2D SIMULATION OF TWO-PHASE FLOW ACROSS A TUBE BUNDLE WITH NEPTUNE_CFD CODE

D. Soussan, S. Pascal Ribot, M. Grandotto

CEA, DEN, Département d'Etudes des Réacteurs, CEA Cadarache, 13108 St Paul-lez-Durance, France

1 Abstract

Nowadays, the life time extension of a Pressurized Water Reactor (PWR) steam generator (SG) is a world-wide concern, jeopardized by several factors, among which tube wear due to flow induced vibrations. Therefore, increasing accuracy in understanding and predicting two-phase flows across the tube bundle is required. Nonetheless, due to the device complexity (around 6000 tubes), industrial computational tools are based on porous medium concept, which means solid obstacles are homogenized inside a homogenization cell. Consequently, studies describe the flow in the subchannel scale, and predictive models are either founded on two-fluid approach (balance equations for both phase) or homogeneous model (mixture balance equations). However, current trend turns towards CFD tools in open medium to go beyond the limits of the component scale for a finer description of the flow. Hence we have chosen as a primary application to study a bubbly two-phase mixture upflowing across a square rod bundle (1.44 pitch to diameter, non boiling). Comparisons between experiment and simulation are based on void fraction, bubble velocity and bubble mean diameter. Experimentally, void fraction and interfacial velocity inside a central subchannel are measured by bi-optical probes. Numerical simulation is performed with the NEPTUNE_CFD module for open medium. It offers advanced physical models (two-fluid model in the present case combined with interfacial area transport and turbulence). Then, in order to assess the information feedback from CFD analysis (at local scale) to industrial softwares (at component scale), an analysis of predicted kinematic disequilibrium at both scales, local scale (computed with two-fluid model in open medium) and subchannel scale (computed with homogeneous model in porous medium) is proposed.

Nomenclature

ia : Abbreviation for interfacial area
C_D : Drag coefficient
D_b : Bubble diameter (m)
g : Gravitational acceleration (= 9.81 m/s^2)
G : Mass flux (kg/m^2/s)
SG : Steam generator
J : Velocity of the center of volume of the mixture (m/s)
La : Laplace length (m)
P : Pressure (Pa)
S Mock-up section (m^2)
v : Velocity (m/s)
X : Quality

Greek characters :

α :	Void fraction (time fraction of gas phase)
μ :	Dynamic viscosity (kg/m.s)
ρ :	Density (kg/m^3)
σ :	Surface tension (N/m)

Subscripts :

G :	Gas
L :	Liquid
r :	Relative

Formalisms :

« » :	Phase fraction weighted spatial average, «X» = $\frac{<\alpha X>}{<\alpha>}$
< >:	Spatial average (in the homogenized volume, porous medium concept)
v :	Local velocity (CFD average meaning)
V:	Spatial average velocity, $V = << v >>$ (porous code average meaning)

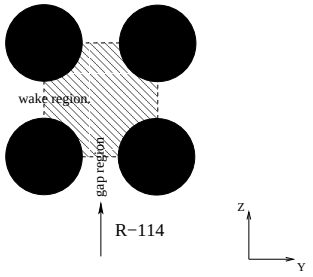

Figure 1: Region definition in the subchannel

2 Introduction

Nowadays, the life time extension of steam generators (SG) in a Pressurized Water Reactor (PWR) is a world-wide concern, jeopardized by several factors, among which tube wear due to flow induced vibrations. As a result, the prediction of two-phase flow across a rod bundle, and its consequences on mechanical structure behaviour, are of major concern for nuclear power plant safety and dependability. In this view, the improvement of the simulation tools will lead to a reduction of the uncertainties linked to safety margins.
Historically, and due to the device complexity (around 6000 tubes are involved in a steam generator), industrial computation tools were based on porous medium concept, which means solid obstacles are homogenized inside a homogenization cell. Therefore, relevant data are spatially averaged within the subchannel, bound by 4 quarters of tube, in a subchannel analysis. This is the case of so-called component codes such as Genepi [Obry et al., 1990], or Thyc [David, 1999]. But the current trend in the field of flow induced vibrations is to catch the physical phenomena at a scale smaller than the subchannel in view of better understanding and predicting the fluid-elastic coupling [Pascal-Ribot and Blanchet, 2007, 2008]. Thus, simulation studies turn towards the use of smaller scale models and related codes to get more local and more accurate information on flow behaviour. This has led to a new field of investigation, namely the multi-scale approach seen as an additional way to the experimental way. For instance, Jamet et al. [2008] presented the use of Direct Numerical Simulation for larger scale models in the context of Departure of Nucleate Boiling mod-

elling. Our purpose here is to establish the preliminary bases for the use of Computational Multifluid Flow Dynamics (CMFD) in open medium to improve the modelling at porous scale, based on subchannel analysis. In this view, we propose in the present paper to simulate an upward two-phase flow across a horizontal rod bundle with the CFD scale module for open medium of NEPTUNE platform project [Guelfi et al., 2006]. Experimental data are issued from Minnie 2 cross-flow program [Haquet and Gouirand, 1995], using R-114 refrigerant fluid to simulate steam generator flow conditions. Following a brief description of Minnie 2 test section and related program, we present the results obtained for a 20 % void fraction test. Comparisons between computations and experiment are discussed, the challenge being to reproduce the measured void fraction distribution inside the rod bundle.
Then, to go further, the kinematic disequilibrium between phases, which is a closure law of crucial importance in industrial component codes, is used to illustrate the multi-scale approach. As a matter of fact, in vertical ducts or vertical arrays of tube, the well-known one dimensional drift flux model [Ishii and Zuber, 1979] usually provides an area averaged velocity difference between phases. Besides, in much more complex configuration such as cross-flows, the drift-flux model is not so appropriate due to recirculation occurences and 2D flow development. Nevertheless, its use is maintained with sometimes multi-dimensional extension attempts [François, 2001]. Moreover, although several experimental studies under cross-flow conditions are reported in the literature [Serizawa et al., 1997], [Suzuta et al., 1999], [Noghrekhar et al., 1999], [Xu et al., 1998], [Aprin et al., 2007], they are mostly reduced to void fraction measurement and are mostly related to air-water mixture. Thus, there are too few information to develop a specific cross-flow model and to extrapolate it to steam-water flow under high pressure. This gives an opportunity to multi-scale approach in the aim of providing missing information. Thus, the last part of the analysis endeavours to link the relative velocity of gas and liquid, as predicted at porous scale by the Drift flux model, with the relative velocity assessed from the computed local data fields of both gas and liquid velocities.

3 Simulation of a vertical R-114 two-phase flow across a horizontal tube bundle

3.1 Experimental setup and test operating condition

Basically, the Minnie 2 cross-flow test section consists of a rectangular channel (0.0975 x 0.08 m^2 section) with a square pitch horizontal tube bundle (30 rows of 5 tubes: 4 tubes plus 2 half tubes on the wall per row), see Fig.2. The mixture, liquid-vapor R-114 under 9 bars, is produced upstream from the test section through a water-Freon boiler, then travels vertically upward in the rod bundle. Initially, R-114 has been selected to simulate water steam under the nominal conditions of the secondary flow inside a steam generator: in particular, the liquid density to vapor density ratio is respected, and the Weber number, defined as $\frac{g v_r D_b}{\sigma}$, with pool boiling approximate diameter and Ishii relative velocity, are comparable, see Table 1.
As explained before, the present study is not devoted to validate a CFD tool, but is aimed at linking two predictive scales around a common physical phenomenon which is the kinematic disequilibrium between phases. To start with, it is necessary to study a flow whose features are rather well known and quite well modeled at both scales. Consequently, the study is focused on a 20% void fraction test, where the regime according to Ulbrich and Mewes [1995] is assumed to be bubbly. In the central subchannel (located in heart of the bundle), pressure, temperature and quality are measured. Together with the mass flow rate measured upstream from the test section, these parameters define the operating conditions of the test (see Table 2). Moreover, the central subchannel is scrutinized by a bi-optical probe (BOP) in 149 measurement points, providing the spatial void fraction distribution. In the gap region, see Fig. 3, interfacial velocity issued from BOP signals cross-correlation function is reliably associated to gas velocity for bubbly flows. On the basis of past studies dedicated to work out the BOP device and data processing, uncertainty on local void fraction measurement is around 2.5% and relative uncertainty on bubble velocity measurement is around 7% [Gouirand and Haquet, 1991] & [Soussan et al., 2001]. From the interfacial velocity measurement, the mean

Sauter diameter can be assessed providing the bubble mean size assuming a monodisperse bubbly flow. Over the 149 measurement points, 8 characteristic measurement points are extracted for the comparison between the experiment and the computation : 4 are located in the “gap area”, and 4 in the “wake region”, see Fig. 3. Moreover, 2 additional points located respectively at the inlet (probe n° 1) and at the outlet (probe n°3) of the test section, outside the bundle, for which no experimental measurement has been performed, are analyzed to check the inlet boundary conditions and to follow the axial flow development betwen inlet and outlet.

	Steam-water	R-114
ρ_L (kg/m^3)	736	1267
$\rho_G - \rho_L$ (kg/m^3)	698	1200
D_b (m)	0.003	0.001
σ (N/m)	17 10^{-3}	6^{-3}
La (m)	1.57 10^{-3}	7 10^{-4}
for 20 % void fraction		
C_D	1.45	1.064
v_r en m/s	0.143	0.096
We	2.55	1.71

Table 1: Specific features of steam water under 6 MPa and R-114 under 0.9 MPa

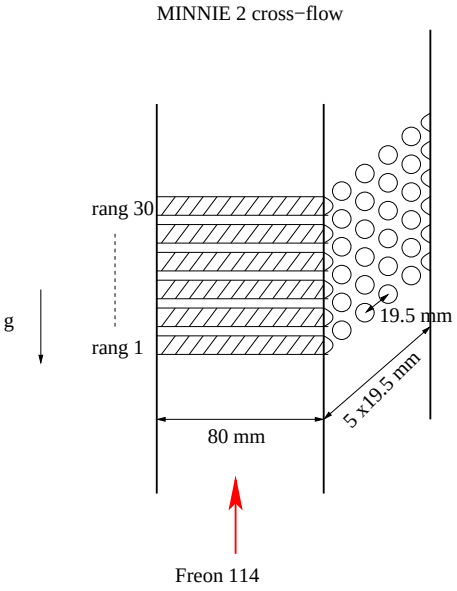

Figure 2: Minnie 2 test section

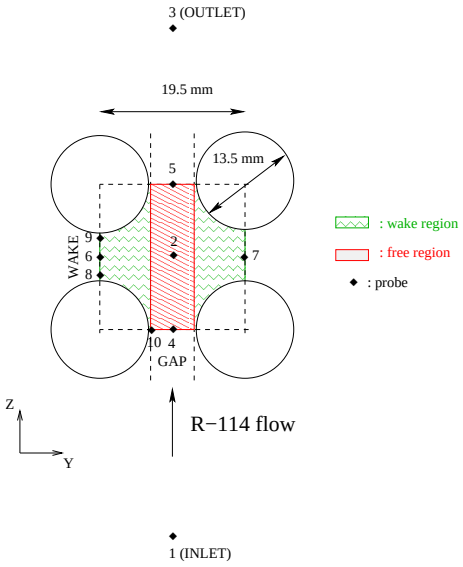

Figure 3: Probe location within the central subchannel

3.2 Test simulation

Computations have been performed with the version 1.0 of NEPTUNE_CFD, a computational fluid dynamics tool for open medium. The average scale (millimeter or less) allows to get a finer description of the flows than component scale, since it provides the volume fraction and velocity distribution for both phasis inside the subshannel. NEPTUNE_CFD is based on a cell centered finite volume method and solves a six equation two-phase flow model [Guelfi et al., 2006]. These equations have been obtained using a Reynolds Averaged Navier-Stokes (RANS) process extended to two-phase flows. The test section is adiabatic. Only the flow

Pressure (Pa)	903400
Saturation temperature (K)	352.078
Mass flow-rate (kg/s)	2.344
Quality	0.0186
Mean void fraction	0.203
Mean Sauter Diameter (mm)	1.16

Table 2: Selected test operating conditions

dynamics under the tube bundle confinement is of interest. Hence, the physical origin of the closure models are twofold: turbulence and mass momentum interfacial transfers. For simplicity, the computed domain has only 1 cell in depth, assuming flow is 2 dimensional. Furthermore, in order to limit the CPU time of the simulation, and on the basis of preliminary calculations, the number of rows has been reduced to 11 rows compared to the actual 30 rows. The investigated cell is located between the 7th and the 8th row (Fig. 6). The computation mesh has 32720 hexahedric cells, to describe an area of 0.0975 x 0.3915 m^2 see Fig. 4. To give an idea of the discretization, the space between two adjacent tubes, of 6 mm width, is described with 12 cells (see Fig. 5).

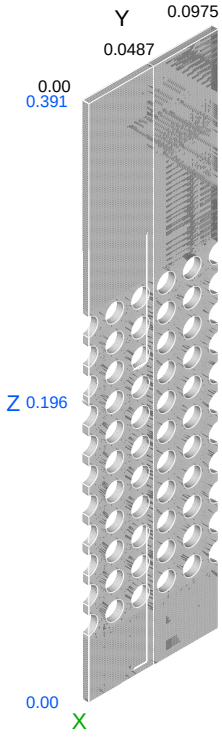

Figure 4: Computation mesh

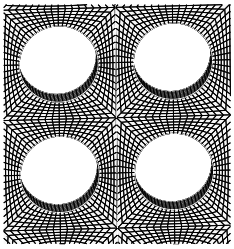

Figure 5: Zoom on the mesh

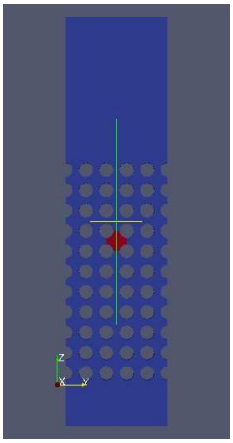

Figure 6: Central subchannel

As for the fluid flow simulation, the liquid phase is the continuous phase. Turbulence is modelled with a k-ε model for the liquid phase that imposes turbulence on the gas phase (one-way coupling). Momentum interfacial transfers are based on the Ishii drag coefficient, constant coefficient for the added mass (= 0.50), lift (= 0.29) and turbulent dispersion (= 1.). On the wall, a logarithmic wall law is applied to the liquid

phase whereas a condition of vanishing velocity is used for the gas. Computations have systematically been run over 5 seconds (physical time). The boundary conditions are reached in less than 1 second, thus resulting data have been time averaged over the last 4 seconds and compared to experimental data. However, one can notice after 1 second that the histograms inside the bundle (probe n° 2) exhibit high fluctuations of both void fraction and gas velocity, see Fig. 7.

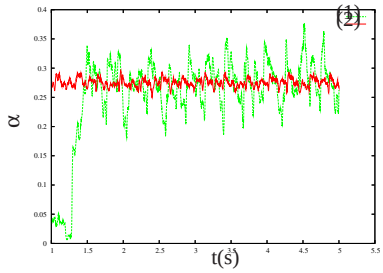

Figure 7: Time evolution of local void fraction at probe 2

The next section presents the result of two runs :

run # 1: with bubble diameter imposed (D_b = 1.16 mm)

run # 2: with the Wei & Yao model relative to the interfacial area transport equation

3.3 Computation results

First, results are analyzed from a "CFD" point of view, that is to say at local scale. Conventionnaly, in all figures, experimental data are reported in blue, data from run # 1 (set diameter) are in green, and data from run # 2 (interfacial area transport) are in red. Fig. 8 compares computed local void fraction to experimental measurements for all probe locations. At first glance, except for probes n° 6, 7, 9, run # 1 with set bubble diameter is closer to experiment. The Wei & Yao model leads to globally overestimate the void fraction. This is the consequence of underpredicting the bubble diameter (see Fig. 9), thus drag coefficient is underestimated (the relative velocity is lower for run # 2 than for run # 1, see Fig. 10), thus gas velocity is underpredicted (Fig. 11), which is consistent with overpredicted void fraction.
Hence, run # 1 is a posteriori the reference computation for the following. Let us examine it in details. Experimentally, in the spanwise direction (probes n° 6-2-7), a slight U-profile is observed, with minimum void fraction at the center (probe # 2) and maximum behind the rods (probes n° 6-7) [Haquet and Gouirand, 1995]. This tendency has been also observed by Ueno et al. [1997] who studied HCFC-123 two-phase flow across tube bundles. Serizawa et al. [1997] in their air-water experiments did not observe exactely this profile. For this latter, the profile in the spanwise direction varies significantly depending on the location within the wake, that is the Z-elevation here between 2 rods. Indeed, in the wake region (probes n° 8-6-9), a void fraction profile increasing with Z is observed, bubbles being entrapped in the vortices behind the rod, the space just behind the tube being quite poor in bubbles. This profile is slightly reproduced by run # 1, but not so well. Following the analyses of Serizawa et al. [1997], the bubble diameter is likely overestimate in this region (small bubbles are more easily trapped in vortices than bigger ones).
The highest discrepancies between computed data and experimental measurements are observed near the tube walls (probes # 8-9-10). The purpose of this study is not to assess the NEPTUNE_CFD code for saturated boiling flows in rod bundle geometry, since experimental data are clearly not sufficient (too few data on velocity). However, the physical models are failing in accurately predicting the gas distribution, specifically in the wake region and near the tube wall. Many physical phenomena are obviously implied: break-up and fragmentation processes in confined medium with free path reduction at each row of tubes, two-phase

turbulence in bundle geometry, influence of bubble size distribution.

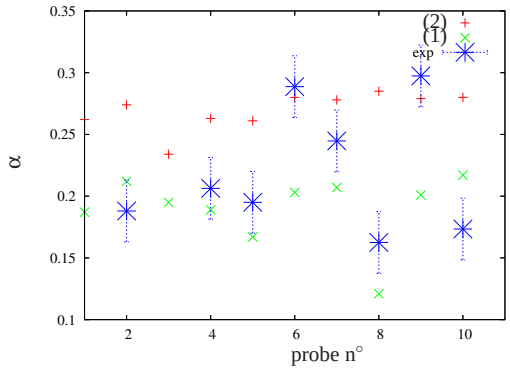

Figure 8: Time averaged local void fraction

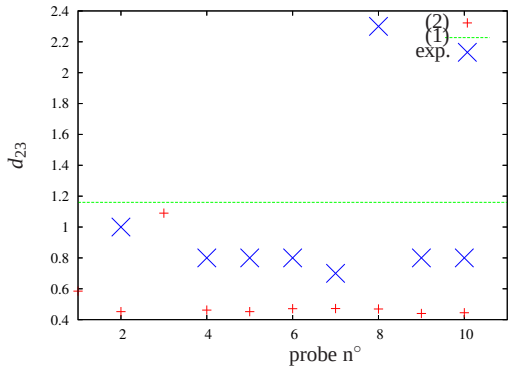

Figure 9: Mean Sauter diameter (mm)

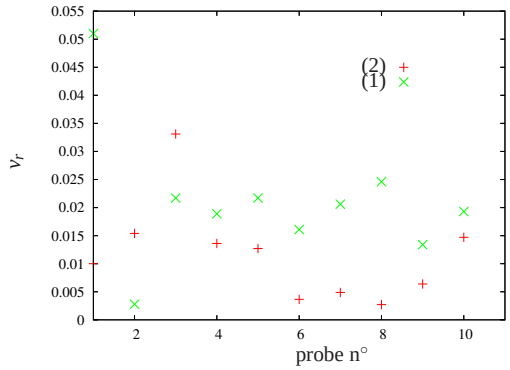

Figure 10: Time averaged axial relative velocity

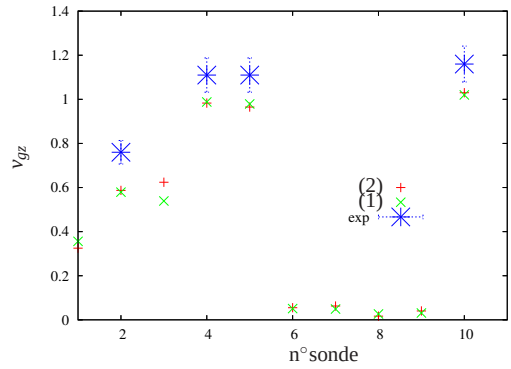

Figure 11: Time averaged axial gas velocity

The next section analyses the simulation runs at a larger scale, the porous scale. Relevant data are no more local but are averaged within the periodic cell.

3.4 Analysis through the drift flux model

Refering to the fundamental paper of Zuber and Findlay [1965] relative to the drift flux model to describe the relative velocity between phases in a vertical duct, we simply recall the main definitions:

- $<\alpha>$ represents the volume averaged void fraction
- J corresponds to the volume averaged velocity of the mixture in axial direction: $J =< j >=< \alpha v_G + (1-\alpha)v_L >$
- $V_{GorL} =<< v_{GorL} >>$ are the phase fraction weighted volume averaged velocity
- V_r stands for the relative velocity issued from previous velocities: $V_r = V_G - V_L$
- $C_0 = \frac{<\alpha j>}{<\alpha><j>}$ is the distribution parameter designed to model the wall effects on phasis distribution within the representative volume
- V_{GJ} is the drift velocity

Table 3 gathers all these values issued either from experiment or computations. We notice that data issued straight forward from experimental measurements (J, $<\alpha>$, V_G) are satisfactory computed. CFD computations allow to obtain the mean gas velocity $<<v_G>>$ within 7 % at porous scale. Besides, high discrepancies are noticed on drift flux parameters, which enhances the differences on resulting relative velocity. The drift velocity V_{GJ} in particular varies significantly between computations first. Those results have to be considered very cautiously since first, the gas velocity has not been reliably measured all over the central subchannel. Moreover, the physical modelling at porous scale is based on the assumption that flow characteristics vary weakly between two nearby subchannels, thus considering the flow in only one direction (Z direction, axial development). This assumption has to be checked all over the computed domain, which means the volume averaged data have to be computed everywhere, by means of a continuous volume average processing. Additional computations based on a volume average over the entire 5 subchannels in the spanwise direction exhibit a non flat profile for run # 1. This leads us to suggest to control accurately the experimental profile regularity before extrapolating a physical model. Currently, the correlations issued from experimental analysis have not been reproduced by CFD computations.

	RUN #1	RUN #2	exp.
$<\alpha>$	0.199	0.269	0.20
J	0.500	0.498	0.51
C_0	0.996	1.013	0.92
V_{GJ}	0.010	0.006	0.085
$<<v_G>>$	0.508	0.510	0.55
V_r	0.024	.017	0.058*

Table 3: Relevant data at porous scale (* not measured but deduced from the drift flux model)

4 Conclusion

A bubbly two-phase flow, 20 % void fraction R-114 mixture, circulating upwardly across an in-line rod bundle has been simulated with the CFD module for open medium of the NEPTUNE platform project. Due to the lack of accurate experimental data, issued from Minnie 2 cross-flow program, the present study is not used to validate the CFD software. Rather, attention is directed towards analysing kinematic disequilibrium between gas and liquid at two scales: local scale and porous scale. More precisely, we attempt to draw the connection between the two-fluid model at local scale and the mixture model at porous scale. In the former, the relative motion of gas with respect to liquid results from interfacial mass momentum transfers, mainly due to drag force, whereas the latter predicts directly the relative velocity through a closure law based on the drift flux model.

All potentials of NEPTUNE_CFD have not been used. Computations have been focused on the influence of bubble diameter, either set or assessed via the interfacial area transport equation. This exploratory study gives the following results:

- At local scale, computations are in satisfactory agreement with experimental data. The analysis is limited by the lack of experimental information on velocity fields.
- As far as the multiscale approach on kinematic disequilibrium in concerned, high discrepancies are observed between the correlation at porous scale issued from experimental data interpretation and coefficients predicted by computations. It is rather difficult to discreminate the uncertainty linked to the lack of experimental measurements (liquid and gas velocity field) from the numerical issues linked to the change of scaling. For this latter, a way of improvement consists in applying a continuous averaging process all over the computation domain.

- Finally, as the pursued target is to use local computations together with experimental programs to set up physical models at porous scale, CFD tools have to be validated under steam generator conditions. The present study suggests to explore the break-up and coalescence mechanisms which govern the bubble size distribution, as well as the influence of the polydisperse flow features on phasis distribution within the rod bundle. Also, the $R_{ij} - \varepsilon$ turbulence model should be more appropriate to describe the recirculation behind the tubes.

- Yet, this study demonstrates how CFD in open medium can be used to support future experimental programs.

5 Acknowledgments

This work have been achieved in the framework of the NEPTUNE project, financially supported by CEA (Commissariat à l'Energie Atomique), EDF, IRSN (Institut de Radioprotection et de Sûreté Nucléaire) and AREVA-NP.

References

L. Aprin, P. Mercier, and L. Tadrist. Experimental analysis of void fractions measurements for boiling hydrocarbons in complex geometry. *Int. J. Multiphase Flow*, **33**:371–393, 2007.

F. David. Three dimensional thermal-hydraulic simulation in steam generators with THYC exchangers code Application to the UTSG model 73/19. In *NURETH-9*, San Francisco, California, USA, 3-8 October 1999. NURETH-9.

F. François. *Etude des écoulements diphasiques dans les REP embarqués*. PhD thesis, Institut National Polytechnique de Grenoble, Grenoble, 2001. Thèse de Docteur Ingénieur.

J.M. Gouirand and J.F. Haquet. High performances bi-optical probe for two-phase flow measurements. In *Advances in Multiphase Flow*. Elsevier Science B.V., 1991.

A. Guelfi, D. Bestion, M. Boucker, P. Boudier, P. Fillion, M. Grandotto, J.M. Hérard, E. Hervieu, and P. Péturaud. A new software platform for advanced nuclear thermal-hydraulics. *Nuclear Science and Engineering*, 2006.

J.F. Haquet and J.M. Gouirand. Local two-phase flow measurements in a cross-flow steam generator tube bundle geometry: the Minnie II XF Program. In *Advances in Multiphase Flow*. Elsevier Science B.V., 1995.

M. Ishii and N. Zuber. Drag coefficient and relative velocity in bubbly, droplet and particulate flows. *AIChE Journal*, 25:843–855, 1979.

D. Jamet, O. Lebaigue, C. Morel, and B. Arcen. Towards a multi-scale approach of two-phase flow modelling in the context of dnb modelling. In *XCFD4NRS'08*, 2008.

G.R. Noghrekhar, M. Kawaji, and A.M.C. Chan. Investigation of two-phase flow regimes in tube bundles under cross-flow conditions. *Int. J. Multiphase Flow*, **24**:857–874, 1999.

P. Obry, J.L. Cheyssoux, M. Grandotto, J.P. Gaillard, E. de Langre, and M. Bernard. An advanced steam generator design 3D code. In *ASME'90*, Dallas, Texas, USA, 25-30 November 1990. ASME Winter Annual Meeting.

S. Pascal-Ribot and Y. Blanchet. Buffeting Lift Forces and Local Air-Water Flow Aspects around a Rigid Cylinder. *International Journal of Multiphase Flow*, **33**:1237–1254, 2007.

S. Pascal-Ribot and Y. Blanchet. Scaling buffeting forces of air-water flow in a rod bundle. In *Proceedings of FIV 2008*, Prague, 2008. Paper 211.

A. Serizawa, K. Huda, Y. Yamada, and I. Kataoka. Experiment and numerical simulation of bubbly two-phase flow across horizontal and inclined rod bundles. *Nuclear Engineering and Design*, pages 131–146, 1997.

D. Soussan, A. Gontier, and V. Saldo. Local two-phase flow measurement in a oblique tube bundle geometry - the MAXI program. In *ICMF'01*, New Orleans, Louisiana, USA, 27 May - 1 June 2001. ICMF.

T. Suzuta, T. Ueno, Y. Hirao, K. Tomomatsu, K. Kawanishi, and A. Tsuge. Measurement of interfacial velocities in gas-liquid upward two-phase flow across tube bundle. In *7th Int. Conf. on Nuclear Engineering ICONE*, 1999.

T. Ueno, K. Tomomatsu, H. Takamatsu, and N. Nishikawa. Void fraction and interfacial velocity in gas-liquid upward two-phase flow across tube bundles. In *Proceedings of 8th International Topiacal Meeting on Nuclear Reactor Thermal-Hydraulics, Volume 1*, September 30-October 4 1997.

R. Ulbrich and D. Mewes. Experimental studies of gas void fraction for two-phase flow across tube bundle. *Two-Phase Flow Modelling and Experimentation*, 1995.

G.P. Xu, C.P. Tso, and K.W. Tou. Hydrodynamics of two-phase flow in vertical up and down-flow across a horizontal tube bundle. *Int. J. Multiphase Flow*, **224**:1317–1342, 1998.

N. Zuber and J.A. Findlay. Average volumetric concentration in two-phase flow systems. *Trans. ASME Journal of Heat Transfer*, pages 453–468, 1965.