Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 82 (2015) 893 – 899

Energy Procedia

ATI 2015 - 70th Conference of the ATI Engineering Association

Influence of boundary conditions in numerical simulation of free surface vortices

Luca Cristofano[a]*, Matteo Nobili[a]

[a]*Sapienza University of Rome, Department of Astronautical, Electrical and Energy Engineering, C.so Vittorio Emanuele II 244, Rome 00186, Italy*

Abstract

A numerical evaluation of gas core length in free surface vortices had been performed, applying different approaches. In the past only partial satisfactory agreement with experimental results was obtained because of an excessive simplification of the computational domain. In this work, two-phase CFD simulations (with Volume Of Fluid method) have been performed using a computational domain without any simplification and the evolution of the vortex formation has been analyzed in term of gas core length. Different turbulence models have been tested. A mesh sensitivity analysis has been performed. The new numerical results obtained with the most advanced turbulence model (SST SAS-CC) show an improved agreement with experimental data, with respect to the previous results, confirming the importance of imposing proper boundary conditions to correctly simulate free surface vortices.

Peer-review under responsibility of the Scientific Committee of ATI 2015

Keywords: Free surface vortex, CFD, Volume of Fluid, Gas Entrainment, Scale Adaptive Simulation

1. Introduction

Free surface vortex formation represents a relevant problem in several industrial applications with hydraulic intakes, since strong whirlpools can cause the entrainment of floating solid impurities, with damages to the moving parts of hydraulic devices (i.e. pumps and turbines). Free surface vortices can also entrain gas bubbles under the gas-liquid interface (Gas Entrainment phenomena - GE); if the downward flow is strong enough, the entrapped bubbles can be transported within the liquid and can affect equipment as well as plant operations. GE is a relevant issue from a safety point of view in sodium-cooled fast reactors (SFRs) because entrained bubbles within the reactor core cause a positive reactivity insertion, which jeopardizes the reactor safety operation. Thus, in the last years nuclear industry is facing with the problem of studying and predicting free surface vortex formation and evolution. Experimental studies on

* Corresponding author. Tel.: +39 06 49918639; fax: +39 06 49918604.
E-mail address: luca.cristofano@uniroma1.it.

Peer-review under responsibility of the Scientific Committee of ATI 2015
doi:10.1016/j.egypro.2015.11.836

vortex formation have been carried out by several researchers during the years [2]-[4], but experimental tests are always expensive, and the obtained results are strongly affected by the experimental conditions. The numerical simulation of free surface vortex formation with CFD could be a powerful tool in any industrial fields; therefore, different researchers started studying a way to define a reliable numerical predicting method. However, the accuracy of numerical results is strongly related to a suitable modeling of the gas-liquid interface deformation and the vortical flow near the free surface. Even if the deformation of gas-liquid interface has been studied in several scientific and industrial fields during the last fifty years, the interface transient behavior, combined with the swirling motion in the vortex core, make the numerical simulation of free surface vortex extremely difficult to accurately perform.

Different researchers tried to exploit CFD codes capabilities for the prediction of free surface vortex, testing different numerical models and schemes. Škerlavaj et al [5] tested the Monji et al. [6] benchmark case with single-phase CFD simulations, concluding that Scale Adaptive Simulation with curvature correction (SAS-CC) is the best RANS turbulence model to simulate free surface vortex formation. Merzari et al. [7] analyzed the benchmark case of Moriya [8] applying Large Eddy Simulation (LES) and Detached Eddy Simulation (DES) turbulent models, confirming that single phase simulations can be used for identifying the velocity field only far from the free surface, while in the vortex region LES simulations, have to be preferred. Ito et al. [9] developed a high-precision numerical simulation algorithm for two-phase flows able to simulate GE phenomena in large-sized Japanese Sodium-cooled Fast Reactor (JSFR); their numerical model is able to foresee the evolution of the vortex gas core in complex geometries (e.g. the JSFR tank). In a previous paper, Cristofano et al [1] analyzed a vortex formation transient and tested different methods for the evaluation of the free surface vortex gas core length L_{gc}, but only a partial satisfactory agreement with experimental results was obtained, probably because of unsuitable boundary conditions due to a simplified computational domain.

In this paper, the same vortex formation transient has been numerically reproduced with a computational domain without geometry simplification, in order to verify the influence of boundary conditions on free surface vortex modeling. Two-phase flow transient simulations have been carried out using the same turbulent models previously tested in [1] (the Laminar model and the Shear Stress Transport k-ω coupled with Scale Adaptive Simulation model with curvature correction SAS-CC); in addition, the Standard k-ω model coupled with SAS-CC has been also tested. Numerical results obtained with the different methodologies have been compared with experimental measurements.

2. Experiments

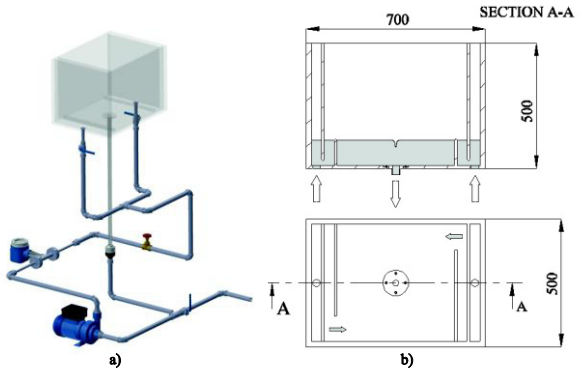


Fig. 1. (a) 3D view of the GETS facility; (b) Vertical section and plant of the modified GETS tank

The reference vortex formation transient considered in the present paper was reproduced in the Gas Entrainment Test Facility (GETS) facility, shown in Fig. 1(a); it consists of a closed loop with a rectangular transparent tank, a pump, a flow meter, valves and fittings. In the present study two baffles are introduced in the tank to give a tangential inlet to the fluid on both side (see Fig. 1(b)). The analyzed transient 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; the pump is turned on at t = 0. Digital images of the occurring vortex have been acquired every 5 s for 300 s, in order to measure the vortex gas core length (L_{gc}). The experimental transient was reproduced 10 times and average results were used as reference to validate numerical results.

3. Modeling

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

The numerical simulations have been carried out with incompressible two-phase flow and constant surface tension. VOF model has been used to describe the interface evolution and a geometric Piecewise Linear Interface Construction (PLIC) method for interpolating the interface has been applied. 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 global Courant number has been kept around 1. Using the VOF explicit scheme, time step size near the interface was internally calculated on the basis of a user-defined maximum Courant number; 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.

3.1. Computational domain and mesh

The computational model adopted in this study is shown in Fig. 2; it reproduces all the geometrical features of the GETS tank. The domain is 660 mm × 460 mm × 70 mm (L × W × H); the height of the computational domain is 20 mm higher than the water level in the tank (50 mm) for an air volume above the free surface. The outlet pipe is 30 mm long to reduce boundary effects on the results.

A structured mesh has been used in the present calculations (Fig 2). The horizontal mesh size is refined in the center to have a better simulation of the vertical flow and the free surface deformation; the vertical mesh is refined near the free surface for a more accurate modeling of the interface. A mesh sensitivity analysis has been performed with three different mesh sizes, summarized in Table 1; the tangential velocity profiles calculated with the three different meshes near the free surface have been compared with preliminary experimental results obtained from PIV (Particle Image Velocimetry) measurements. 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. A pressure inlet condition has been imposed at the top surface of the domain. The hydraulic diameter and the turbulence intensity has been imposed as boundary condition for the turbulent parameters k and ω to all boundaries, as described in ANSYS FLUENT® manuals.

Table 1. Characteristics of the three meshes

MESH	Horizontal mesh sizes [mm]	Vertical mesh sizes [mm]	Total element number
Coarse	1.14 ÷ 10	1.5 ÷ 2.5	≈ 216600
Medium	0.78 ÷ 7.18	1.2 ÷ 2	≈ 333800
Fine	0.56 ÷ 5	1.2 ÷ 2	≈ 588600

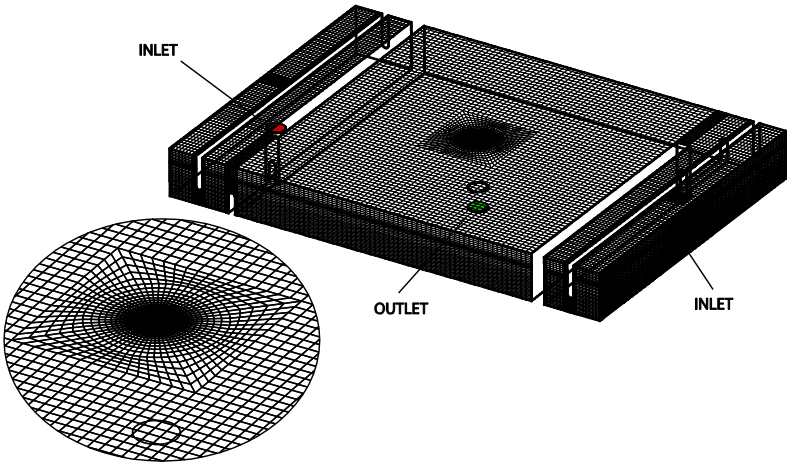


Fig. 2. Computational domain and mesh

The compared numerical and experimental profiles, shown in Fig. 3, refers to a vortex with a gas core length L_{gc} of 0.015 m; the three numerical profiles have been obtained with the same turbulence model (SST k-ω with SAS-CC). From Fig. 3 it can be noted that tangential velocity profiles obtained with MEDIUM and FINE mesh are very close to the experimental data; the agreement worsens moving near the vortex axis since free surface deformation introduces optical errors in PIV measurements. Therefore, in order to reduce the computational costs, the MEDIUM mesh size has been chosen.

3.2. Turbulence models and numerical schemes

Concerning the modeling of turbulent phenomena, a two-phase simulation has been performed without modeling turbulent flows (Laminar model in ANSYS Fluent), while for the other two simulations different turbulence models have been tested:

- the Standard k-ω model coupled with Scale Adaptive Simulation and Curvature Correction model (SAS-CC);
- the Shear-Stress Transport (SST) k-ω model coupled with SAS-CC.

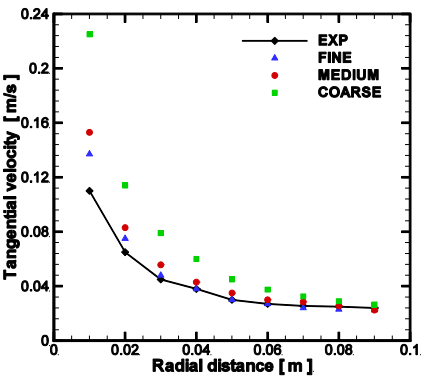


Fig. 3. Comparison between experimental and numerical tangential velocity profiles near the free surface for the three different mesh sizes for a vortex with L_{gc}= 0.015 m

The SAS model has been selected since, according to Škerlavaj et al. [6], it resulted the most suitable way to simulate free surface vortices. The SAS model provides standard RANS capabilities for stable flow regions and LES-like behavior in unsteady regions. It is an improved URANS (Unsteady RANS) formulation based on the von Karman length-scale into the turbulence scale equation and it is able to dynamically adjust to the already resolved scales and allows the development of a turbulent spectrum in the detached regions.

The two turbulence models tested in this study differs in resolving the stable flow regions. In particular the most significant differences of SST model respect to the Standard k-ω model are:

- a gradual change from the Standard k-ω model in the inner region of the boundary layer to a high-Reynolds number version of the k-ε model in the outer part of the boundary layer;
- a modified turbulent viscosity formulation to account for the transport effects of the principal turbulent shear stress.

The PISO algorithm was used for pressure velocity coupling, while momentum equation was discretized using the Bounded Central Differencing scheme. Pressure term was discretized using the Body Force Weighted scheme and the volume fraction equation was discretized using the Geo-reconstruct scheme. Convergence criteria (Scaled Residuals) were set at 10^{-6} for all the equations.

4. Results and discussion

Fig. 4 shows the time evolution of the numerical and experimental results in the first 300 seconds of the transient. The experimental data have been obtained by averaging the results of the 10 experimental test; the error bars in Fig. 4 represent the standard deviation of experimental data, while the two dashed lines give an idea of the maximum dispersion of experimental L_{gc} measured in the different tests, due to the typical unsteadiness of the physical phenomenon.

The gas core length values L_{gc} have been evaluated as the depth, with respect to the water level in the tank, of the lower point of the iso-surface with a value of water volume fraction equal to 0.5. From experimental results it can be noted that a visible free surface deformation appears after about 45 seconds, increasing quite rapidly till a L_{gc} value of about 0.015 mm in the subsequent 30 s; then, L_{gc} continues to increase, even if with a lower growing rate. After 200 s the gas core length stabilizes around an average value of about 0.03 m, reaching a pseudo-steady state; in each single test, in fact, the vortex gas core length shows persistent oscillation with a maximum amplitude of about 0.005 m (dashed lines).

Numerical results obtained with Laminar model (green line in Fig. 4) underestimate the L_{gc} values in the first part of the transient; then, after about 100 s the predicted gas core length reaches larger values than experimental data, settling around an average value of about 0.04 m. In some cases, L_{gc} overcomes the value of 0.05 m, which is the water level in the tank, meaning that the tip of the vortex entered in the outlet hole. The poor accurate results provided by the Laminar model can be probably referable to the lack of turbulent phenomena modeling; in this case, the turbulent viscosity effects are neglected and the diffusion of vorticity from the center of the vortex core to the outer zone is reduced. The higher vorticity accumulation results in a larger value of the gas core length. Respect to the previous results [1] obtained with Laminar model, no improvements have been achieved with the new computational domain, meaning that the Laminar model inaccuracy is not related to unsuitable boundary conditions but to the model itself.

The Standard k-ω model coupled with SAS-CC predicts quite well the gas core length values (red line in Fig. 4) in the first part of the transient (up to about 80 s), but then it overestimates the depth of the free surface vortex, settling around an average L_{gc} value of about 0.038 m. As can be seen from Fig. 4, in this case the gas core length is characterized by strong oscillations, with a maximum amplitude larger than 0.01 m; it seems that the vortex structure is unstable, with continuous variations of its strength.

The L_{gc} values evaluated from the CFD simulation with the SST k-ω model coupled with SAS-CC (blue line in Fig. 4) show the best agreement with experimental data, among the tested numerical methods. Numerical data predicted with this model, in fact, reproduce quite well the first part of the transient and then, after about 120 s, they reach an average value very close to that one of experimental results. Also in this case, the gas core length is characterized by an unsteady behavior; however, the oscillation amplitude of L_{gc} values is larger respect to the amplitude of experimental data.

In Fig. 4 the L_{gc} values (blue diamonds) obtained in the previous work [1] with the simplified computational domain with the SST k-ω SAS-CC turbulence model are also reported. It is evident the improvement achieved with the new domain, which impose more suitable inlet boundary conditions; therefore, it can be stated that the choice of appropriate boundary conditions for the numerical simulation of free surface vortices is very important to obtain reliable results.

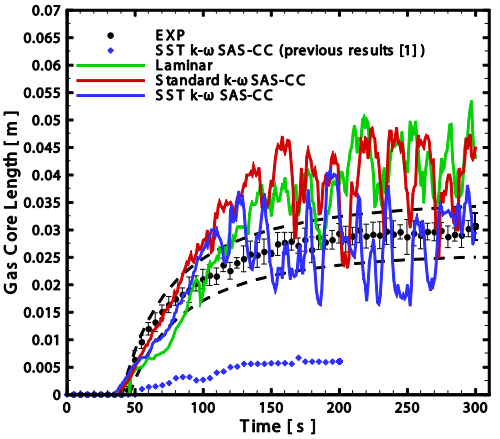


Fig. 4. Experimental and numerical calculated L_{gc} during the transient

In Fig. 5 the free surface deformation numerically predicted (with SST k-ω SAS-CC model) is compared with an experimental photo at 235 s; the comparison is done at this time because the numerical and experimental free surface vortex depth are very similar and it is possible to evaluate any difference in terms of radial dimension of gas-liquid interface. It seems that the gas core predicted by the numerical simulation (Fig. 4a) is slightly thinner respect to experimental evidence; this could be derived from an underestimation of the vorticity diffusion in the outer zone by the numerical simulations.

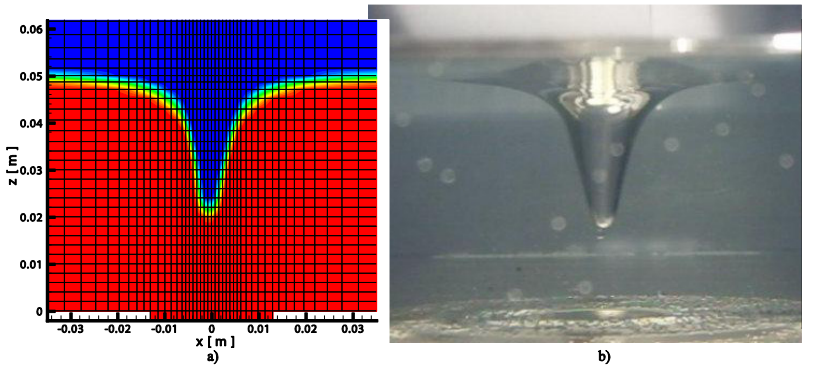


Fig. 5. Comparison of numerical predicted vortex L_{gc} (a) at 235 s with experimental photo (b)

5. Conclusions

The free surface vortex formation transient has been numerically reproduced through two-phase CFD simulations, to evaluate the gas core length values L_{gc}; a detailed computational domain has been adopted in order to evaluate the effects of inlet boundary conditions. Numerical results have been compared with experimental data. Numerical results obtained with Laminar model (coupled with VOF model) underestimate the L_{gc} values in the first part of the transient; then, after about 100 s the predicted gas core length reaches an average value larger than experimental ones. During the transient, L_{gc} overcomes the water level in the tank, meaning that the vortex reaches the outlet hole causing gas entrainment. The inaccurate results provided by the Laminar model are probably due to the reduction of vorticity diffusion from the center of the vortex core to the outer zone; thus, the vorticity accumulation near the vortex core is enhanced resulting in a larger value of the gas core length. The Standard k-ω model coupled with SAS-CC predicts quite well the initial transient, but then it overestimates the depth of the free surface vortex. Also in this case, the calculated gas core length is characterized by strong oscillations, with continuous variations of vortex strength. The L_{gc} values predicted with the SST k-ω SAS-CC model show the best agreement with experimental data. In this case the adoption of a more realistic computational domain and boundary conditions has provided an improved agreement. Consequently, this turbulence model can reproduce accurately the free surface vortex formation if appropriate boundary conditions are imposed. However a further effort is needed to achieve a better modeling of turbulent phenomena, in order to numerically reproduce the free surface vortex structure in vertical and radial directions.

References

[1] Cristofano L, Nobili M, Caruso G. Numerical evaluation of gas core length in free surface vortices. *J. Phys.: Conf. Ser.* 2014;**547**: 012030.

[2] Cristofano L, Nobili M, Caruso G. Experimental study on unstable free surface vortices and gas entrainment onset conditions. *Exp. Therm. Fluid Sci*. 2014;**52**:221-9.

[3] Caruso G, Cristofano L, Nobili M, Vitale Di Maio D. Experimental investigation of free surface vortices and definition of Gas Entrainment occurrence maps. *J. Phys. Conf. Ser.* 2014;**501:** 012019.

[4] Baum M R, Cook M E. Gas entrainment at the free surface of a liquid: entrainment inception at a vortex with an unstable gas core. *Nucl. Eng. Des.* 1975;**32**-2:239-45.

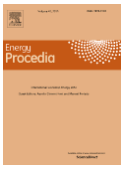
[5] Škerlavaj A, Lipej A, Ravnik J, Škerget L. Turbulence model comparison for a surface vortex simulation. *IOP Conf. Ser.: Earth Environ. Sci*. 2010;**12** 012034.

[6] Monji H, Shinozaki T, Kamide H and Sakai T. Effect of experimental conditions on gas core length and downward velocity of free surface vortex in cylindrical vessel *J. Eng. Gas Turb. Power*. 2009;**132(1):** 012901.

[7] Merzari E, Ninokata H, Wang S, Baglietto E. Numerical simulation of free-surface vortices. *Nucl Technol.* 2009;**165**:313-20.

[8] Moriya S. Evaluation method on gas entrainment from free surface: effects of model scale and working fluid properties. Report U93004, CRIEPI Abiko Research Laboratory 1993 (in Japanese).

[9] Ito K, Kunugi T, Ohshima H, Kawamura T. Formulations and validations of a High-Precision Volume-of-Fluid Algorithm on nonorthogonal meshes for numerical simulations of gas entrainment phenomena. *J. Nucl. Sci. Technol*. 2009;**46**:366-73.



Biography

Luca Cristofano was born in Campobasso (Italy), on June 11, 1985. He studied at the Sapienza, University of Rome and got his master degree in Energy and Nuclear Engineering. He is completing his PhD on “Energy and Environment” at Sapienza, University of Rome.