

Numerical evaluation of gas core length in free surface vortices

L Cristofano, M Nobili and G Caruso

“Sapienza” University of Rome – Dep. of Astronautic, Electrical and Energy Engineering (DIAEE) – Corso Vittorio Emanuele II, 244 – 00186 Rome, IT

Email: luca.cristofano@uniroma1.it, matteo.nobili@uniroma1.it,
gianfranco.caruso@uniroma1.it

Abstract. The formation and evolution of free surface vortices represent an important topic in many hydraulic intakes, since strong whirlpools introduce swirl flow at the intake, and could cause entrainment of floating matters and gas. In particular, gas entrainment phenomena are an important safety issue for Sodium cooled Fast Reactors, because the introduction of gas bubbles within the core causes dangerous reactivity fluctuation. In this paper, a numerical evaluation of the gas core length in free surface vortices is presented, according to two different approaches. In the first one, a prediction method, developed by the Japanese researcher Sakai and his team, has been applied. This method is based on the Burgers vortex model, and it is able to estimate the gas core length of a free surface vortex starting from two parameters calculated with single-phase CFD simulations. The two parameters are the circulation and the downward velocity gradient. The other approach consists in performing a two-phase CFD simulation of a free surface vortex, in order to numerically reproduce the gas-liquid interface deformation.. Mapped convergent mesh is used to reduce numerical error and a VOF (Volume Of Fluid) method was selected to track the gas-liquid interface. Two different turbulence models have been tested and analyzed. Experimental measurements of free surface vortices gas core length have been executed, using optical methods, and numerical results have been compared with experimental measurements. The computational domain and the boundary conditions of the CFD simulations were set consistently with the experimental test conditions.

1. Introduction

The deformation of gas-liquid interface (e.g. the free surface vortex formation) has been studied in several scientific and industrial fields during the last fifty years. Nevertheless, the interface transient behaviors, combined with the swirling motion in the vortex core, make the free surface vortex extremely difficult to understand. Strong whirlpool can cause gas entrainment (GE) phenomena, which consist in the entrapment of gas bubbles under the gas-liquid interface. If the downward flow is strong enough, the entrapped bubbles will be transported within the liquid and can affect equipment as well as plant operations. In sodium-cooled fast reactors (SFRs), entrained bubbles within the reactor core cause a positive reactivity insertion, which jeopardizes the reactor safety operation. Consequently, in the last years, also nuclear industry faces the problem of studying and predicting free surface vortex formation and evolution. Several experimental studies on GE occurrence have been carried out, and empirical correlations have been obtained [1]-[3]; however, experimental tests are always expensive, and the obtained results can be strongly affected by geometry and/or experimental conditions. For these reasons, different researchers started to study GE phenomena performing numerical simulations



with CFD codes. The definition of a reliable method, for the numerical prediction of free surface vortex formation and evolution, could be very useful since it allows to easily investigate several computational domains, boundary conditions and operating fluids (and their physical properties). The main drawback of this approach is the needed for accurate modeling of both the gas liquid interface curvature and the vortical flow near the deformed free surface.

Different researchers have tried to exploit CFD codes capabilities for the simulation of GE phenomena, testing different numerical models and schemes. Satpathy et al. [4] applied the VOF model to a 2D idealized slab of a FBR hot pool in order to obtain the threshold sodium velocity, at free surface, to avoid gas entrainment; afterwards, they applied this threshold value to the results from a 3D single-phase simulation, in a 90° symmetric sector of the hot-pool, to verify if GE onset conditions are reached. Chen et al. [5] performed simulations with standard k- ϵ model and RNG k- ϵ model (both coupled with VOF); the comparison of numerical results showed that standard k- ϵ model is not suitable for surface vortex simulations since it does not allow to predict turbulent kinetic energy, turbulent dissipation rate and turbulent viscosity distributions in vortex zone. In two different papers Kuljarni et al [6] and Durve et al. [7] numerically investigated shear type entrainment in surface aeration systems, applying Large Eddy Simulation (LES), coupled with VOF, and showed that shear type GE can be suitably simulated with this model. Merzari et al. [8] analyzed the benchmark case of Moriya [9] using LES and Detached Eddy Simulation (DES) turbulent models for CFD simulations (VOF has been used for two-phase flow calculations). The conclusions were that single phase simulations can be used for identifying the velocity field far from the free surface, while in the vortex region LES simulations, with an appropriate grid, have to be preferred. Škerlavaj et al [11] tested the Monji et al. [10] benchmark case with single-phase CFD simulations, concluding that Scale Adaptive Simulation with curvature correction (SAS-CC) is the best RANS turbulence model since it is in good agreement with laminar and LES simulations. A GE prediction method, based on CFD simulations and on Burgers' analytical model [12], has been proposed by Sakai et al. [13] and it is used to define design criteria for free surface GE in a fast breeder reactor. Recently, Ito et al. [14] developed a high-precision numerical simulation algorithm, for gas-liquid two-phase flows, able to simulate GE phenomena in large-sized Japanese Sodium-cooled Fast Reactor (JSFR). They showed that the algorithm is able to foresee the evolution of the vortex gas core also in complex geometries (e.g. the JSFR tank).

In this paper, the vortex formation transient has been analyzed and several numerical methods, aimed at evaluating the free surface vortex gas core length (L_{gc}), have been tested. A single-phase flow transient simulation has been performed and the Sakai's CFD-based prediction methodology has been applied. With the aim of numerically reproducing the gas liquid interface deformation, two-phase flow transient simulations have been carried out with the same computational domain and boundary conditions. Numerical results obtained with the different methodologies have been compared with experimental measurements.

2. Experimental setup

Onset of GE due to free surface vortex has been experimentally investigated in a test facility (Gas Entrainment Test Section-GETS) that does not provide any imposed rotation to the flow and the results have been presented in previous papers [1]-[2]. In the present study the GETS tank was modified introducing two baffles, which give a tangential inlet to the fluid on both side (see figure 1 and figure 2-a).

The reference vortex formation transient considered in the present paper and reproduced in the modified GETS facility is characterized by cold water as operating fluid, mass flow rate of 0.1 kg/s, water level at 0.05 m and a drain hole diameter of 0.026 m. The initial conditions are: stagnant water in the tank and no inlet mass flow rate; the pump is turned on at beginning of the analysis. Digital images of the occurring vortex (figure 5) have been acquired every 5 s during the transient in order to measure the vortex gas core length (L_{gc}).

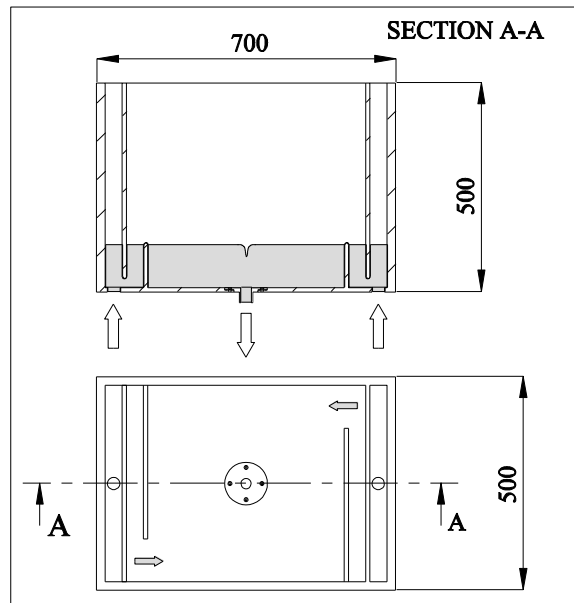


Figure 1. Vertical section and plant of the modified GETS tank.

3. CFD modeling

Three different simulations of the vortex formation transient have been simulated with ANSYS FLUENT® v.15, starting from stagnant conditions in the tank.

A first evaluation of the L_{gc} has been carried out by a single-phase simulation applying the Sakai's prediction method [12] (described in subsection 3.4). The circulation (Γ) and the downward velocity gradient (α) are evaluated from the CFD results and then they are combined with the Burgers' analytical model.

Two other numerical simulations have been carried out with incompressible two-phase flow and constant surface tension. Both of them use the VOF model to describe the interface evolution and a geometric Piecewise Linear Interface Construction (PLIC) method for interpolating the interface. Each of them uses a different turbulence model with the aim of identifying the influence of turbulence models on vortex gas core length evaluation.

Water at 20° C has been used as operating fluid.

The same time step (0.001 s) has been set for all CFD simulations and the Courant number has been kept around 0.5. Using the VOF explicit scheme, time step size near the interface was internally calculated on the basis of a user-defined maximum Courant number: .

$$Courant\ number = \frac{\Delta t}{\Delta x_{cell} / v_{fluid}} \quad (1)$$

In order to limit the computed sub time step to a maximum value equal to a quarter of the minimum transit time, a value of 0.25 has been selected as limit for Courant number near the interface (see [15]).

3.1. Computational domain and boundary conditions

The computational model adopted for single-phase flow simulation is shown in figure 2; it represents the GETS tank geometry and it is composed by a parallelepiped, joined to a cylinder in the bottom center, and connected to two smaller volumes (for tangential inlets) at two opposite sides. The larger volume has a 460 x 460 mm squared base and a height equal to the water level (i.e. 50 mm in the present case); the two smaller inlet zones are 100 mm wide and 20 mm high. The outlet connection, which has an inner diameter of 26 mm, is 30 mm long to reduce possible boundary effects on the CFD results.

Tank walls have been treated as no-slip boundary condition and a mass flow rate equal to 0.1 kg/s has been imposed to inlet and outlet boundaries. Top surface of the tank has been treated as a free-slip wall in single-phase flow simulations.

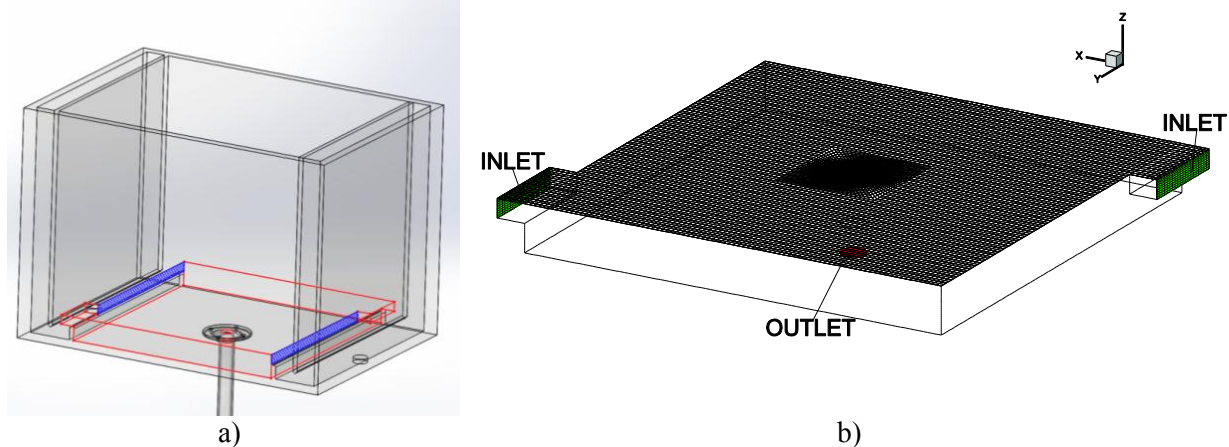


Figure 2. a) 3D view of the GETS tank; b) Computational domain.

The two-phase-flow computational model is slightly different, since it is 20 mm higher in order to consider a volume of the air above the free surface. The upper boundary of the simulation, which is no longer the gas-liquid interface, has been set to a wall with no-slip; moreover, a pressure inlet condition has been tested for this boundary, but errors in interface reconstruction have been detected.

3.2. Turbulence models and discretization scheme

As proposed by Sakai et al. [12], no turbulent model has been used in single-phase flow calculation in order to reduce the dependence of numerical results on the selected model.

The two-phase flow simulations have been tested by using a Scale Adaptive Simulation (SAS) model. The SAS model has been selected since it results the most suitable way to simulate free surface vortices, as per Škerlavaj et al. [11]. The SAS model is an improved URANS (Unsteady RANS) formulation based on the von Karman length-scale into the turbulence scale equation; this enables SAS model to dynamically adjust to the already resolved scales and allows the development of a turbulent spectrum in the detached regions. Therefore, the SAS model provides standard RANS capabilities for stable flow regions and LES-like behavior in unsteady regions.

PISO algorithm was used for pressure velocity coupling, while momentum equation was discretized using QUICK scheme. Pressure term was discretized using PRESTO scheme and volume fraction equation, in two-phase flow simulations, was discretized using the Geo- reconstruct scheme. Convergence criteria (Scaled Residuals) were set at 10^{-5} for all the equations, after a sensitivity analysis reported below. All simulations were carried out on i7 CPUs @2.93 GHz and 8 Gb RAM.

3.3. Grids

Structured mesh has been used in the present calculations; two different grids (Mesh A and Mesh B) were tested, at the beginning of the present analysis, in order to identify the most suitable one for surface vortex simulations. The two grids differ only near the drain region (see figure 3). The horizontal mesh size is between 5 mm in the outer region and 0.5 mm in the center; the axial mesh size is 2 mm except near the free surface where a refined mesh has been used (up to 1 mm).

A convergence criteria sensitivity analysis has been also carried out on two scaled residuals levels; the standard convergence criterion (10^{-3} for scaled residuals) results to be not suitable and a convergence criterion of 10^{-5} for scaled residuals has been selected. In table 1 the results of the sensitive analysis, for grid type and convergence criteria, are reported. Mesh B with scaled residuals limit equal to 10^{-5} , which give the best results, is used as reference configuration in the present study.

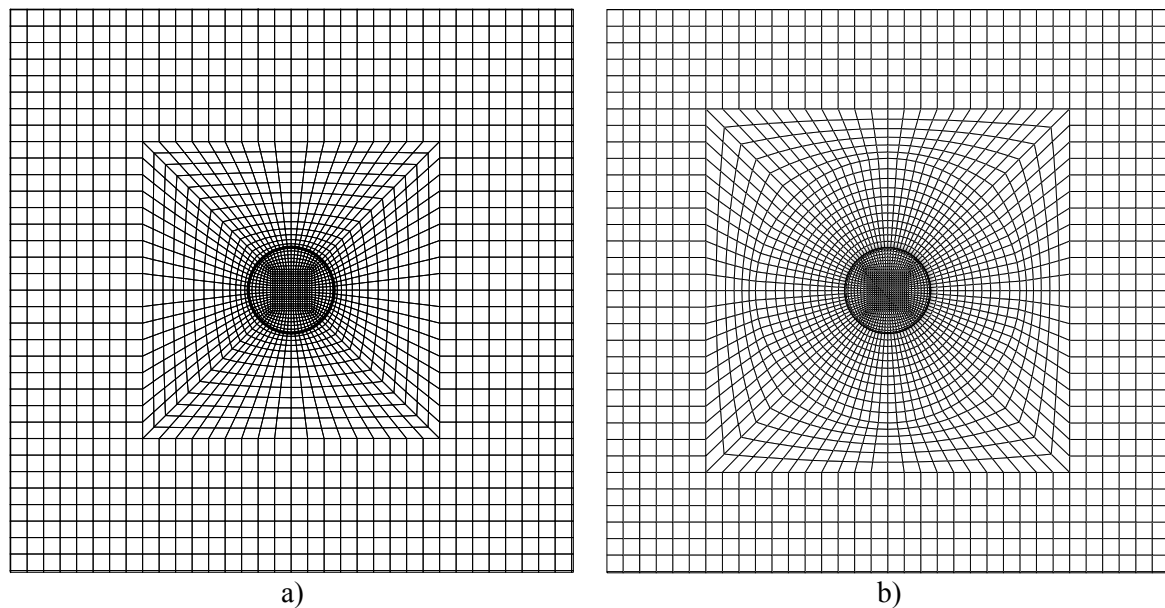


Figure 3. Mesh types: a) Mesh A, b) Mesh B.

Table 1. Tangential velocity peak at $h = 35$ mm.

Mesh type	Convergence criteria (Scaled Residuals)	Peak velocity CFD/experimental
A	10^{-3}	0.23
A	10^{-5}	0.53
B	10^{-3}	0.19
B	10^{-5}	1.18

3.4. Sakai's prediction method

Sakai et al. [13] proposed a gas core length evaluation methodology combined with CFD calculations. This method foresees a transient numerical simulation of vortical flows without interfacial deformations to reduce the computational cost (the interfaces are modeled as free-slip walls in the transient numerical simulation), then, the Burgers analytical model is applied to the CFD results to determine the strengths of the vortical flow.

Burgers' model was derived as a strict solution of Navier-Stokes equations and the velocity components are:

$$\begin{cases} u_r = -\frac{1}{2}\alpha r \\ u_\theta = \frac{\Gamma_\infty}{2\pi r} \left[1 - e^{-\frac{r^2}{r_0^2}} \right] \\ u_z = \alpha(z - h_\infty) \end{cases} \quad (2)$$

where r , θ e z represent radial, tangential and axial direction respectively, α is the downward velocity gradient, r_0 is the vortex radius, h_∞ is the submergence depth in the tank and Γ_∞ is the circulation of the vortical flow.

In the theoretical model, the relation between α and r_0 is:

$$r_0 = 2\sqrt{\frac{\nu}{\alpha}} \quad (3)$$

Applying a momentum balance in radial and axial directions and neglecting the advection terms in the Navier-Stokes equation, Sakai proposed to calculate the gas core length L_{gc} as:

$$L_{gc} = \frac{\log 2\alpha}{4g\nu} \left(\frac{\Gamma_\infty}{2\pi} \right)^2 \quad (4)$$

while the vortex circulation (Γ_∞) is calculated as:

$$\Gamma_\infty = \int_C \vec{u} \cdot \vec{ds} \quad (5)$$

In the first step, Γ_∞ is calculated assuming the isoline of the second invariant of the velocity deformation tensor with the value of zero as the initial integrating edge C . Then, the radius of the outer edge C is expanded gradually up to a value twice larger than the initial radius, and the maximum value of the so calculated circulations is selected as the circulation Γ_∞ of the vortical flow.

The downward velocity gradient α is calculated on the initial outer edge C as:

$$\alpha = \frac{\int_C \vec{u} \cdot \vec{n}_C ds}{A} \quad (6)$$

where $\vec{n}_C$ is the unit vector normal to C , ds is the local length of the outer edge and A is the area of the inner region. Introducing these two parameters into (4), the L_{gc} is calculated on the Burgers' theory.

4. Results and discussion

The gas core length L_{gc} evaluated through CFD simulations have been compared with experimental measurements; figure 4 shows the time evolution of the numerical and experimental results in the first 200 seconds of the transient. To calculate L_{gc} values of two-phase simulations, the height of the lower point of the iso-surface with a value of water volume fraction equal to 0.5 has been subtracted to the water level in the tank (0.05 m).

Experimentally, from 0 to about 30 seconds L_{gc} is equal to zero but the vortex begins to form. Then, the L_{gc} appears and it starts to increase almost linearly up to about 140 seconds. During the last part of the transient, L_{gc} presents an unsteady behavior with evident fluctuations; however, it seems that the average value of L_{gc} stabilizes around 30 mm. Images are acquired by a high-speed camera with 1024×1024 pixel resolution; each pixel corresponds to $8 \cdot 10^{-5}$ m. The absolute error of experimental data is the same for all L_{gc} values and it is equal to about 0.0015 m. The relative errors are, therefore, between 3% and 40% (greater uncertainties refer to L_{gc} values less than 0.005 m).

During the first seconds of the transient, when the vortex gas core has not yet formed, CFD predictions and experimental data are in good agreement; later, after 40 seconds, the three numerical methods foresee different trends.

It is evident from figure 4 that Sakai's method fails completely as it predicts an increase of L_{gc} up to too large values; for this reason, the single-phase simulation has been stopped after 100 seconds. Probably, the reason of this failure is that, neglecting the free surface deformation (modeling the interface as a flat free slip wall), the resulted velocity fields, and therefore circulation Γ and downward velocity gradient α , are not accurately calculated. In particular, the imposed flat interface affects the velocity component in z -direction near the free surface (and therefore the parameter α); in fact, since the velocity component perpendicular to a *wall-type* boundary by definition must be zero at the boundary, and since no interface deformation is enabled (because of the flat free slip wall condition), the axial velocity assumes small values over the whole free surface. In a real vortex core, instead, the

direction perpendicular to the interface no longer coincides with z -direction when interface is deformed and axial velocity assumes significant values at the free surface.

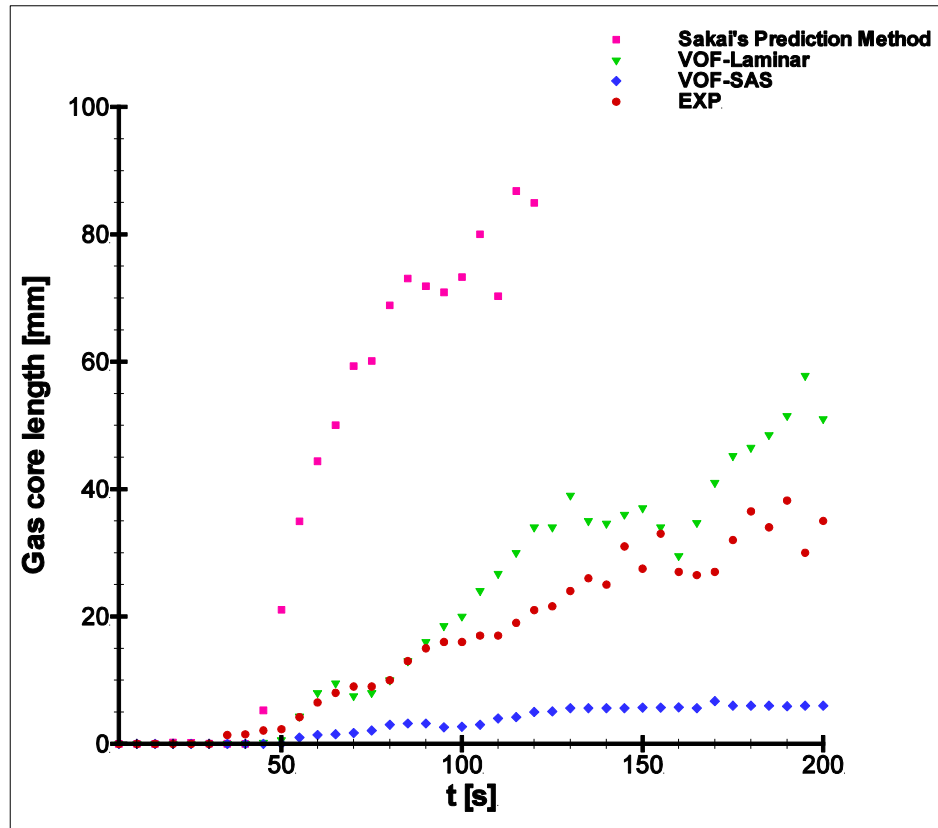


Figure 4. Experimental and numerical calculated L_{gc} during the transient.

Concerning L_{gc} values derived from the two-phase flow simulation with Laminar model agree very well with experimental data in the first 90 seconds of the transient; then, larger values respect to experimental L_{gc} are predicted. At the end of this numerical transient, values of about 50 mm (equal to the water level in the tank) are obtained. This discrepancy can probably be explained by an inaccurate modelling of turbulent phenomena by the Laminar model. In all kind of vortices, tangential velocity increases as the radius decreases (for the conservation of angular moment); in this zone turbulence' effects, like an increased turbulent diffusion, can be significant. Laminar model cannot reproduce this effect, and it results in a lower diffusion of vorticity in the outer zone, and a higher vortex intensity.

Two-phase flow simulation with SAS model, instead, underestimates the L_{gc} during the whole transient (except for the first 30 seconds where L_{gc} is null); it seems that a stationary condition is reached at the end of the transient with a value of L_{gc} equal to about 5 mm. Probably it can be caused by an excessive simplification of the inlet boundary conditions. The GETS tank, in fact, is characterized by a particular shape of the two inlet zones that reduce the free surface perturbations caused by the vertical inlet flow (see figure 1). This shape forces the flow to change several times its direction before reaching the tangential inlet, and provides an initial vorticity to the inlet fluid. In CFD simulation, instead, the flow is uniform and it is directed orthogonally to the inlet boundary, with zero inlet vorticity, resulting probably in lower L_{gc} values.

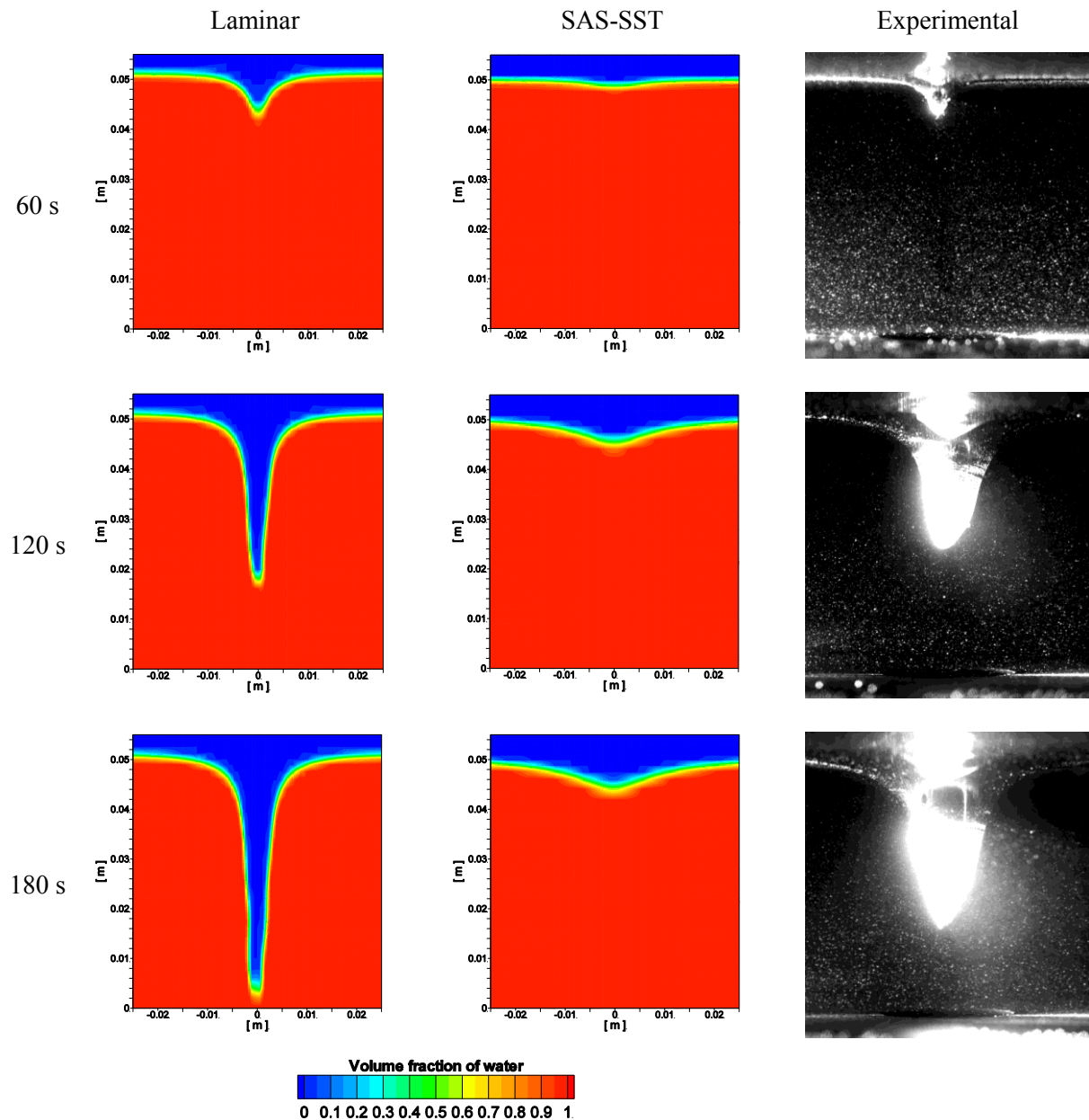


Figure 5. Comparison of experimental and numerical vortex L_{gc} for different phases of the transient.

In figure 5 the volume fraction maps (for the two-phase flow simulations) and the corresponding digital images from the experiment are shown, related to three different times: 60 s, 120 s and 180 s. It is clearly visible that the VOF simulation, with Laminar model, overestimates the L_{gc} during the second half of the transient; moreover it can be noted that the vortex gas core is thinner than the experimental case. As said previously, this is probably due to the inaccurate modeling of turbulent phenomena by Laminar model that results in a lower diffusion of vorticity from the vortex center to the outer zone. VOF simulation with SAS model seems to predict a larger gas core but it does not allow to obtain an acceptable evaluation of the vortex depth.

Figure 6 shows a details of the volume fraction map at 120 s for VOF simulation with Laminar model; it can be seen that the color-shift at the interface is confined in a very thin zone when the

interface is almost planar, while it is a bit larger when the curvature of the free surface is significant. However, the color-shift region is everywhere confined within few millimeters.

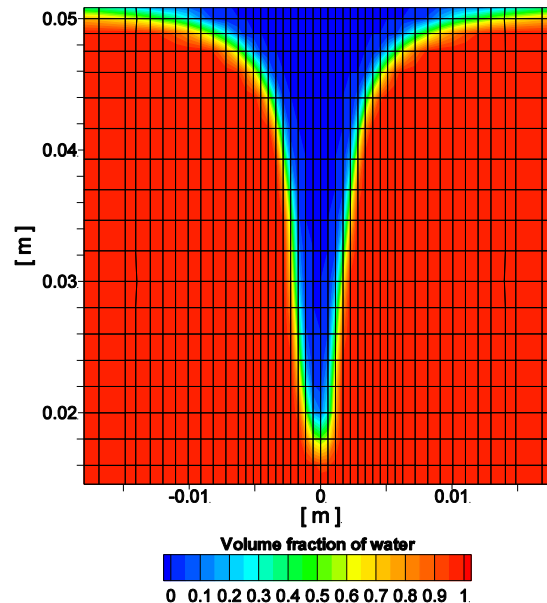


Figure 6. Detail of the volume fraction map at 120 s for VOF simulation with Laminar model with computational mesh.

5. Conclusions

A free surface vortex formation transient has been analyzed experimentally and numerical simulations to evaluate the gas core length L_{gc} have been carried out.

The experimental results show that the vortex gas core formation starts at about 35 seconds, then increases its length almost linearly; after about 140 seconds the L_{gc} has an unsteady behavior with an average value of about 30 mm.

Numerical results, obtained with different methods, predict well the first part of the transient, but then show different trends. Sakai's prediction method provides L_{gc} values completely in disagreement with experimental data; probably because of an incorrect evaluation of the velocity field (and hence of circulation Γ and downward velocity gradient α) near the free surface. Concerning two-phase flow simulations, Laminar model, coupled with VOF method, provides good results during the first 100 seconds, later it overestimates the gas core length L_{gc} . Moreover, the predicted vortex is thinner than the experimental one. SAS model underestimates the vortex depth probably because the inlet boundary conditions are not correctly imposed in term of vorticity.

Future studies on Laminar and SAS models are needed to establish which turbulent model is the most suitable to numerically simulate free surface vortices. Moreover, in order to improve these models, it is definitely required to provide more accurate boundary conditions in term of inlet vorticity; therefore it will be necessary to avoid any simplification of the computational domain, reproducing the exact geometry of GETS tank. Further experimental data, available in the next few months, will be very useful for a better assessment of the turbulence models capability.

Furthermore, it could be interesting to evaluate the capabilities of Large Eddy Simulation (LES) models in the analysis of free surface vortices. Although LES models need larger computational costs, they provide an improved modeling of turbulent phenomena that could lead to more accurate results.

List of symbols

A	area of the inner region, m ²	Δx_{cell}	cell dimension, m
C	initial outer edge of vortex	ε	turbulent dissipation, m ² s ⁻³
g	gravitational acceleration, m s ⁻²	ν	kinematic viscosity, m ² s ⁻¹
h_{∞}	submerged depth, m	<i>LES</i>	Large Eddy Simulation
k	turbulent kinetic energy, m ² s ⁻²	<i>DES</i>	Detached Eddy Simulation
L_{gc}	gas core length, m	<i>SAS</i>	Scale Adaptive Simulation
n_C	unit vector normal to C	<i>RANS</i>	Reynolds Averaged Navier-Stokes
r	radial coordinate, m	<i>JSFR</i>	Japanese Sodium-cooled Fast Reactor
r_0	vortex core radius, m	<i>GETS</i>	Gas Entrainment Test Section
u	velocity, m s ⁻¹	<i>VOF</i>	Volume of Fluid
u_r	radial velocity, m s ⁻¹	<i>GE</i>	Gas Entrainment
u_z	axial velocity, m s ⁻¹	<i>SFR</i>	Sodium-cooled Fast Reactor
u_{θ}	tangential velocity, m s ⁻¹	<i>FBR</i>	Fast Breeder Reactor
z	axial coordinate, m	<i>RNG</i>	Re-Normalisation Group
α	coefficient related to downward velocity, s ⁻¹	<i>URANS</i>	Unsteady Reynolds Navier Stokes
Γ	circulation, m ² s ⁻¹	<i>PRESTO</i>	Pressure Staggering Option
Γ_{∞}	circulation of the vertical flow, m ² s ⁻¹	<i>QUICK</i>	Quadratic Upstream Interpolation for Convective Kinematic
Δt	time step, s		

References

- [1] Cristofano L, Nobili M, Caruso G 2014 *Exp. Therm. Fluid Sci.* **52** 221-229
- [2] Caruso G, Cristofano L, Nobili M, Vitale Di Maio D 2014 *J. Phys. Conf. Ser.* **501**
- [3] Baum M R and Cook M E 1975 Gas entrainment at the free surface of a liquid: entrainment inception at a vortex with an unstable gas core, *Nucl. Eng. Des.* **32-2** 239-245
- [4] Satpathy K, Velusamy K, Chellapandi P 2011 *Energy Procedia* **7** 333-339
- [5] Chen Y, Wu C, Wang B, Du M 2012 *Procedia Engineering* **28** 55-60
- [6] Kulkarni A L, Patwardhan A W 2013 *Chem. Eng. Res. Des.* 025 (in press)
<http://dx.doi.org/10.1016/j.cherd.2013.10>
- [7] Durve A P, Patwardhan A W 2012 *Chem. Eng. Sci.* **73** 140-150
- [8] Merzari E, Ninokata H, Wang S, Baglietto E 2009 *Nucl Technol* **165** 313-320
- [9] Moriya S. 1993 *Report U93004, CRIEPI Abiko Research Laboratory* (in Japanese)
- [10] Monji H, Shinozaki T, Kamide H and Sakai T 2009 *J. Eng. Gas Turb. Power* **132(1)**
- [11] Škerlavaj A, Lipej A, Ravnik J, Škerget L 2010 *IOP Conf. Ser.: Earth Environ. Sci.* **12**
- [12] Burgers J M 1948 *Adv. In Appl. Mech* 171-99 (New York:Academic Press)
- [13] Sakai T, Eguchi Y, Monji H, Ito K, Ohshima H 2008 *Heat Transfer Eng.* **29** 731-739
- [14] Ito K, Kunugi T, Ohshima H, Kawamura T 2009 *J. Nucl. Sci. Technol.* **46** 366-373
- [15] ANSYS Fluent User's Guide, Release 15.