CFD CODE VALIDATION AND BENCHMARKING AGAINST BFBT BOILING FLOW EXPERIMENT

M. C. Galassi, F. Moretti, F. D'Auria

University of Pisa - 2, Via Diotisalvi, 56126 Pisa (PI) ITALY

Abstract

The paper presents CFD code validation and benchmarking against the NUPEC BFBT (BWR Full-Size Fine-Mesh Bundle Tests) data, performed within the OECD/NRC BFBT void distribution benchmark and as part of the NURESIM (European Platform for Nuclear Reactor Simulations) Integrated Project.
Sub-cooled flow boiling occurs in many industrial applications and is characterized by large heat transfer coefficients until reaching the critical heat flux (CHF) condition. This phenomenon is a crucial design limitation factor in water cooled nuclear reactors, since it causes a rapid heater temperature excursion which potentially leads to heater melting and destruction.
A common objective of NURESIM IP and BFBT Benchmark is to improve the use of two-phase CFD codes as a tool for understanding boiling flow processes, in order to subsequently help new fuel assembly design and to develop better CHF predictions. In this framework, numerical simulations of sub-cooled boiling in BWR fuel assembly were performed with NEPTUNE_CFD and ANSYS CFX codes. Sensitivity studies on problem parameters led to satisfactory overall void production, with some differences in the vapour distribution through the section probably due to the absence of spatial grids in the model. Further investigations should focus on non-drag forces modelling and grid refinement.

1. INTRODUCTION

Sub-cooled flow boiling occurs in many industrial applications and is characterized by large heat transfer coefficients. However, this efficient heat transfer mechanism is limited by the critical heat flux (CHF) where the heat transfer coefficient decreases leading to a rapid heater temperature excursion, which potentially leads to heater melting and destruction. The critical flux phenomenon in forced convective boiling is a design limitation factor in water cooled nuclear reactors, since ensuring the integrity of fuel rods in the core is a primary issue for nuclear reactor safety. CHF in LWRs is currently estimated by empirical correlations, with limited applicability to specific conditions and geometries, and expensive full-scale measurements. To overcome these problems and improve two-phase flows modelling, it is desirable to develop a mechanistic approach able to predict CHF conditions in a boiling channel based on local parameters. The need to refine models for best-estimate calculations has already been expressed in many recent conferences in the field of nuclear applications. Moreover, these needs should not be limited to the currently available macroscopic methods but should be extended to next-generation analysis techniques that focus on more microscopic processes, such as CFD approaches. Infact, CFD methods could provide accurate prediction of CHF conditions, since they combine different scale effects such as flow distribution in a rod bundle (macro-scale) and bubble nucleation at the heated surface (micro scale). Nevertheless, two-phase CFD predictions should be compared to relevant experimental data in order to validate two-phase flow modelling in geometrical and thermal-hydraulics conditions possibly representative of the industrial ones.
The BFBT Void Distribution Benchmark (OECD-NEA web site, 2007) is based on BWR Full-size Fine-mesh Bundle Tests data made available by the NUPEC (Nuclear Power Engineering Corporation) database, which is one most valuable databases identified for thermal-hydraulics modelling. The NUPEC database includes sub-channel void fraction and critical power measurements in a representative (full-scale) BWR Fuel Assembly. The high resolution and high quality of sub-channel void fraction data encourage advancement in understanding and modelling complex two-phase flow behaviour in real bundles, and make BFBT experiments valuable for CFD multiphase models validation.

In this paper, CFD code validation and benchmarking against BFBT data will be presented within the OECD/NRC BFBT void distribution benchmark and as part of the NURESIM (European Platform for Nuclear Reactor Simulations) Integrated Project (Guelfi, 2005). Numerical simulations of sub-cooled boiling flow in BWR fuel assembly were carried out with the research code NEPTUNE_CFD V1.0.6 (Méchitoua, 2003) and the commercial code ANSYS CFX 11.0.

2. DESCRIPTION OF BFBT EXPERIMENTS

The test facility, shown in Figure 1, is able to simulate the high-pressure and high temperature fluid conditions found in BWRs (Inoue, 1995). An electrically-heated rod bundle has been used to simulate a full scale BWR fuel assembly. De-mineralized water is used as a cooling fluid. The maximum capacity of the facility are extensively Sartori et al. (2004).

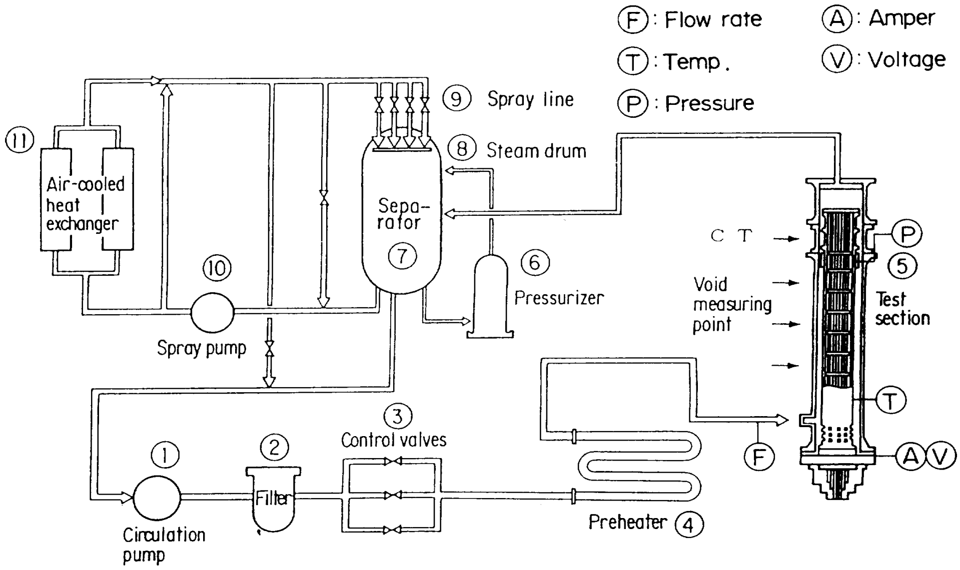

Fig. 1: System diagram of the test facility.

The test section consists of a pressure vessel (where the full-scale model of BWR fuel assembly is installed), a simulated flow channel, and electrodes. The test assembly is constituted by electrically heated rods fixed in an array identical to that of the BWR fuel assembly being simulated. Each rod in the test assembly is indirectly heated, to reproduce an actual reactor power profile generated by nuclear fission. Five types (Type 0 to Type 4) of test assemblies with different combinations of cross section geometries and power shapes were considered.

For the microscopic grade benchmark on void distribution, four steady state tests on Assembly 4 were proposed, with the same nominal conditions for pressure (7.2 MPa), inlet subcooling (50.2 kJ/kg), flow-rate (55 t/h), and different exit quality (2-25%); the selected assembly was the 8×8 high burn-up fuel bundle shown in Figure 2, with uniform distribution in the axial direction and not uniform radially through the section (Utsuno, 2004).

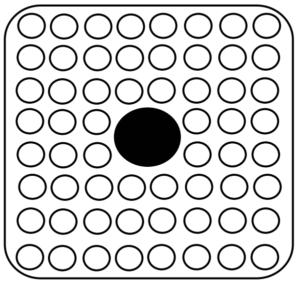

Fig. 2: Assembly type No.4 (high burn-up).

Two kinds of void distribution measurement systems were employed in this tests: the X-ray CT scanner and the X-ray densitometer. Under steady-state conditions, fine mesh void distributions were measured using the X-ray CT scanner located 50 mm above the heated length (which measures 3.708 m).

3. WALL BOILING MODELLING

During subcooled boiling flow, heat and mass exchange between the liquid and gas phases takes place at the heated wall and in the subcooled liquid flow. Where boiling occurs, the specified external heat flux is distributed between the liquid and the gas phases: vapour bubbles are generated on the heated surface (through nucleation processes), then they migrate towards the (subcooled) liquid bulk, where they condense and release the latent heat. Related to this flow configuration, the general modelling approaches, the mathematical formulations, and the particular constituent model correlations are mostly common for NEPTUNE_CFD and CFX (Muehlbauer, 2004).

The wall boiling model implemented in the codes, as documented by Morel et al. (2005) and Burns et al. (2004), is based on the "wall heat partitioning" model of Kurul and Podowski (1991). According to such approach, the total wall-to-liquid heat flux splits into three different modes of heat transfer:

$$q''_{wL} = q_c + q_q + q_e \tag{1}$$

where q_c is the single-phase convection heat flux transferred to the liquid phase near the wall outside the area influenced by nucleating bubbles A_b (A_c=1- A_b); q_q is the quenching heat flux transferred to the subcooled liquid from the bulk flow that fills the volume vacated by departing bubbles; q_e is the fraction of the wall heat flux that is directly used to generate vapour bubbles.

The single-phase convection heat flux is given by

$$q_c = A_c h_c (T_w - T_L) \tag{2}$$

where T_w is the wall temperature and h_c is a heat exchange coefficient given by

$$h_c = \rho_L C_{pL} \frac{u^+}{T^+} \tag{3}$$

with u^+ the wall function velocity and T^+ the non-dimensional liquid temperature. The first is calculated from to the logarithmic law of the wall for liquid velocity in the wall boundary layer; the second is modelled in an analogous way, according to the thermal boundary layer definition.

The quenching heat flux can be expressed trough the following correlation:

$$q_q = A_b h_b (T_w - T_L) \tag{4}$$

where the heat exchange coefficient h_q is defined as

$$h_q = t_q f \frac{2\lambda_L}{\sqrt{\pi a_L t_q}} \tag{5}$$

and A_b is the wall portion occupied by bubble nucleation; f is the bubble detachment frequency, t_q is the quenching time and a_L is the liquid thermal diffusivity.

$$\begin{aligned} A_b &= \min(1, n\pi d_d^2/4) \\ A_c &= 1 - A_b \end{aligned} \tag{6}$$

where n corresponds to the active nucleation sites density and d_d is the bubble detachment diameter. The factor 1/4 is not present in the A_b definition for CFX 11, and A_c assumes the maximum value between (1-A_b) and 10^{-4}.

The nucleation site density is given by

$$n = [X(T_w - T_{sat})]^Y \tag{7}$$

where for NEPTUNE_CFD, X=210 and Y=1.8, while for CFX, X=185 and Y=1.805.

The bubble departure diameter is differently modelled in the two codes; the Unal's model for pipe boiling is implemented in NEPTUNE_CFD:

$$d_d = 2.42 \cdot 10^{-5} p^{0.709} \frac{a}{\sqrt{b\varphi}} \tag{8}$$

with the pressure p and

$$a = \frac{(T_w - T_{sat})\lambda_s}{2\rho_v \ell \sqrt{\pi a_s}} \qquad b = \begin{cases} \dfrac{T_{sat} - T_L}{2(1-\rho_v/\rho_L)} & St < 0.0065 \\ \dfrac{1}{2(1-\rho_v/\rho_L)} \dfrac{q_c + q_q + q_e}{0.0065\rho_L C_{pL} V_L} & St > 0.0065 \end{cases} \tag{9}$$

$$\varphi = \max\left(1, \left(\frac{V_L}{V_0}\right)^{0.47}\right) \tag{10}$$

where λ_s and a_s are respectively the wall conductivity and thermal diffusivity; ρ_v denotes the vapour density and ℓ is the latent heat of vaporization. V_L is the norm of liquid velocity and V_0=0.61 m/s.
The following model by Tolubinsky and Kostanchuk is implemented in CFX:

$$d_{bw} = \min\left(1.4[\mathrm{mm}], 0.6[\mathrm{mm}] \cdot \exp\left(-\frac{\Delta T_{sub}}{45[\mathrm{K}]}\right)\right) \tag{11}$$

where $\Delta T_{sub}=T_{sat}-T_L$. This model is derived for the high-pressure water boiling experimental data with the upper limit for the bubble departure diameter (d_b = 1.4 mm). This correlation depends solely on characteristic liquid subcooling, which makes this correlation case dependent (Burns, 2004).
The quenching time and the bubble detachment frequency are given by:

$$t_q = 1/f \qquad f = \sqrt{\frac{4}{3}\frac{g|\rho_v - \rho_L|}{\rho_L d_d}} \tag{12}$$

The second expression gives a simple estimation of the bubble departure frequency as the terminal rise velocity over the bubble departure diameter.
Finally, the evaporation heat flux can be derived from the evaporation mass flux:

$$q_e = \dot{m}_w \ell = f\,\frac{\pi d_d^3}{6}\rho_V n \ell \tag{13}$$

with $\dot{m}_w$ the evaporation mass flux on the wall, modelled (in a mechanistic way) taking into account the total mass of bubbles periodically departing from nucleation sites.

4. NEPTUNE_CFD SIMULATIONS

Among the available tests for the microscopic-grade benchmark on void distribution, only the first one (smallest power and exit quality) was simulated with both NEPTUNE_CFD and CFX codes. Considered boundary conditions are summarised in Table 1.
Due to the long calculation time required for a fuel assembly simulation and to the complexity of the problem, first attempt investigations were focused on a single sub-channel.

4.1 Sub-channel Simulations

Thanks to the sub-channel geometrical symmetry, simulations were run considering only 1/8 of a sub-channel; the real (asymmetric) boundary conditions for a specific sub-channel in the assembly were not considered, imposing symmetric boundary conditions to the side faces (except to the heated wall). All remaining BCs were imposed according to experimental data reported in Table 1. Simulation time of 10 s was largely enough to reach steady state conditions.

Table 1: Sample Table of experimental data

Measured quantity	TEST 4101-53
Pressure [MPa]	7.159
Liquid inlet velocity [m/s]	2.19
Liquid inlet Temperature [K]	551.24
Heat flux at heated wall [W/m^2]	$1.44 \cdot 10^5$

Calculations were run adopting some standard models available in NEPTUNE_CFD (exhaustively described by Morel et al. (2005)) and then performing several sensitivity studies, varying the main modelling parameters and the grid refinement (from 8000 to 36000 elements), as suggested from Best Practice Guidelines for the use of CFD codes (NEA/CSNI, 2007). Specifically, calculations were run changing the bubble mean diameter value (constant value imposed by the user), the contribution of interface interaction forces (Drag; Lift, Turbulent Dispersion and Added Mass forces), the adopted models for turbulence, heat transfer and wall boiling.

As a result, increasing the mean bubble diameter produced a different void distribution, with higher vapour concentration near the heated wall and higher section averaged void fraction. Without vapour turbulence modelling, all produced steam remained confined in the near-wall region, were the void fraction rapidly increased to very high values; on the contrary, modelling vapour turbulence made the vapour generated at the wall to diffuse into the sub-channel, while the averaged vapour volume fraction was nearly unchanged. Neglecting the non-drag forces contribution generated a high vapour concentration in the near wall region and a relatively smaller one in the centre of the channel, while considering Turbulent Dispersion, Added Mass and Lift forces effects gave a more uniform void distribution taking away the vapour from the wall. Considering a finer grid did not change significantly void fraction predicted values (neither the averaged value) but put in evidence some numerical oscillations in the near wall regions.

4.2 Fuel Assembly Simulations

Due to increasing computational costs only one eighth of the fuel assembly was considered, taking into account appropriate boundary conditions and geometrical symmetries.

Results coming from sub-channel analysis helped in the input setup: reference calculation was run considering NEPTUNE_CFD standard models for interface drag, heat transfer and wall boiling, together with vapour turbulence modelling, mean bubble diameter equal to 0.1 mm, and also taking into account non-drag forces contribution (Lift, Added Mass and Turbulent Dispersion); steady state conditions were reached within 3 seconds.

Obtained results are shown in Figure 3 (section at probe location), while Figure 4 shows corresponding measured void fraction. As can be observed, void production was qualitatively well predicted but with significant differences in the vapour distribution. In NEPTUNE_CFD simulation the highest vapour concentration was found near the heated rods while experimentally it was observed in the central part of sub-channels; moreover, predicted vapour concentration did not reach the maximum experimental values (> 60%), even if the section averaged void fraction is slightly overestimated (~ 30% with respect to the measured 25%).

Same input was run adopting a refined grid with 578565 hexahedral elements (1935×299) instead of 172732 (868×199). As a result, void production was consistent with previous calculation but some differences can be found in the local distribution through the section (Figure 5): mesh refinement evidenced that the adjacent wall area is not the region at highest vapour concentration, and smoothened differences between sub-channels at higher and lower vapour concentration. Section averaged void fraction diminished to ~ 28%. Calculations run adopting two further refined grids did not converged, suggesting a deeper investigation on compatibility between meshing characteristics and boiling model.

From sub-channel study it was also observed a strong sensitivity to the imposed (constant) value of mean bubble diameter. The reference input was then adopted including a balance equation for the interfacial area concentration, in order to predict the local two-phase geometrical parameters of the boiling flow; the "Wei Yao" model for volumetric interfacial area prediction was selected, which is exhaustively described by Yao et al. (2004). As shown in Figure 6, the predicted void distribution did not change significantly, even if vapour seems to be less uniformly spread among the sub-channels; section averaged void fraction reduced to ~ 26%.

Adopting modified ("GRE") models for the interface heat transfer and wall boiling (Morel, 2005) produced a slightly different vapour distribution, more uniform in the region at higher steam concentration (Figure 7) and with smaller overall void fraction (~ 25.5%).

Adopting the same models but neglecting the vapour turbulence resulted in the void distribution as expected from sub-channel analysis (Figure 8), with nearly all vapour confined in the regions adjacent to heated rods walls; the section averaged void fraction also changed (~ 27%).

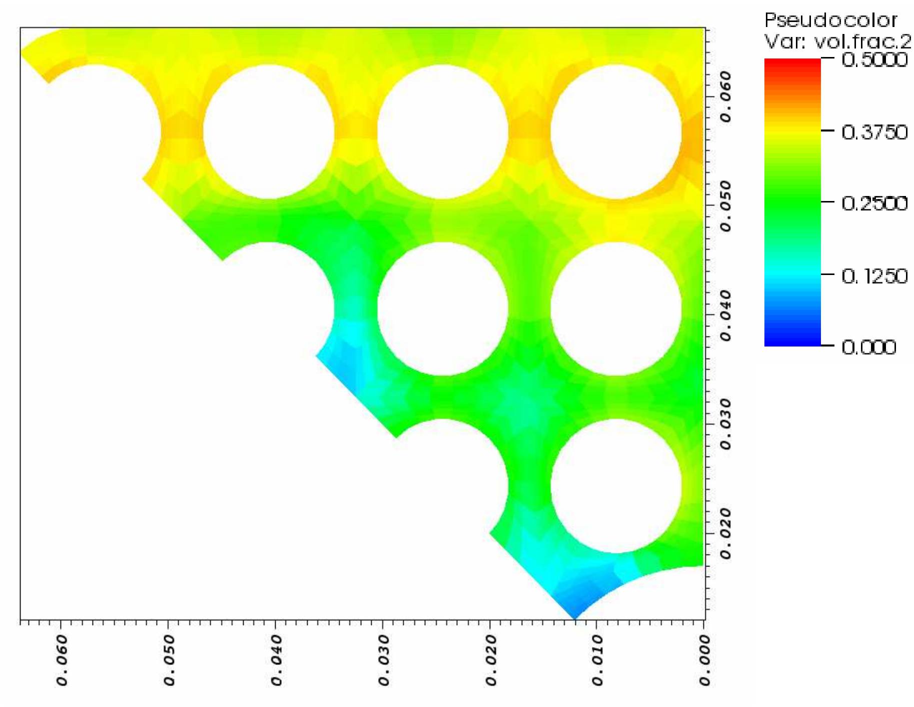

Fig. 3: Predicted void fraction - reference calculation.

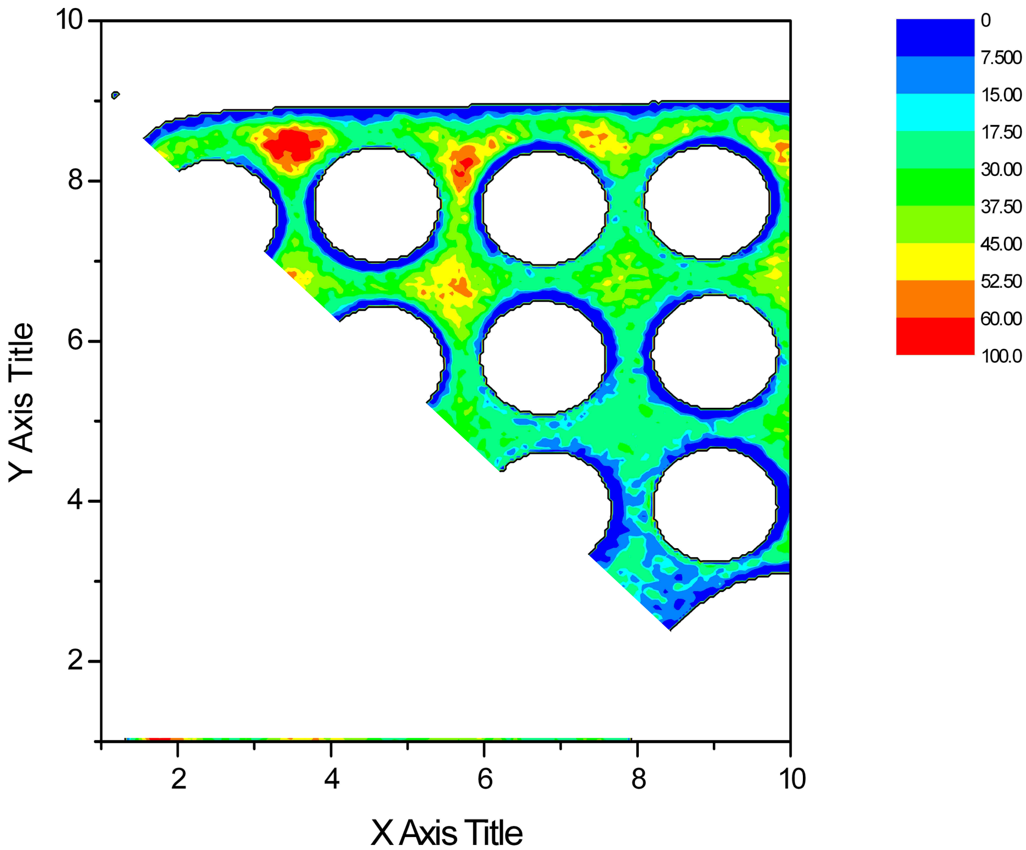

Fig. 4: Experimental data on void fraction distribution (TEST 4101-53).

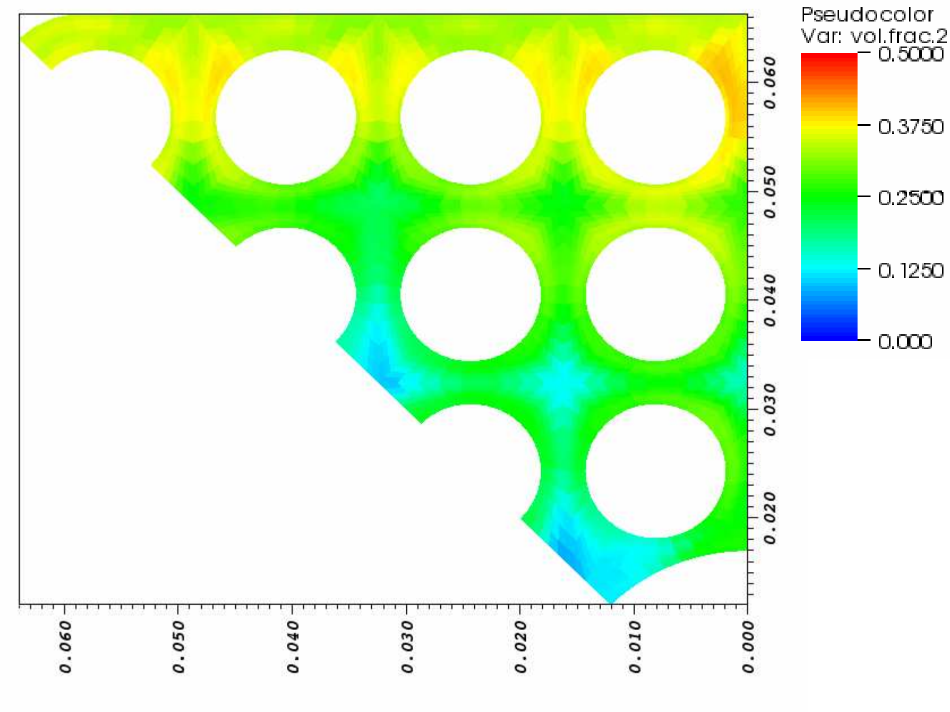

Fig. 5: Predicted void fraction – sensitivity to grid refinement.

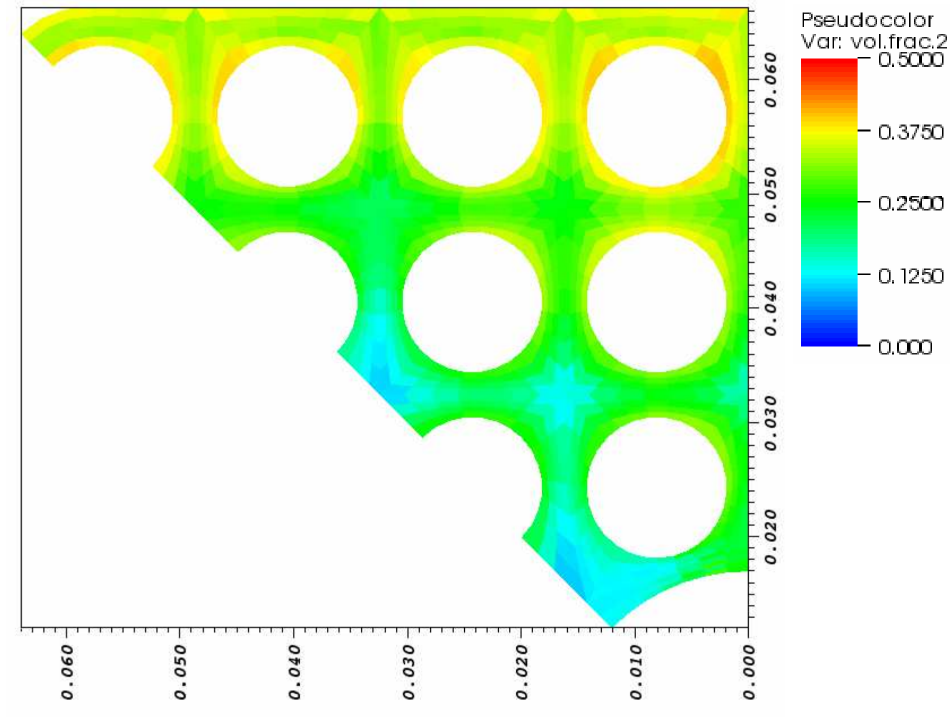

Fig. 6: Predicted void fraction modelling the interfacial area concentration.

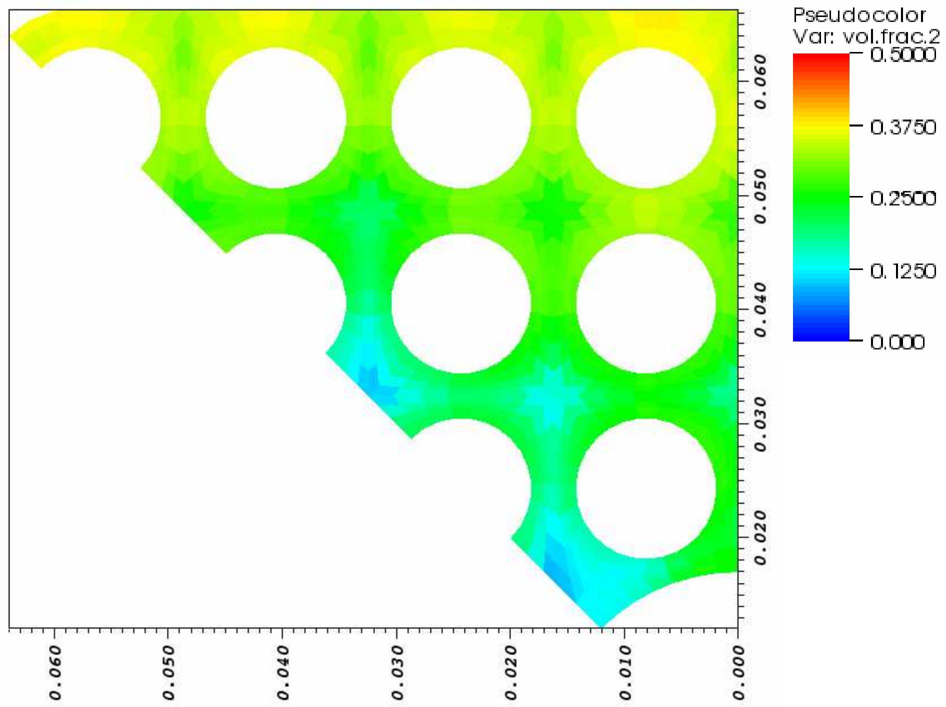

Fig. 7: Predicted void fraction – "GRE" models for interface heat transfer and wall boiling.

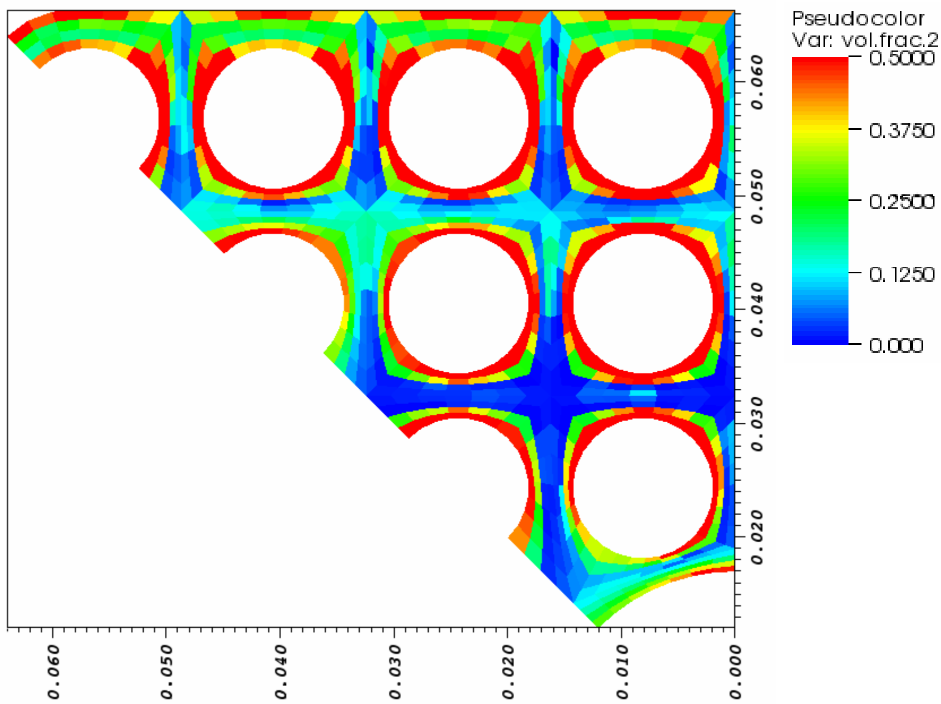

Fig. 8: Predicted void fraction – "GRE" models without vapour turbulence modelling.

5. ANSYS CFX SIMULATIONS

The same computational domain (1/8 of the Fuel Assembly) was simulated with ANSYS CFX; calculations were run considering three successively refined grids, namely a1 (202400 nodes), a2 (402600 nodes), a3 (741000 nodes) and a grid with meshing quality is improved, a4 (637065 nodes). The Eulerian two-fluid model was applied, with the following assumptions:

- k-epsilon turbulence model for liquid phase
- Vapour contribution to turbulence taken into account by means of Sato model
- Interface interaction forces contribution accounted for (Ishii-Zuber model for Drag; Tomiyama model for Lift; Antal model for Lubrication and Favre-Averaged model for Turbulent Dispersion
- Fully developed inlet profiles for velocity and turbulent quantities

The convergence of calculations was achieved in accomplishment with Best Practice Guidelines (NEA/CSNI, 2007). In particular, the RMS residuals were decreased by at least ten orders of magnitude, the imbalances were also reduced to very small amounts, and the monitor points were completely stabilized, as shown in Figure 9. Obtained results are shown in Figure 10 and Figure 11.

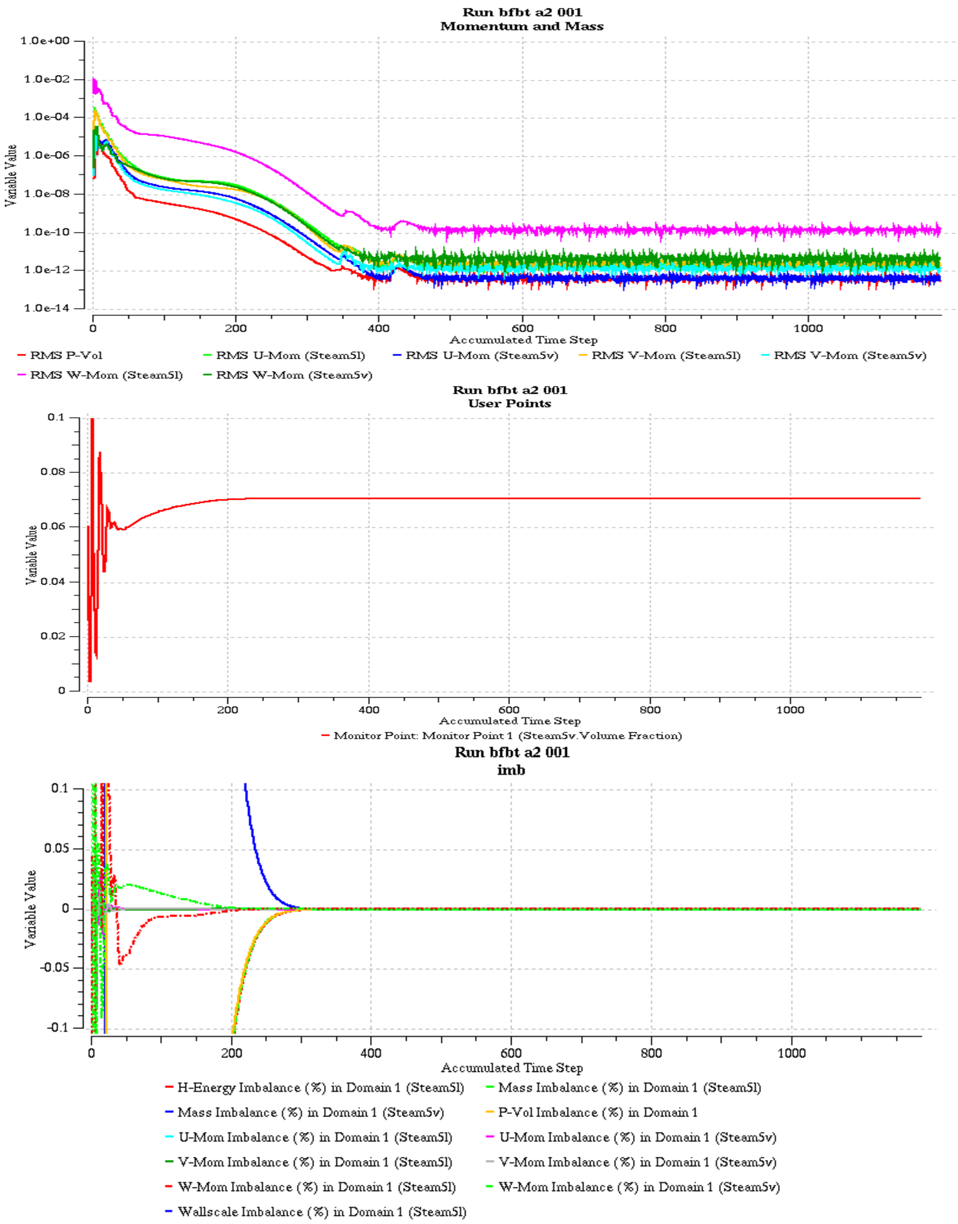

Fig. 9: Monitoring of CFX calculations convergence.

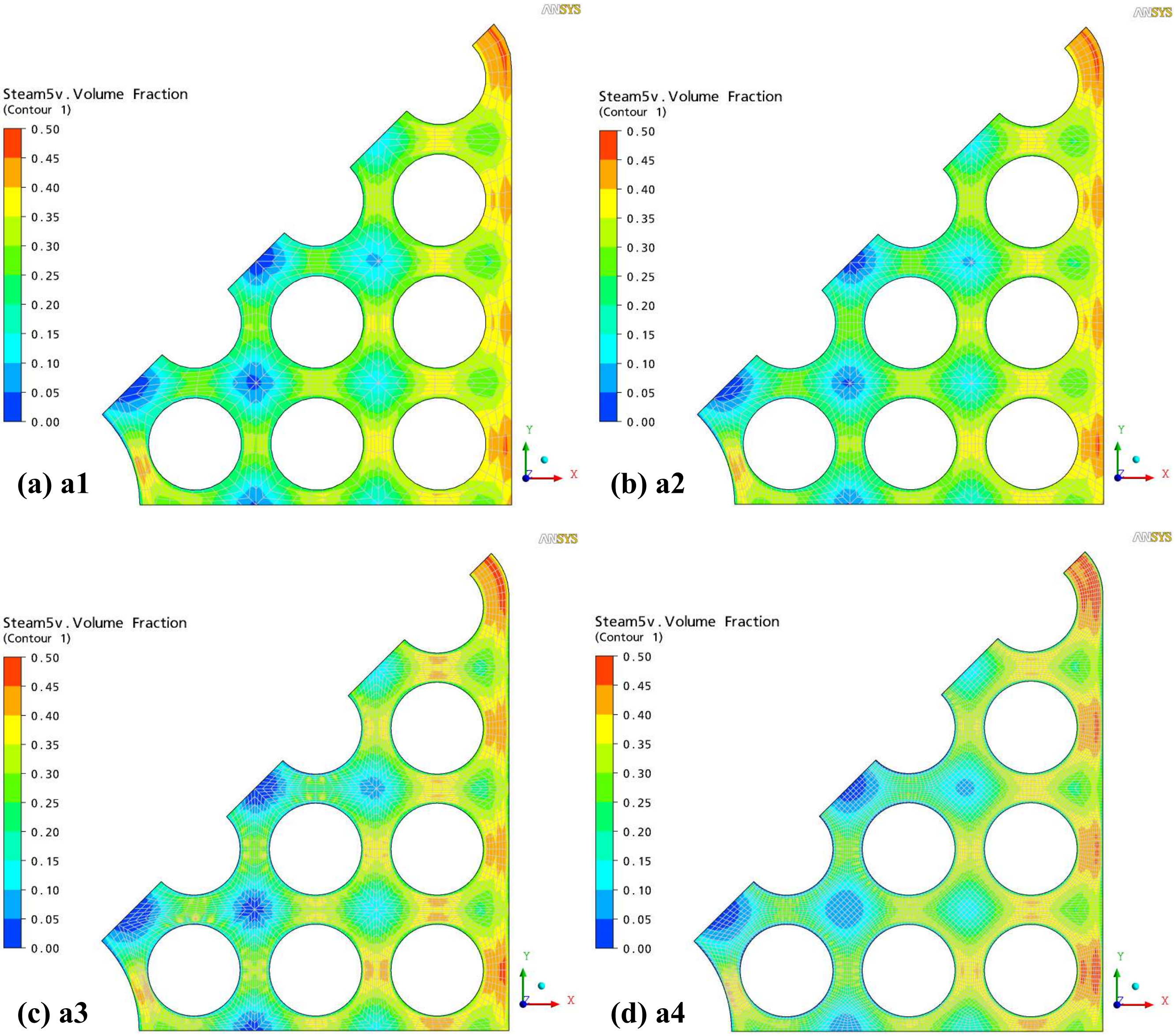

Fig. 10: CFX void fraction predictions – Sensitivity to grid refinement:
(a) grid a1; (b) grid a2; (c) grid a3; (d) grid a4.

Grid	Void	error %
a1	0.268	7.2
a2	0.270	8.0
a3	0.268	7.2
a4	0.264	5.6

Table 2: Calculated cross-section averaged void fraction at probe location

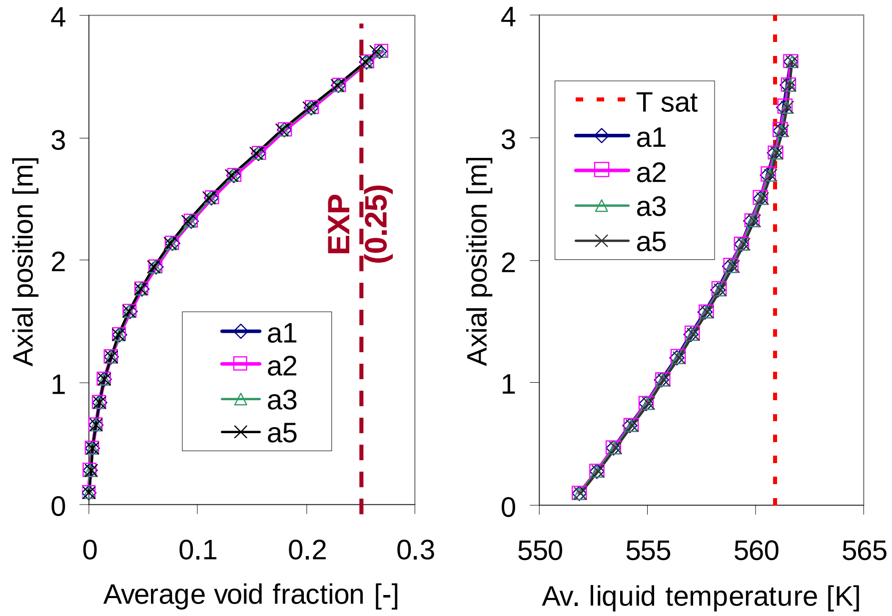

Fig. 11: CFX predictions for cross-section averaged void fraction and liquid temperature (axial profiles).

As can be observed from the previous Figures and from Table 2, CFX predictions were in quite good agreement with experimental data and some improvements were achieved with grid refinement, even if convergence on spatial discretization was not reached. Calculated void distribution was not matched, but was consistent with NEPTUNE_CFD predictions: vapour concentration is higher in the near wall regions and lower in sub-channels centre.

6. CONCLUSIONS

Computational Fluid Dynamic simulations of BFBT experimental tests (BWR Full-size Fine-mesh Bundle Tests) were presented. The aim was to evaluate NEPTUNE_CFD V1.0.6 and ANSYS CFX 11.0 capabilities for predicting boiling bubbly flow in a real fuel assembly.

Due to the complexity of the problem and to the high calculation time required for each simulation, a simplified problem was considered for NEPTUNE_CFD input settings and first attempt calculations. Attention was then focused on a single, generic sub-channel; thanks to its geometrical symmetry, the considered fluid domain was furthermore reduced to 1/8 of the sub-channel. Calculations were run adopting some standard models available in NEPTUNE_CFD and then performing sensitivity studies on fundamental problem parameters, such as the mean bubble diameter, the vapour turbulence modelling, the effect of non-drag forces contribution and grid refinement. As a result, predicted void fraction was found to be very sensitive to all of selected parameters; in particular, the vapour turbulence modelling and non-drag forces contribution strongly influenced the void distribution through the channel section, while the overall vapour production was more sensible to the value of mean bubble diameter. These findings were considered in the set-up of NEPTUNE_CFD fuel assembly simulations. Starting from the problem symmetry, only 1/8 of the assembly was modelled. Void production was qualitatively well predicted but significant differences were observed in the vapour distribution through the section, since the highest void concentration was found near heated rods wall and not within sub-channels central region. Sensitivity analysis on modelling parameters confirmed the importance of representing vapour turbulence; moreover, it suggested the use of a volumetric balance equation for the interfacial area concentration, in order to predict the local two-phase geometrical parameters of the boiling flow.

CFX predictions were in quite good agreement with experimental data as well; some improvements were achieved with grid refinement, even if convergence on spatial discretization was not reached. Calculated void distribution did not matched measured data, but was consistent with that found by

NEPTUNE_CFD simulations; this may suggest the relevance of secondary flows induced by spacer grids, which contribute to flow mixing.
Although qualitative conclusions should not be affected by using a converged meshing, for both codes void distribution resulted to be significantly influenced by grid refinement, so that a further mesh sensitivity analysis is recommended besides modelling spacer grids into the computational domain. Future studies should also address the effect of non-drag forces modelling.

REFERENCES

A. Burns, Y. Egorov, F. Menter, "Experimental Implementation of the RPI Wall Boiling Model in CFX-11", *ANSYS CFX Germany*, April, 2004.

A. Guelfi, M. Boucker, J.M. Hérard, P. Péturaud, D. Bestion, P. Boudier, P. Fillion, M. Grandotto, E. Hervieu, "A New Multi-Scale Platform for Advanced Nuclear Thermal-Hydraulics Status and Prospects of the NEPTUNE Project", *Proc. of The 11th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11)*, Avignon, France, October 2-6, 2005.

A. Inoue, T. Kurosu, T. Aoki, M. Yagi, T. Mitsutake, S. Morooka, "Void Fraction Distribution in BWR Fuel Assembly and Evaluation of Subchannel Code", *Journal of Nuclear Science and Technology*, Vol. 32 No. 7, 629-640, 1995.

N. Kurul and M. Z. Podowski, "On the modelling of multidimensional effects in boiling channels", *ANS Proceedings of 27th National Heat Transfer Conference* Minneapolis, MN, 1991.

N. Méchitoua, M. Boucker, J. Laviéville, J. Hérard, S. Pigny, and G. Serre, "An unstructured finite volume solver for two-phase water-vapour flows based on an elliptic oriented fractional step method", *Proc. of The 10th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-10)*, Seoul, Korea, October 5-9 (2003).

C. Morel, S. Mimouni, J. M. Laviéville, M. Bouker, "R113 Boiling Bubbly flow in an annular geometry simulated with the NEPTUNE Code", *Proc. of The 11th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11)*, Avignon, France, October 2-6, 2005.

P. Muehlbauer et al., "Review of two-phase modelling capabilities of CFD computer codes and feasibility of transient simulations", *EVOL-ECORA-D04,* 2004.

NUCLEAR ENERGY AGENCY, Committee on the Safety of Nuclear Installations, *Best Practice Guidelines for the use of CFD in Nuclear reactor Safety Applications*, 2007.

OECD-NEA web site, "OECD-NEA/US-NRC Benchmark based on NUPEC BWR Full-size Fine-mesh Bundle Tests (BFBT)", http://www.nea.fr/html/science/egrsltb/BFBT/, 2007.

E. Sartori, L. E. Hochreiter, K. Ivanov, H. Utsuno, **"**The OECD/NRC BWR Full-Size Fine-Mesh Bundle Tests Benchmark (BFBT) - General Description", *The 6th International Conference on Nuclear Thermal Hydraulics, Operations and Safety* (NUTHOS-6), Nara, Japan, 2004.

H. Utsuno, N. Ishida, Y. Masuhara, F. Kasahara, "Assessment of Boiling Transition Analysis Code Against Data from NUPEC BWR FULL-SIZE FINE-MESH BUNDLE TESTS", *The 6th International Conference on Nuclear Thermal Hydraulics, Operations and Safety* (NUTHOS-6), Nara, Japan, 2004.

W. Yao, C. Morel, "Volumetric interfacial area prediction in upward bubbly two-phase flow", *International Journal of Heat and Mass Transfer*, Vol.47, 307-328, 2004.