MODELING FREE SURFACE FLOWS RELEVANT TO A PTS SCENARIO: COMPARISON BETWEEN EXPERIMENTAL DATA AND THREE RANS BASED CFD-CODES. COMMENTS ON THE CFD-EXPERIMENT INTEGRATION AND BEST PRACTICE GUIDELINE

Yann Bartosiewicz, J.-M. Seynhaeve,
UCL Université catholique de Louvain, Louvain School of engineering, Mechanical department,
TERM division, Place du Levaant 2, 1348 Louvain la neuve, Belgium, T. +32 10 47 22 06,
yann.bartosiewicz@uclouvain.be

C. Vallée, T. Höhne,
Forschungszentrum Dresden-Rossendorf e.V., Institute of Safety Research,
PO-box 51 01 19, 01314 Dresden, Germany, T. +49 / 351 260-3227, c.vallee@fzd.de

J.-M. Laviéville
EDF - Fluid Dynamics Power Generation Environment,
6 quai Watier, 78401 Chatou, France, jerome-marcel.lavieville@edf.fr

Abstract

This paper presents some results concerning a benchmark for stratified two-phase flows conducted in the frame of the European Platform for NUclear REactor SIMulations (NURESIM). This benchmark relies on the FZD slug flow experiment performed in the Horizontal Air/Water Channel (HAWAC). For this test bench, special experimental arrangements have been taken in order to be able to properly model the boundary and initial conditions with CFD. A picture sequence recorded with a high-speed camera was used as reference for comparison with the simulations. For this benchmark, three different codes have been tried out. CFX was used with a turbulent two-fluid model in which a special turbulence damping function was implemented in the specific dissipation rate of the turbulent kinetic energy. This allowed a good qualitative representation of the slug dynamics, even though quantitative comparison were less relevant because of difficulties in modelling the inlet instabilities. The VOF approach in its laminar and turbulent form was also tried out trough the FLUENT code and was found to be inappropriate for those conditions due to the high velocity slip between phases. Moreover, NEPTUNE_CFD was tested with a new implemented model allowing free surface location and the computation of momentum transfer across this interface. This Large Interface Model (LIM) enables to detect "stratified cells" from the other and hence to apply local closure law. With this model, the results agreed well with experimental data qualitatively and quantitatively. This benchmark experience also allowed to draw basis concerning a best practice guideline in numerical simulation related to those flows in nuclear thermal hydraulics.

1. INTRODUCTION

The European project NURESIM (Nuclear Reactor SIMulation) aims at developing and validating a numerical platform to model complex multiphase flows, relevant to nuclear reactor thermal hydraulics. In this way, the NEPTUNE_CFD code has been developed within the framework of the EDF-CEA co-development project with the support of AREVA-NP and IRSN. One of the issues of NURESIM project is to set up relevant benchmarks in order to assess the code potentials for a variety of situations encountered in nuclear reactors. Among these situations, safety related flows are those that are more complex and of a great interest. One of the possible scenarios (Fig. 1) is cold water Emergency Core Cooling (ECC) into the cold leg during a Lost of Coolant Accident (LOCA). A relevant problem occurring in this situation is the development of wavy stratified flows which can lead to slug generation.

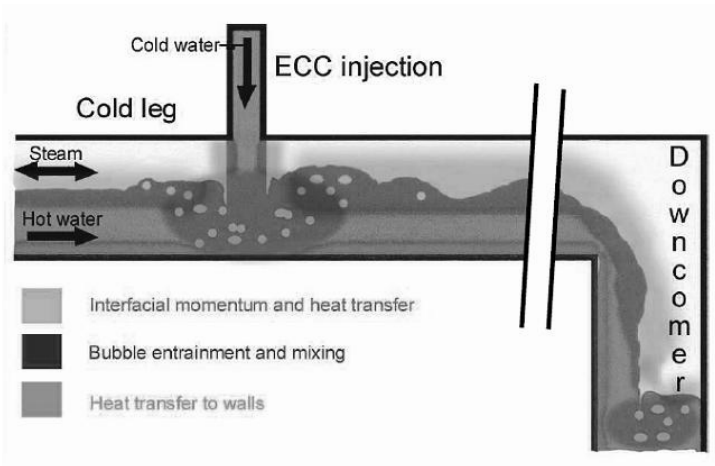


Fig. 1: Different flow situations during an ECC

In order to focus the analysis on momentum transfer across the interface only, it may be relevant to study this kind of flows but without any mass and heat transfer between the phases. In a previous study (Bartosiewicz, 2008), a first benchmark was to set up in order to model instabilities in a stratified flow with small phase slip and low density difference, where surface tension was the key parameter. This test was based on the Thorpe experiment and allowed to implement a consistent surface tension model in a two-fluid approach. This model was satisfactorily validated and compared with available measurements and a VOF method. However, as numerical tools give access to a lot of quantitative values, some new experiments with modern techniques such as PIV or LDV would be necessary to go further in the analysis. Indeed, The Thorpe experiment dates from before the seventies.

The present study aims also at focussing on stratified flow in an horizontal channel but in the case of higher slip between phases. In this case, the surface tension is not the most important phenomenon but the flow dynamics is primarily driven by friction momentum transfer trough the interface.

The proposed benchmark relies on the HAWAC experiment (for Horizontal Air/Water Channel) performed at Forschungszentrum Dresden-Rossendorf (FZD). This experiment deals with wavy stratified flows and slug formation in a rectangular channel. This channel is 8 meters long (height x width = 100 x 30 mm). Special experimental arrangements have been taken in order to properly control the inlet boundary conditions of the water and air flows. The inlet device is designed for a completely separated injection of water and air into the channel which allows to be sure about flow-rates and velocities at the entrance of the channel. Indeed a previous design where both streams were not separated raised some difficulties in terms of CFD-experiment boundary conditions. Four wire mesh filters have been mounted on both sides (water and air) of the inlet device. They aim at providing homogenous velocity profile at the test section inlet. Moreover, the filters produce a pressure drop attenuating the effect of the pressure surge created by slug flow on the fluid supply systems. This set of experiments can thus be considered as reference data for benchmarking of CFD tools.

The slug flow experiment at a superficial water velocity of 1.0 m/s and a superficial air velocity of 5.0 m/s has been chosen as the reference test for simulation with different CFD tools. According to the imposed inlet boundary conditions, 50 % of the inlet cross-section (lower part) is available for the water and 50 % (upper-part) is available for the air. Three different CFD tools have been investigated for this benchmark:

- The ANSYS CFX tool has been investigated at Forschungszentrum Dresden-Rossendorf (FZD). The two-fluid model and the k-ω turbulent model were applied with a specific treatment at the interface.

- The FLUENT tool has also been investigated to simulate this reference test. The VOF approach was chosen in the simulation. Different viscous models and grids were tested until full convergence was reached: laminar, k-ε, k- ω , k-ω SST.

- The NEPTUNE_CFD 1.0.7 (last version) has also been used to simulate this slug flow experiment. In this version of NEPTUNE_CFD, a new model based on Large Interface (LI) recognition were recently developed and implemented in the code (Coste, 2007). It is dedicated to the computation of the interface momentum exchange between the phases for stratified flow with large interface curvature.

2. THE HORIZONTAL AIR/WATER CHANNEL (HAWAC)

Experiments were carried out at the **H**orizontal **A**ir/**Wa**ter **C**hannel (Fig. 2), which is devoted to co-current flow experiments. A special inlet device provides defined inlet boundary conditions by a separate injection of water and air into the test-section. The test-section is 8 m long and its cross-section dimensions are 100 x 30 mm² (height x width). Therefore, the length-to-height ratio *L*/*H* is 80.

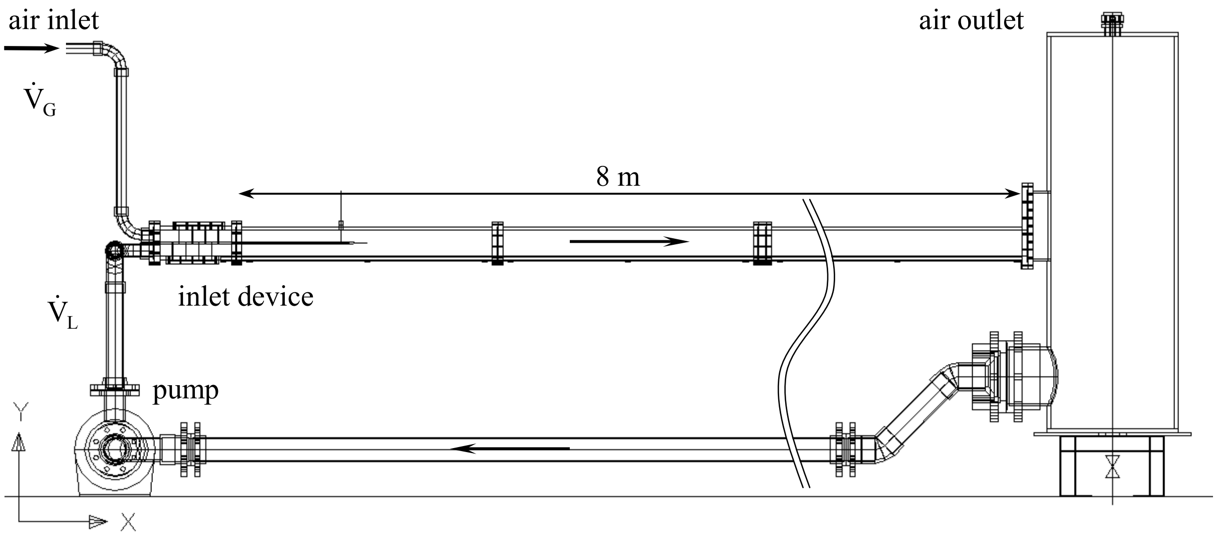


Fig. 2: Schematic view of the horizontal air/water channel (HAWAC)

The inlet device (Fig. 3) is designed for a separate injection of water and air into the channel. The air flows through the upper part and the water through the lower part of this device. Because the inlet geometry produces perturbations in the flow (bends, transition from pipes to rectangular cross-section), 4 wire mesh filters are mounted in each part of the inlet device. The filters are made of stainless steel wires with a diameter of 0.63 mm and have a mesh size of 1.06 mm. They aim at providing homogenous velocity profiles at the test-section inlet. Moreover, the filters produce a pressure drop that attenuate the effect of the pressure surge created by slug flow on the fluid supply systems. Air and water come in contact at the final edge of a 500 mm long blade that divides both phases downstream of the filter segment. The free inlet cross-section for each phase can be controlled by inclining this blade up and down. Both, filters and inclinable blade, provide well-defined inlet boundary conditions for the CFD model and therefore offer very good validation possibilities.

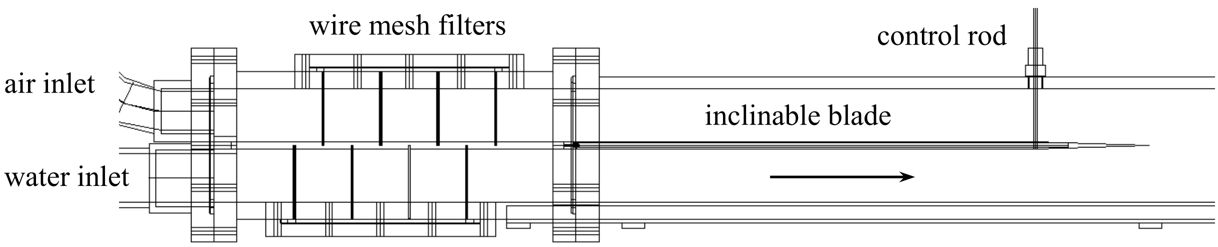


Fig. 3: The inlet device

The water flow rate is measured with a paddle-wheel flow transmitter and is adjusted via the frequency inverter of the pump motor. The air flow rate is measured and controlled with the thermal mass flow meters of the TOPFLOW-facility. These are mounted in parallel in order to ensure a high precision over a large measuring range. The flow rates are measured with an accuracy of ±0,2 l/s for the water and ±1,5% for the air. The maximum superficial velocities achieved in the test-section are 2 m/s for the water and 8 m/s for the air.

3. OPTICAL MEASUREMENT

The presented experiment was focused on the generation of slug flow. It was performed at following superficial velocities in the test-section: J_L = 1.0 m/s for the water flow (i.e. 3.0 L/s) and J_G = 5.0 m/s for the air flow (i.e. 15.0 L/s). Furthermore, the inlet blade was horizontal and therefore the cross-section opening at vertex of the inlet blade was 50 mm for each phase. Due to the rectangular cross-section, the flow can be observed very well from the side of the duct. Optical measurements were performed at the channel inlet with a high-speed video camera at 400 frames per second. As an example, the following picture sequence (Fig. 4) shows the generation of a slug over the first 3.2 m of the test-section (zero mark corresponds to the end of the inlet blade).

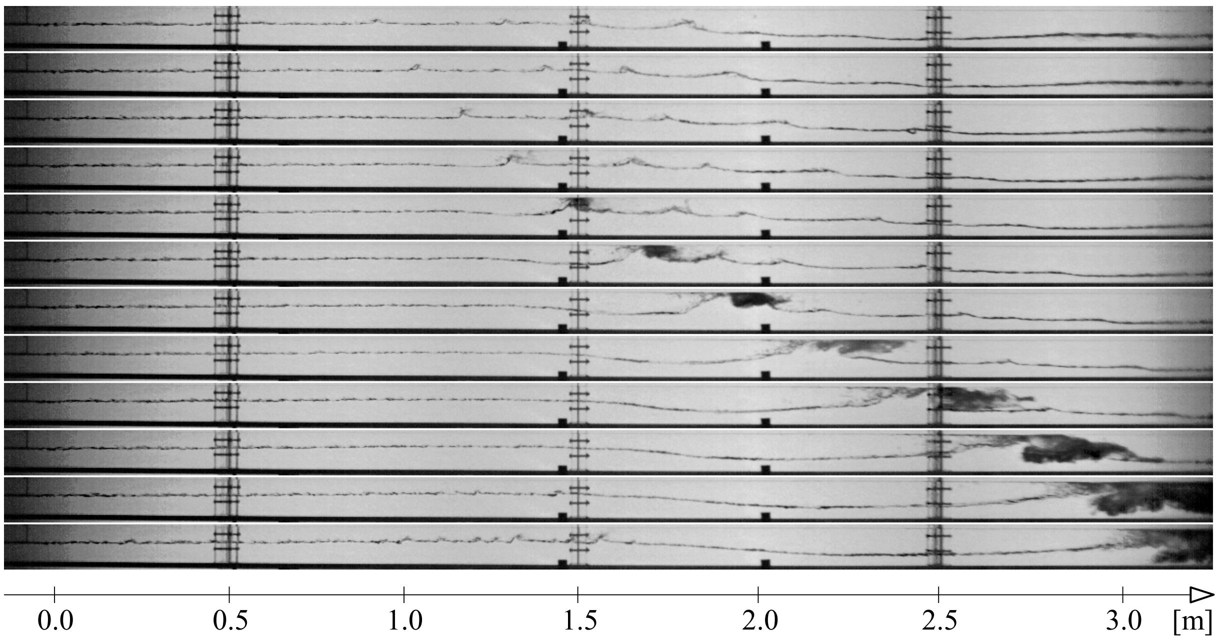


Fig. 4: Measured picture sequence at J_L = 1.0 m/s and J_G = 5.0 m/s with Δt = 50 ms

The camera pictures show that the inlet blade properly defines the water level: directly after the inlet, the flow is stratified. However, the interface is slightly wavy, due to the supercritical flow conditions imposed at the inlet. In fact, the Froude number Fr at the inlet cross-section is greater than unity:

$$Fr = \frac{\dot{V}}{W \cdot h} \cdot \frac{1}{\sqrt{g \cdot h}} = 2.86 \tag{1}$$

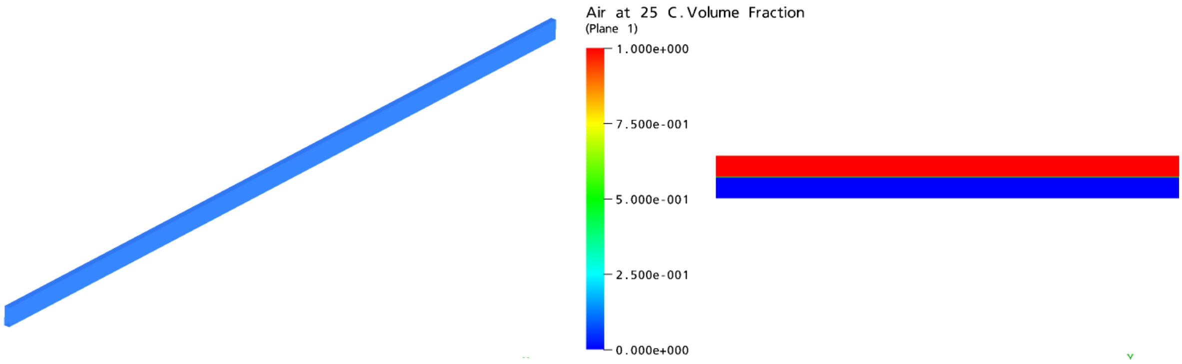
where $\dot{V}$ is the volume flow rate, W the channel width, h the water level and g the gravitational acceleration.

One of the small waves generated at the inlet grows rapidly downstream of 1.0 m and develops into a slug with closing of the cross-section at about 1.5 m. This slug travels along the channel faster than waves, catching some up and merging with them. At the slug front, an important droplet entrainment is visible which is driven by the air flow through the gap on top of the slug. Furthermore, apart from the slug, a sensible decrease of the water level was observed between 1.5 and 2.5 m. This shows an acceleration of the water flow, which can be attributed to the momentum exchange between the phases.

4. SLUG FLOW MODELING WITH *ANSYS-CFX* (FZD)

4.1 Main characteristics of the simulation

The channel with rectangular cross-section was modelled using *ANSYS-CFX* (2007). The three dimensional model dimensions are 8000 x 100 x 30 mm³ (length x height x width, see Fig. 5-a). The grid consists of 1.2×10^5 hexahedral elements. Accordingly to the experimental boundary conditions (horizontal inlet blade), the model inlet was divided into two parts: in the lower 50% of the inlet cross-section, water was injected and in the upper 50% air. An initial water level of y_0 = 50 mm was assumed for the entire model length (Fig. 5-b).



a. Fluid domain (channel) b. Air volume fraction, initial state (Zoom)

Fig. 5: Model and initial conditions of the mass fractions

In the simulation, both phases have been treated as isothermal and incompressible, at 25°C and at a reference pressure of 1 bar. A hydrostatic pressure was assumed for the liquid phase. Buoyancy effects between the two phases are taken into account by the directed gravity term. At the inlet, the turbulence properties were set using the “Medium intensity and Eddy viscosity ratio” option of the flow solver. This is equivalent to a turbulence intensity of 5% in both phases. The inner surface of the channel walls has been defined as hydraulically smooth with a non-slip boundary condition applied to both gaseous and liquid phases. The channel outlet was modelled with a pressure controlled outlet boundary condition.

4.2 Modeling of the interface

As it was the goal of the CFD calculation to induce surface instabilities, which are later generating waves and slugs, the interfacial momentum exchange and also the turbulence parameters had to be modelled correctly. Without any special treatment of the free surface, the high velocity gradients at the free surface, especially in the gaseous phase, generate too high turbulence throughout the two-phase flow when using the differential eddy viscosity models like the k-ε or the k-ω model (*ANSYS*, 2007). Therefore, certain damping of turbulence is necessary in the interfacial area because the mesh is too coarse to resolve the velocity gradient in the gas at the interface. On the gas side of the smooth free surface, this damping should be similar to that used near a solid wall. Moreover, on the liquid side the advanced model should take the anisotropy between the normal and the tangential Reynolds stresses into account. Yegorov (2004) proposed a simple grid dependent symmetric damping procedure. This procedure provides for the solid wall-like damping of turbulence in both gas and liquid phases. It is based on the standard ω-equation, formulated by Wilcox (1994) as follows:

$$\frac{\partial}{\partial t}(\rho\cdot\omega)+\nabla\cdot(\rho\cdot\mathbf{U}\cdot\omega)=\alpha\cdot\frac{\rho\cdot\omega}{k}\cdot\tau_t\cdot\dot{S}-\beta\cdot\rho\cdot\omega^2+\nabla\left[\left(\mu+\sigma_\omega\cdot\mu_t\right)\cdot\nabla\omega\right] \tag{2}$$

where α = 0.52 and β = 0.075 are the k-ω model closure coefficients of the generation and the destruction terms in the ω-equation, σ_ω = 0.5 is the inverse of the turbulent Prandtl number for ω, τ_t is the Reynolds stress tensor, and $\dot{S}$ is the strain-rate tensor.

In order to mimic the turbulence damping near the free surface, Yegorov (2004) introduced the following source term in the right hand side of the gas and liquid phase ω-equations (2):

$$A\cdot\Delta y\cdot\beta\cdot\rho_i\left(B\cdot\frac{6\cdot\mu_i}{\beta\cdot\rho_i\cdot\Delta n^2}\right)^2 \tag{3}$$

Here A is the interface area density, Δn is the typical grid cell size across the interface, ρ_i and μ_i are the density and viscosity of the phase i. The factor A activates this source term only at the free surface, where it cancels the standard ω-destruction term of the ω-equation $\left(-r_i\cdot\beta\cdot\rho_i\cdot\omega_i^2\right)$ and enforces the required high value of ω_i and thus the turbulence damping.

The parallel transient calculation of 15.0 s of simulation time on 4 CPU took 20 days. A high-resolution discretization scheme was used. For time integration, the fully implicit second order backward Euler method was applied with a constant time step of dt = 0.001 s and a maximum of 15 coefficient loops. A convergence in terms of the RMS values of the residuals to be less then 10^{-4} could be assured most of the time.

4.3 Results with *ANSYS-CFX*

In the following picture sequence (Fig. 6), the calculated phase distribution during slug generation is visualized. The first slug develops spontaneously at approximately t = 16.65 s after the beginning of the simulation, induced by instabilities. The simulated sequence shows that the qualitative behaviour of the creation and propagation of the slug is similar to the experiment (Fig. 4). The single effects leading to slug flow that can be simulated are:

- Instabilities and small waves are generated by the interfacial momentum transfer randomly. As a result bigger waves are generated.
- Bigger waves roll over and can close the channel cross-section. In this case, an important two-phase mixture is produced at the slug front.
- The slug can catch up waves and merge with them.
- A sensible decrease of the water level appears downstream of the region where slugs are generated.

However, a detailed comparison shows quantitative deviations between simulation and measurement. The needed entrance length for slug generation was defined as the length between the inlet and the location nearest the inlet where a wave closes nearly the entire cross-section. This was observed at about 1.5 m in the experiment and 3.5 m in the calculation.

These quantitative differences can be explained with the flow regimes observed at the test-section inlet. In fact, the flow pattern has an important influence on the momentum exchange between gas and liquid, especially at high velocity differences between the phases. Small disturbances of the interface provide a more efficient momentum transfer from the air to the water than in a stratified smooth flow. A high momentum transfer induces a rapid wave growth and therefore slug generation. In this case, a smooth interface was obtained over the 3 first meters in the simulation, whereas in the experiment supercritical flow waves were observed from the inlet of the channel. This means that the boundary conditions chosen for the CFD model do not reproduce the small disturbances observed in the experiment. In the end, a quite long channel length is needed before waves appear spontaneously in the simulation, deferring accordingly the instable wave growth to slugs downstream of the 3 m mark.

Finally, the quantitative differences noticed between simulation and experiment concern in particular the inlet boundary conditions. Because these have an important influence on the generation of the two-phase flow, future work should focus on the proper modelling of the small instabilities observed at the channel inlet.

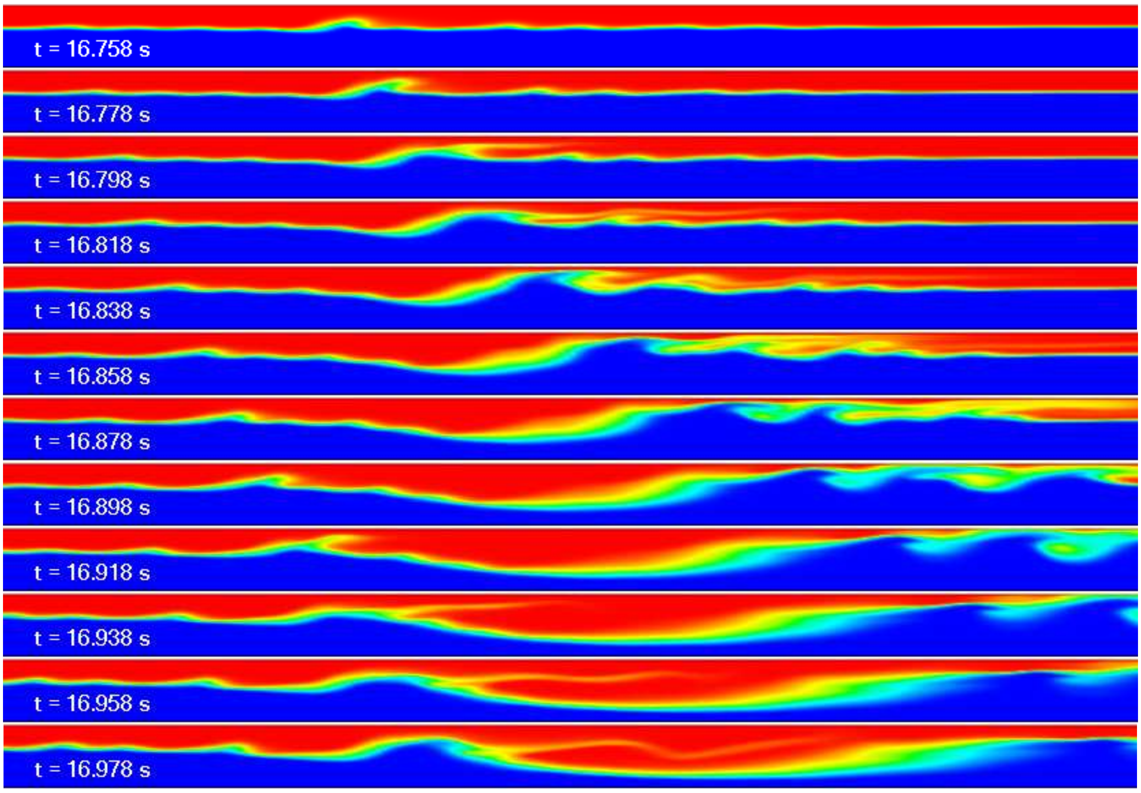


Fig. 6: Calculated picture sequence at J_L = 1.0 m/s and J_G = 5.0 m/s
(depicted part of the channel: 4.58m to 6.48m m after the inlet)

5. SLUG FLOW MODELING WITH *FLUENT*

The FLUENT CFD tool has also been investigated to simulate the HAWAC test at UCL. The VOF method was tried out for the simulation, as it provided good results for instabilities in free surface flow in a previous study (Bartosiewicz, 2008). The grid convergence was assessed in the same manner than in Bartosiewicz et al. (2008). Different viscous models were tested but without success: laminar, k-ε, , k- ω , k-ω SST, etc. No slug formation was observed in the simulations, but the interface remained smooth for all the transient. This was not attributed to an artificial high diffusion since it was even observed with a laminar model and second order central difference scheme.

The invoked reason is mainly that the VOF model is basically not able to simulate stratified flow with high slip between the phases at the interface, which is the case for the slug flow experiment chosen here. In fact, the slip between the phases is approximately equal to 8 m/s for the envisaged test. This fact raises an important issue in terms of best practice guide line. Indeed, for moderate slip and when surface tension may play an important role, it was shown that VOF approach is quite efficient and cheap. However, when phase slip becomes important, it seems that VOF is phased out. Consequently, it could be interesting to know this threshold in terms of slip and then to compare both approaches (VOF, multi-field) over a large range of conditions to determine it.

6. SLUG FLOW MODELING WITH *NEPTUNE_CFD*

6.1 Main characteristic of the simulation

The NEPTUNE_CFD 1.0.7 (last version) has also been used to simulate the HAWAC slug flow experiment. In the NEPTUNE code, the general compressible Eulerian multi-field balance equations are solved. In the case of two components, and for an isothermal, flow this two-field formulation is actually the classical two-fluid model and can be written for the field k :

$$\frac{\partial \alpha_k \rho_k}{\partial t} + \frac{\partial}{\partial x_i}\left(\alpha_k \rho_k u_{k,i}\right) = 0 \quad \text{for k} \in \left[1, n-1\right] \tag{4}$$

$$\sum_k \alpha_k = 1 \tag{5}$$

$$\frac{\partial\left(\alpha_k \rho_k u_{k,i}\right)}{\partial t} + \frac{\partial}{\partial x_j}\left(\alpha_k \rho_k u_{k,i} u_{k,j}\right) = -\alpha_k \frac{\partial P}{\partial x_i} + \frac{\partial \alpha_k \tau_{ij}}{\partial x_j} + \alpha_k \rho_k g_i + F_{k' \to k} \tag{6}$$

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\frac{\partial u_q}{\partial x_q}\delta_{ij} \tag{7}$$

Consequently, NEPTUNE solves a complete set of equation for each phase ($k-1$ for the continuity) while VOF method solves a momentum equation for the mixture. The turbulence is computed with classical k-ε model applied for each phase. This is an extension to multi-phase flows of the classical model widely used in single-phase flows.

A 2D grid was chosen to simulate the slug flow experiment of HAWAC. The computation domain extends up to 4 m, so the whole channel length is not modeled. The grid was relatively refined in the height (120 cells over 100 mm) which leads to 252.000 cells after grid convergence study. In order to develop correct velocity and turbulence profiles, the inlet is divided in two equivalent sections, and the two fluids join beyond 0.5m. The initial water level is 50 mm and the outlet condition is the hydrostatic pressure. The transient calculation uses an adaptative time step based on a maximum courant number equal to 1, which leads to a mean time step of 0.12ms; the total cost of the parallel simulation is about 60 h cpu on 64 processors, for a physical simulation of 10 s.

6.2 Modeling of the interface

For modelling the momentum transfer trough the interface, a dedicated Large Interface Model (LIM) was developed (Coste, 2007). This method directly allows for the location of the free surface, without any reconstruction method and takes into account momentum and turbulence exchanges between phases. The first step is to locate the Large Interface (LI). For this a refined gradient method is used on the liquid fraction. The refined gradient method is based on an interpolation of harmonic and anti-harmonic values. Then by comparing the three components of this refined gradient (Coste, 2007; Coste, 2008) to the maximum surface contained in the cell, which is only dependant on cell geometry, the single cells containing the interface are located. This method allows to detect "stratified cells" from the others. Since the interface is larger than the cell size and gas and liquid velocities are required to the closure, neighbouring gas and liquid cells are associated to this interface on each side to build the three-cell LI. These neighbouring cells are chosen in the direction given by the gradient of liquid volume fraction. Inside this three-cell layer, gas and liquid velocities are used to calculate the velocity of the interface and the momentum exchange by using a classical wall-function approach. In non

"stratified cells" and in the direction normal to the interface, a SIMMER-like isotropic drag force is applied (Coste 2004).
The model has been implemented in the NEPTUNE_CFD code and validated on several configurations including mass and energy exchanges in water-steam flows (Coste, 2008).

6.3 Result of the simulation with *NEPTUNE_CFD*

The calculated phase contours during the slug generation is given Fig. 7 for different times of the transient. The first slug is created at approximately between t = 1.5 s and t =1.6 s after the beginning of the simulation, which is much earlier than observed in the *ANSYS CFX* simulation. In addition, the location of the slug, where the whole cross section is closed by the water wave, is in good accordance with the experimental observations. This location is about 2 meters from the domain inlet or 1.5 meters from the end of the blade. In addition, the water level decrease is well represented as shown experimentally Fig. 4. Moreover, it appears that the computed slug velocity is roughly the same than that observed Fig. 4, e.g. about 5 m/s.

It is interesting to check that the slug formation is induced an instability or a perturbation appearing dowstream the blade which separate the water and the air flows. Fig. 8 gives a zoom of the interface which clearly shows the instability propagation during the time leading to the slug formation. It is believed that this perturbation is induced from the blade lip. The involved wave has a crest oriented in a counter current way and it is also visible during the first snapshots Fig. 7. However at t = 1.45 s the wave crest is turning back toward the flow direction, giving rise to the slug. This assumption could explain the reason why there is no significant growing of instabilities from the inlet in the CFX simulations where the blade is not included in the simulations. This fact could then induce a delay in the apparition of the slug.

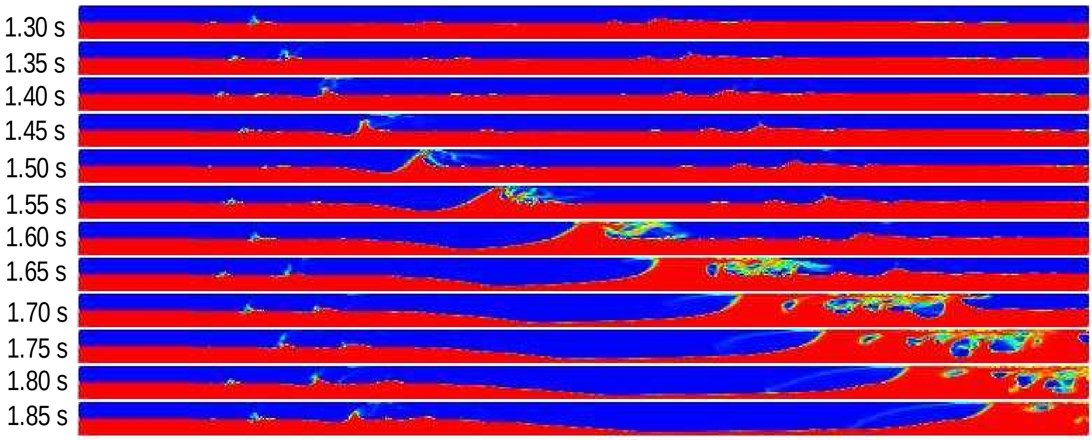


Fig. 7: Phase contours calculated with NEPTUNE_CFD at J_L = 1.0 m/s and J_G = 5.0 m/s

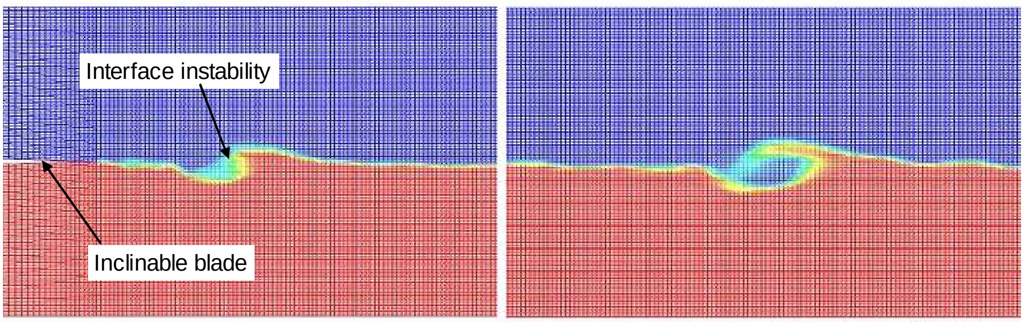


Fig. 8: Interface instability predicted by NEPTUNE_CFD at J_L = 1.0 m/s and J_G = 5.0 m/s

7. CONCLUSIONS: COMMENTS ON CFD-EXPERIMENT INTEGRATION

In a process of CFD and experimental validation, the most important and difficult issue is the matching of boundary and initial conditions, providing geometric conditions are perfectly matched. This requires not classical experimental stands but "CFD-grade" test benches, where special cares are taken in order to satisfy and control as far as possible this constraint. Even though this issue is satisfactorily addressed, it does not mean a successful validation. Indeed, the measurement should be taken with relevant quantities and at relevant locations according to the computational domain which will be further modeled. Those criteria are even more difficult to achieve if different computations and experimental tests are performed in different institutions as it is the case in this study. These points raise some crucial questions in such scientific collaborations where the coordination of the research in this CFD-experiment integration process is a key point. Therefore, through such experience, it is obvious that a general best practice guideline should be set up and scrupulously followed by both people involved in CFD and experiments. From our experience about the modeling of instabilities in stratified flows (Bartosiewicz, 2008; Vallée, 2008), some thoughts can be drawn:

- It is important that the design step of the experimental stand should be performed from the beginning by both people involved in experiments and in CFD in order to guaranty a simple stand, but with conditions and measured quantities matching those in the numerical model. For instance previous works showed how it was more difficult to perform some validation studies when experimental data come from a other study and long ago (Bartosiewicz, 2008). Also, the present analysis was firstly performed with a different stand where there was not a separating blade. In this case, the instability was initiated by a pump surge and boundary conditions for both stream was difficult to control separately, providing a very difficult task for CFD validation.
- The experimental stand has to be simple, providing access to both quantitative and qualitative values. Also, boundary and initial conditions should be controlled and known with a minimum uncertainty. In this study special cares are taken to fulfil those criteria. The blade separating both stream allows to know and control separately both flow conditions, and it is able to easily implement in the CFD model as it was found to play a key role in the generation of the instability. However, in this kind of flows, it is more difficult to fix the issue of outlet conditions. A solution was to put a long channel to limit end effects and to model a part of the channel with appropriate outlet conditions. A last but not least issue is the initial conditions. In our case, it is impossible to match experimental and CFD initial conditions. Consequently, it is not possible to compare results at a given absolute time. In this first preliminary study, validation dealt more on qualitative and relative parameters such as flow pattern, location and average velocity of the slug, etc… In the future, a deeper analysis should consist in comparing spectral information as it was performed in Bartosiewicz and Seynhaeve (2008).
- From the CFD point of view the convergence and the accuracy of simulations have to be assessed. The question about the relevance of a 3D or 2D calculation should be raised. As its was shown in this work, a 3D model may be not necessary in a first step. Indeed, the square experimental channel enable to limit 3D effects; in addition in a RANS strategy, we are not able to track some transverse secondary instabilities at small scales. As 2D calculations performed with NEPTUNE_CFD are encouraging, a deeper analysis in 2D should be then firstly performed. The grid convergence is also of primary importance. Normally, several grids should be tested and a control parameter at a key location should be chosen and plotted for those different grids until no variation is observed. However, in this particular case, the best way to choose the right grid should be based on spectral information in the flow dynamic (slug formation): for instance, as performed in (Bartosiewicz, 2008) the spectral analysis of the interface could be used to control the most amplified mode. In addition, the choice of the physical model should be also carefully chosen and justified. For instance in the case of stratified flows the use of VOF methods should be assessed according to the slip. For two-fluid models special cares have to be taken for the modeling of surface tension effects (Bartosiewicz, 2008) or momentum transfer to the interface. However, for those issues, encouraging models were recently developed an applied satisfactorily (Bartosiewicz, 2008, Coste, 2007, 2008) in the NEPTUNE-CFD platform. Moreover, in finite volume approach,

special discretization should be second order (central difference or bounded central difference) as far as possible to avoid additional numerical diffusion, and calculations should assess local residual convergence but also global conservation. Finally, the issue of turbulence modeling, especially in RANS, where different models should be investigated and compared to measurements and/or proper DNS calculations.

In regard of these different points, that are far to be exhaustive, the process of CFD-experiment integration is a difficult and long task if seriously conducted. This task becomes even more difficult if different parts are achieved in different institutions: in this case the coordination of the research becomes the limiting factor. However, base on our experience, if a rigorous best practice guideline is set up from the beginning within such consortium, we believe that such a collaborative research could success in this process of CFD-experiment integration.

ACKNOWLEDGEMENTS

This work is carried out in the frame of a current research project funded by the German Federal Ministry of Economics and Labour, project number 150 1329 in the one hand, and in the frame of the NURESIM project in the other hand. The NURESIM project is partly funded by the European Commission in the framework of the Sixth Euratom Framework Program.

NOMENCLATURE

Sign	Unit	Denomination
A	m^{-1}	interfacial area density
B	-	model parameter (damping of turbulent diffusion)
Fr	-	Froude number
g	m/s^2	gravitational acceleration
h	m	water level
H	m	channel height
J	m/s	superficial velocity
k	J/kg	turbulent kinetic energy
L	m	channel length
$\dot{S}$	s^{-1}	strain-rate tensor
t	s	time
U	m/s	velocity
$\dot{V}$	m^3/s	volume flow rate
W	m	channel width
z	m	z coordinate (channel length)
α	-	k-ω model closure coefficient of the generation term
β	-	k-ω model closure coefficient of the destruction term
Δn	-	typical grid cell size across the interface
ρ	kg/m^3	fluid density
σ_ω	-	inverse of the turbulent Prandtl number for ω
τ_t	N/m^2	Reynolds stress tensor
μ	$kg.m^{-1}.s^{-1}$	viscosity
ω	s^{-1}	specific dissipation

Indexes and abbreviations

G	index for gaseous phase
HAWAC	Horizontal Air/Water Channel of *FZD*
L	index for liquid phase
TOPFLOW	Transient Two Phase Flow test facility of *FZD*

REFERENCES

ANSYS Inc., *ANSYS CFX-10.0 User Manual*, 2007.

Y. Bartosiewicz, J. Laviéville, and J. M. Seynhaeve, "A first assessement of the NEPTUNE_CFD code: Instabilities in a stratitfied flow. Comparison between the VOF method and a two-field approach", *International Journal of Heat and Fluid Flow*, Vol. 29, pp. 460-478 (2008).

P. Coste, "Computational simulation of multi-D liquid-vapor thermal shock with condensation", *Proc. of Int. Conf. On Multiphase Flow 04*, Yokohama, Japan (2004).

P. Coste, "Modeling and Validation of NEPTUNE-CFD code about Turbulence and Friction in an Adiabatic Stratified Flow", 6th Euratom Framework Program NURESIM, *Deliverable D2.1.1.2b* (2007a).

P. Coste and J. Pouvreau, "NEPTUNE-CFD code calculations of COSI tests without weir", 6th Euratom Framework Program NURESIM, *Deliverable D2.1.1.1b* (2007b).

P. Coste, J. Pouvreau, C. Morel, J. Laviéville, M. Boucker, and A. Martin, "Modeling Turbulence and Friction around a Large Interface in a Three-Dimension Two-Velocity Eulerian Code", *Proc. of Int. Conf. NURETH 12*, Pittsburgh, USA (2007c).

P. Coste, J. Pouvreau, J. Laviéville, and M. Boucker, "A two-phase CFD approach to the PTS problem evaluated on COSI experiment", *Proc. of Int. Conf. ICONE16*, Orlando, USA (2008).

Y. Taitel, and A. E. Ducker, A, "Model for Predicting Flow Regime Transitions in Horizontal and Near Horizontal Gas Liquid Flow", *AICHE J.*, Vol. 22, pp. 47-55 (1976).

C Vallée, T. Höhne, H.-M. Prasser, and T Sühnel, "Experimental modeling and CFD simulation of air/water flow in a horizontal channel", *Proc. Of Int. Conf NURETH 11*, Avignon, France (2005).

C. Vallée, T. Höhne, H. –M. Prasser, and T. Sühnel, "Experimental investigation and CFD simulation of horizontal stratified two-phase flow phenomena", *Nuclear Engineering and Design*, Vol. 238/3, pp. 637-646 (2008).

G. B. Wallis, and J. E. Dobson, "The onset of Slugging in Horizontal Stratified Air Water Flow", *Int. J. Multiphase Flow*, Vol. 1, pp. 173-193 (1973).

D. C. Wilcox, *Turbulence modeling for CFD*, La Cañada, California: DCW Industries Inc, 1994.

Y. Yegorov, "Contact condensation in stratified steam-water flow", *EVOL-ECORA – D 07* (2004).