type thermocouples are placed at different depths from the surface in order to evaluate the local superficial temperature as well as the local heat flux; in addition to the usual calibration, the thermocouples lying along the axial centreline of the aluminium plate underwent a thermal gradient zeroing procedure, performed during nearly adiabatic tests carried out at different temperatures; this assures an improved accuracy in the measurement of heat flux. Condensate flow rate is collected at the bottom of the plate by a gutter and routed to a collecting vessel, where its level is measured as a function of time by a differential pressure transducer.

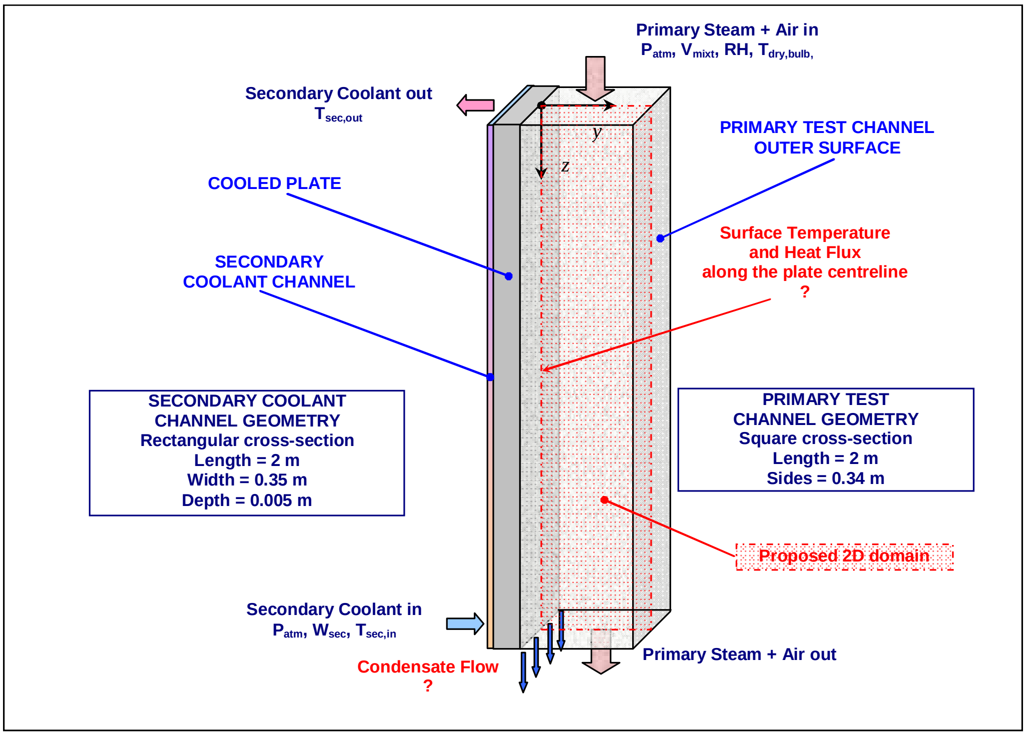


Figure 4. Sketch of the CONAN facility test section.

Measurement	Device	Uncertainty
Primary volumetric flow rate	Vortex Flow Meter	± 21 m^3/s corresponding to ± 0.05 m/s on channel velocity
Mass flow rate in secondary loop	Coriolis Flow Meter	± 3%
Temperature measurements (general)	K-type TCs RTDs	± 0.25 °C
Temperature difference between plate centerline thermocouples at different depths	K-type TCs	max. ± 0.10 °C
Wet and dry bulb temperatures	RTDs	max. ± 0.10 °C
Heat flux (resulting from centerline TCs)	K-type TCs	± 700 W/m^2
Condensate flow rate	Differential pressure transducer (level increase rate)	Smaller than ± 1%

Table 3. Best estimates of measuring errors relevant for the Step-1 of the benchmark exercise.

The uncertainties involved in the relevant measurements for the benchmark exercise are reported in Table 3. It should be understood that these estimates are related to stationary measurements, while experimental tests, performed recording nearly steady-state conditions in time periods as long as 600 – 1000 s, are unavoidably affected by small fluctuations, eliminated in data post-processing by averaging the time series.

Experimental data from five experimental tests were proposed to Participants. These data were related to operating conditions characterised by a nominal value of the secondary coolant close to 30 °C, a steam generator power of 10 kW and mixture velocities from 1.5 to 3.5 m/s. In Figure 4, the data provided to Participants for each proposed data point are highlighted using the blue colour, while the values to be provided are the distributions of plate surface temperatures and heat fluxes along the centreline and the total collected condensation rate.

On the basis of previous modelling experience at the University of Pisa, it was suggested to make use of a 2D computational domain addressing the channel middle plane orthogonal to the cooled plate and parallel to its longitudinal axis (see Figure 4). Values of the turbulence intensity and of integral turbulence length scale were also suggested, leaving anyway to Participants the freedom to perform sensitivity analyses on these parameters on the basis of their engineering skills.

For dealing with condensing plate boundary conditions, the limited number of values of plate surface temperatures obtained from the measurements suggests that it is not advisable to assign any elaboration of such data in the form of interpolating functions. Previous experience at the University of Pisa showed that the use of the secondary coolant parameters and the heated plate conductivity to define appropriate 3rd kind boundary conditions at the rear of the plate surface is much more effective in providing less questionable input parameters to perform the calculations. In this regard, two different modelling strategies were suggested:

- simulating the heat conduction behaviour of the 4.5 cm thick cooled aluminium plate by a conjugated heat transfer approach, imposing a linear trend of the secondary coolant temperature distribution and a suitable value of the heat transfer coefficient at the rear surface;

- considering an equivalent plate thermal resistance to be assigned in series with the convective one at the rear of the plate.

Since exploratory calculations previously performed at the University of Pisa showed only minor differences between the results of the two approaches, the choice between them was left to the Participants.

Nine organizations analyzed the proposed 1st Step data (CEA, FzJ, FzK, JRC Petten, JSI, NRG, UJV, UNIPI and VEIKI). The adopted models were mostly the ones applied by the same organizations in the case of 0th Step. Most of the Participants simulated the cooled plate making use of an equivalent resistance (the second of the above described approaches), with the exception of the FzJ and UNIPI, that applied also the conjugated heat transfer approach Also, CEA performed analyses for heat conduction in the plate, though the related results are not reported herein.

The results obtained in terms of overall condensation rates are compared with experimental data in Figure 5. As it can be noted, there is a general tendency to underestimate the measured condensation rate. In particular, it can be noted that a group of models (adopted by CEA, NRG, UJV, UNIPI and VEIKI) provides very similar results, slightly underestimating the experimental data of condensation rate. The common qualitative trend of the discrepancies with respect to experimental values suggests also common reason for this behavior. Systematic trends in relation to the effect of the slight degree of superheating existing in some conditions suggest a future specific analysis of the effect of this parameter.

The model by JSI, owing to the use of the Uchida correlation, seems not to catch the correct trend of condensation rate; in particular, since the experimental data involve different values of the ratio between steam and noncondensable gas density, being the most relevant parameter in the Uchida correlation, the obtained predictions are very sensitive to this ratio, disregarding other effects. Such discrepancies were completely within the expectations of JSI that adopted the Uchida correlation as a simple trial to estimate the error coming from the use of such a simplified formulation in the present conditions.

The effects of the assumptions made for heat transfer through the plate (conjugated heat transfer or equivalent plate thermal resistance) appears negligible. This can be noted comparing the results obtained by FzJ and UNIPI with the two approaches. The obvious reason for this behavior is the minor role of 2D heat conduction effects within the cooled plate.

The general underestimation of condensation rate seems to be the consequence of a systematic inadequacy in evaluating the heat flux distribution at channel inlet. This can be noted, for instance, in Figure 6 and Figure 7, reporting the data obtained by CEA and UJV. It can be noted, that the local heat flux is systematically underestimated by the calculations in the entrance region of the channel, where the codes predict a quicker extinction of the entrance effects with respect to the experiments. This behavior, not yet completely understood, has been the subject of subsequent sensitivity analyses performed by changing the turbulence intensity and the integral length scale at the inlet as well as considering velocity distributions different from the simple flat one suggested for the analysis. It is anyway possible that other effects, e.g., the actual 3D nature of phenomena in the entrance region, should be called upon to justify the observed discrepancy.

Nevertheless, it is apparent that the use of different turbulence models plays a role in this respect. The near-wall model adopted by FzK in GASFLOW, in fact, provides a different heat flux distribution at channel entrance with respect to other models, showing compensating effects that make the calculated results for overall condensation rate generally closer to the experimental ones.

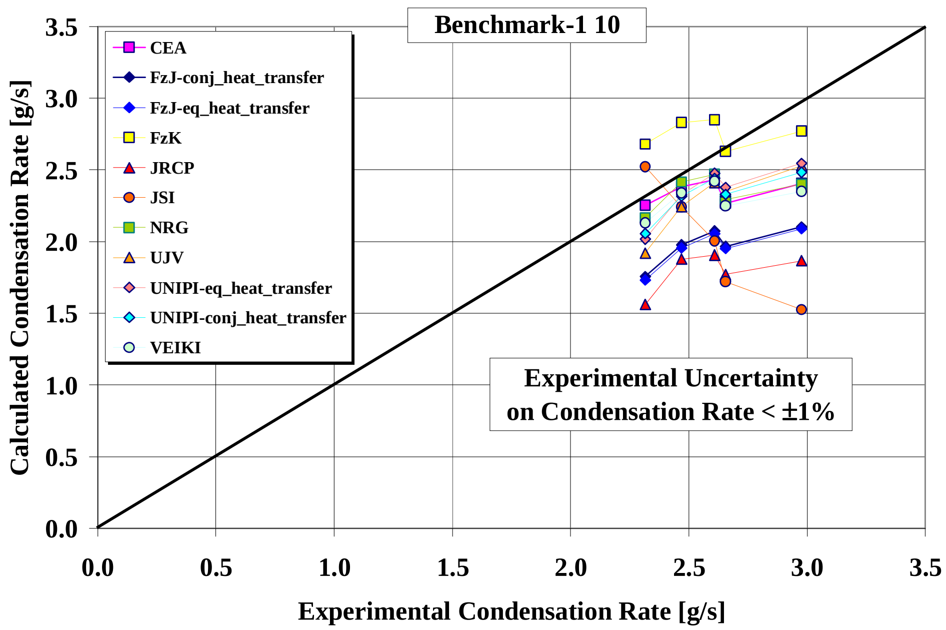


Figure 5. Calculated vs. experimental overall condensation rate predicted by Participants for the 1st Step of the benchmark exercise.

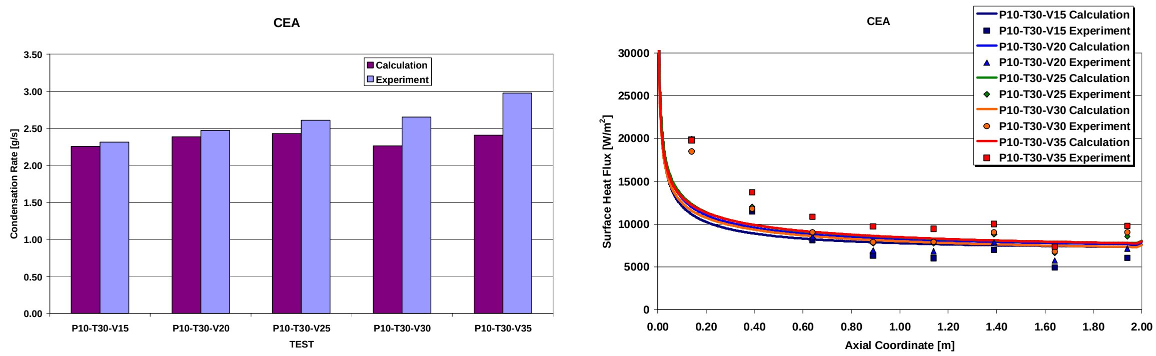


Figure 6. Sample comparison of calculated and experimental values of condensation rate and local heat flux (data by CEA).

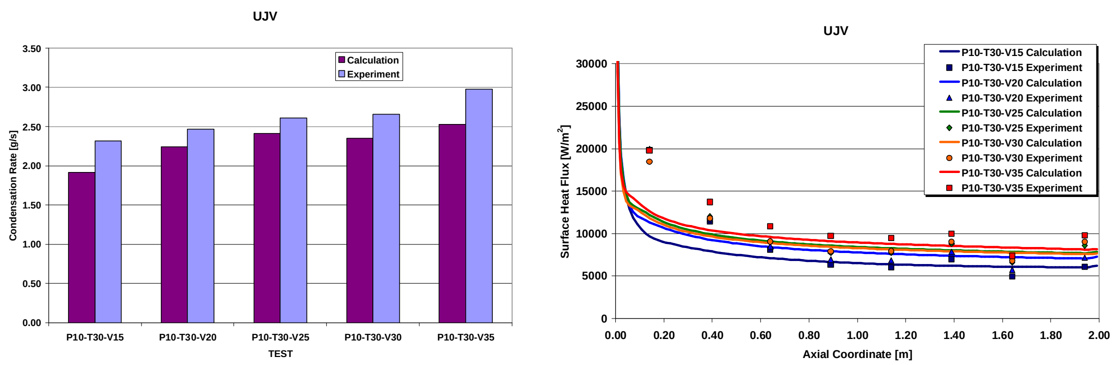


Figure 7. Sample comparison of calculated and experimental values of condensation rate and local heat flux (data by UJV).

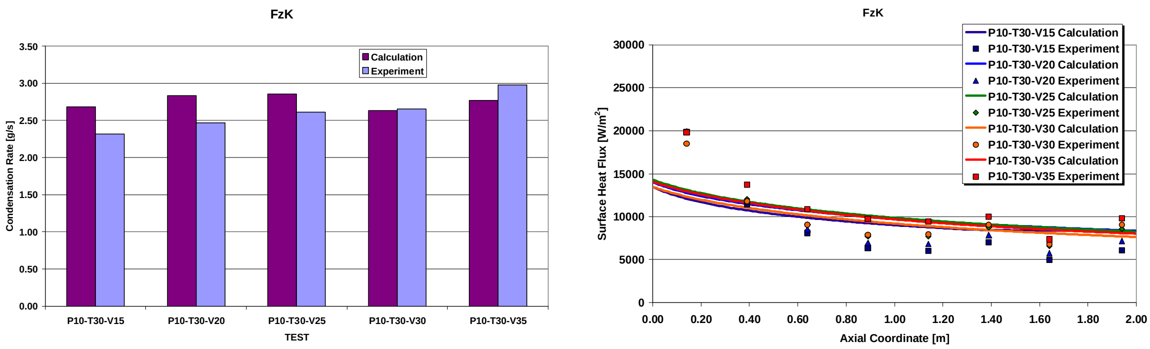


Figure 8. Sample comparison of calculated and experimental values of condensation rate and local heat flux (data by FzK).

D. CONCLUSIONS AND FUTURE WORK

The two steps of the benchmarking activity performed up to now allowed assessing the behaviour of the models adopted in the frame of the Severe Accident Research Network. Though the proposed problems were related to forced convection condensation, which is often of lower interest for containment analyses with respect to free convection, the availability of the CONAN facility offered the opportunity to address in a systematic way the capabilities of codes in predicting experimentally observed behaviour.

Summarising the relevant conclusions achieved up to now, it can be noted that:

- though at different extents, all the adopted CFD models are reasonably in agreement with the information at the basis of a well known correlation for forced convection heat transfer on a flat plate;
- even in the idealised case of condensation over an isothermal flat plate, most codes provided a reasonable prediction of the behaviour expected on the basis of the analogy between heat and mass transfer, though in this case the spread in the obtained results around the correlation was larger;
- the application to actual experiments revealed more details on the behaviour of models, highlighting a general tendency to underestimate entrance effects, whose reasons are still matter of analysis.

It must be noted that most of the models applied in both steps made use of near-wall treatments having low-Reynolds number capabilities. This approach is promising but still relatively expensive from the computational point of view if applied to large facilities or a full scale plant. Further efforts must be therefore spent in order to develop accurate but affordable techniques for predicting near-wall behaviour without loosing too much about the necessary quantitative details.

The participation of a number of different organizations in both the steps of the benchmarking activity performed up to now testifies the strong interest in the addressed subject. In addition, the results obtained by the exercise underline the need to refine CFD tools in accordance with the available experimental information. The rather enthusiastic adhesion for both steps of the activity and the repeated encouragement expressed by Participants to the University of Pisa to go ahead in proposing new experimental data for code assessment, possibly including also the effect of light gases, represent a strong stimulus to continue this activity in the next future.

Nomenclature

Latin Letters

c	molar concentration [mol/m^3]
$\mathcal{D}$	diffusion coefficient [m^2/s]
$\dot{m}_i''$	mass transfer rate [kg/(m^{2}s)]
M_v	molar weight of vapour [kg/mol]
Nu_x	local Nusselt number [-]
P	pressure [Pa]
Pr	Prandtl number [-]
Re_x	local Reynolds number [-]
Sc	Schmidt number [-]
$Sh_{0,x}$	local Sherwood number [-]
T	temperature [°C]
x	axial coordinate [m]
X	molar fraction [-]
Y	mass fraction [-]

Greek Letters

ρ	fluid density [kg/m^3]

Subscripts

bulk	referred to bulk fluid
inlet	inlet conditions
n	noncondensable (air)
v	vapour
w	wall

References

[1] R. Bird, W.E. Stewart and E.N. Lightfoot, 1960, Transport Phenomena, Wiley, New York, 1960.

[2] D. Butterworth and G.F. Hewitt, 1977, Two-Phase Flow and Heat Transfer, Oxford University Press, 1977.

[3] J.G. Collier, 1972, Convective Boiling and Condensation, McGraw -Hill Book Co., 1972.

[4] J.H. Lienhard, 1987, A Heat Transfer Textbook", Prentice Hall, Englewood Cliffs, 1987.

[5] W. Ambrosini, N. Forgione, A. Manfredini, F. Oriolo, “On Various Forms of the Heat and Mass Transfer Analogy: Discussion and Application to Condensation Experiments”, Nuclear Engineering and Design, Vol. 236, pp. 1013-1027, 2006.

[6] D.C. Wilcox, Turbulence Modeling for CFD, DCW Industries, Second Edition, 1998.

[7] S.B. Pope, Turbulent Flows, Cambridge University Press, 2000.

[8] W. Ambrosini, M. Bucci, N. Forgione, F. Oriolo, S. Paci, Results of the SARnet Condensation Benchmark No. 0, Università di Pisa, DIMNP 006(2007), Pisa, August 2007.

[9] W. Ambrosini, M. Bucci, N. Forgione, F. Oriolo, S. Paci, Quick Look on SARnet Condensation Benchmark-1 Results, Università di Pisa, Step 1 - 10 kW Heating Power Exercise, Pisa, February 22nd, 2008

[10] Walter Ambrosini, Nicola Forgione, Francesco Oriolo, Caro.line Dannöhl, Holger J. Konle, Experiments and CFD Analyses on Condensation Heat Transfer on a Flat Plate in a Square Cross Section Channel, 11th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11), vol. CD-ROM, pp. 1-18, Popes’ Palace Conference Center, Avignon, France, October 2-6, 2005.

[11] M. Bucci, M. Sharabi, W. Ambrosini, N. Forgione, F. Oriolo, S. He, Prediction of transpiration effects on heat and mass transfer by different turbulence models, *Nuclear Engineering and Design*, 238 (2008) 958–974.