EXPERIMENTAL AND NUMERICAL INVESTIGATION OF THE FLOW AROUND A SUBMARINE SNORKEL MAST

Salvatore Marrone[1], Benjamin Bouscasse[1], Andrea Colagrossi[1], Matteo Antuono[1]
Franco Di Ció [1], Marco Masia[1], Emilio Campana [1]

1 CNR-INSEAN, Rome, Italy,
s.marrone@insean.it, b.bouscasse@insean.it, a.colagrossi@insean.it, m.antuono@insean.it
f.dicio@insean.it, m.masia@insean.it, e.campana@insean.it

ABSTRACT

The present work is dedicated to the experimental and numerical investigation of the flow field around a submarine snorkel mast moving with a forward speed in calm sea. The main objective is the determination of the pattern and extension of the outgoing jet which generates around the deplumer positioned on the upper part of the mast. Numerical simulations performed with a Smoothed Particle Hydrodynamics solver have been used as a complement to the experimental tests achieved in the INSEAN Towing Tank $n^\circ 1$ facility. The experiments have been conducted on full model scale to avoid effects related to the water surface tension which can reduce and inhibit the jet formation on a model scale. The numerical description of such a kind of flow is very challenging because of the large deformation and fragmentation of the free surface, especially for the highest forward speed conditions. To overcome the limit of the standard SPH solver, an innovative and promising SPH model has been adopted.

1 INTRODUCTION

After an operative submerged condition, a submarine needs to take air from above the water surface. This implies that the intake mast has to be designed in order to avoid the water inflow when operating in snorkel condition. A first interesting study is conducted on a steady-state condition, the submarine cruising at constant forward velocity in calm sea. The upper part of the snorkel mast is topped by a deplumer. To accurately predict the flow around the deplumer, the wave pattern generated by the submarine itself should be taken into account. Indeed, in the region close to the snorkel, the local wave pattern may modify the flow during the interaction with the snorkel mast.

As a first approach, a simplified problem is defined by considering the snorkel mast cruising in forward velocity and neglecting the effect of the submarine, as well as the effect of other infrastructures (e.g. periscope, antennas) positioned upstream. This approximation should help in understanding the capability of the deplumer profile to avoid entries of water in the air-intake system. Further, in such a simplified condition it is possible to perform an experiment in scale 1:1, reproducing the flow at the correct Weber number (i.e. wrong scale effects related to the water surface tension are avoided). Indeed, the nature of the flow around a submarine snorkel mast is strongly affected by the surface tension effects. If this flow is reproduced using the Froude analogy on a model scale, the surface tension may inhibit the jet fragmentation, leading to the generation of a uniform sheet of water. On the contrary, in real scale the jet around the mast fragments generating a clouds of droplets. Further, on a model scale the jet separation from the edges of the deplumer profile may be delayed or even suppressed. Using the Weber number to scale the experiments, the model speed must be increased by the square of the scale ratio. This can be unfeasible for a given experimental facility. Further, the violation of the Froude analogy can lead to a completely different flow pattern.

The numerical tests are challenging because of the large deformation of the free surface and the necessity of correctly simulating the water jets generated. Then, an innovative and promising numerical model, which is a variant of the Smoothed Particle Hydrodynamics scheme, has been adopted. A preliminary analysis on the influence of the domain size and, in particular, of the water depth used in the simulations have been performed. Indeed, for computational reasons related to the model adopted, the smaller the domain, the higher is the maximum resolution of the simulation. The main objective of the CFD analysis is the determination of the pattern and extension of the jets around the deplumer. The simulations have been used as a complement to the experimental tests performed in the INSEAN Towing Tank $\mathrm{n}^{\circ}1$ facility. Details on the mathematical model, as well as on the limitations of the present implementation, are given in sections 4 and 5.

2. EXPERIMENTAL SET-UP

The main aim of the experimental campaign is to reproduce the jets released by the deplumer edges in conditions as similar as possible to the submarine operative one. For this reason, a full scale model of the snorkel mast has been built. Tests with different immersion configurations and speed ranges have been set. A similar experimental campaign was previously performed at INSEAN in 1990 on a scaled model. The results of this previous campaign have been used to properly define the new experimental set-up. The snorkel mast model is obtained by milling the wood through a CNC machine and painting the outer skin.

A picture of the model is depicted in figure 1. The mast is a simple elliptical cylinder surmounted by a deplumer. The latter one has a flared geometry and the top part consists in a grooved crown. The groove is characterized by very small rays of curvature (of the order of centimeters). The spatial resolutions of the numerical simulations have been selected to capture the flow around the crown profile. The final height of the model is about $1.8m$. The support used to fix the mast at a given immersion has been designed to allow the maximum speed tested of $6m/s$ (about 12 knots). The bottom part has been closed by an aluminium plate. The top part of the deplumer is interchangeable, allowing different geometries to be studied. In front of the deplumer an horizontal cylinder has been fixed. In the real model, this hosts a control sensor and has to be considered a part of the problem in the study of the flow around the snorkel mast.

The support for mounting the model was built at INSEAN specifically for this project. An electric motor allowed setting the snorkel mast immersions in different configurations. The model was towed at INSEAN Tank $n^{\circ}1$, which has a length of $470m$, a cross section of $13.5m$ and a depth of $6.5m$. A plane of reference has been set from the top of the model in order to define the distance from the water surface. Figure 1 shows the carriage of the Tank $n^{\circ}1$ and the snorkel mast mounted below it. Figure 2 shows a rear view of the violent flow generated by the snorkel mast moving at $6m/s$. Note that the size of the water dome is about one-third of the tank width. This results show that large experimental facilities are needed to perform this kind of test.

Figure 1: Left: Snorkel mast model. Right: INSEAN Tank $n^{\circ}1$. The snorkel mast is mounted below the carriage through a designed support which rigidly fixes the mast at a given immersion.

Figure 2: INSEAN Tank $n°1$. Rear view of the violent flow generated by the snorkel mast moving at $6m/s$.

3. SPH NUMERICAL MODEL

Among the many numerical models available, the Smoothed Particle Hydrodynamics (hereinafter SPH) scheme seems to be a promising method in dealing with violent free-surface flows. It relies on a Lagrangian approach, that is, the fluid motion is described as a fluid-particle system and the derivatives of the flow quantities are computed along the trajectory of each particle. This leads to an intrinsic meshless character of the solver. The latter implies that the numerical grid points have no predetermined topological connections as in the case of mesh based methods. The derivatives of the fluid quantities are evaluated for each particle through an integral interpolation procedure over its neighbouring particles. Thanks to these features the SPH method can naturally treat breaking waves and fragmentation that generally are not easily handled by standard CFD methods. Due to its underlying structure, SPH was initially thought to solve astrophysical problems ([11]; [15]). Anyway, a few years later, its application range was extended to incompressible inviscid free-surface flows (Monaghan, 1994). For an extensive review of recent progress and developments in the SPH context, see [20], [18], [4], and [14].

For an inviscid, weakly compressible and barotropic fluid, the governing equations are:

$$\left\{ \begin{aligned} \frac{D\rho}{Dt} &= -\rho\,\mathrm{div}(\boldsymbol{u})\,; \quad p = c_0^2\,[\rho - \rho_0] \\ \frac{D\boldsymbol{u}}{Dt} &= -\frac{\nabla p}{\rho} + \boldsymbol{g} \\ \frac{D\boldsymbol{r}}{Dt} &= \boldsymbol{u} \end{aligned} \right. \tag{1}$$

where $\boldsymbol{r}$, $\boldsymbol{u}$, p and ρ are, respectively, the position of a generic material point, its velocity, pressure and density, $\boldsymbol{g}$ represents the gravity acceleration acting on the fluid, ρ_0 is the density at the free surface and c_0 the sound speed of the fluid.

In the SPH scheme the fluid domain is discretized in a finite number of particles representing elementary fluid volumes, each one with its own local mass m and other physical properties. In this context a generic field f at the position $\boldsymbol{r}_i$ of the i-th particle is approximated through the convolution sum:

$$\langle f\rangle(\boldsymbol{r}_i) = \sum_j f_j \, W(\boldsymbol{r}_j - \boldsymbol{r}_i; h) \, V_j \tag{2}$$

where f_j is the value associated to the generic particle j, V_j is its volume and, finally, W is a weight function that, in the SPH literature, is generally called kernel function (see sketch in figure 3). The kernel function has a compact support whose radius is proportional to h (also known as smoothing length) and tends to a delta Dirac function when h goes to zero. The integration of the kernel function on its support is imposed to be equal to one.

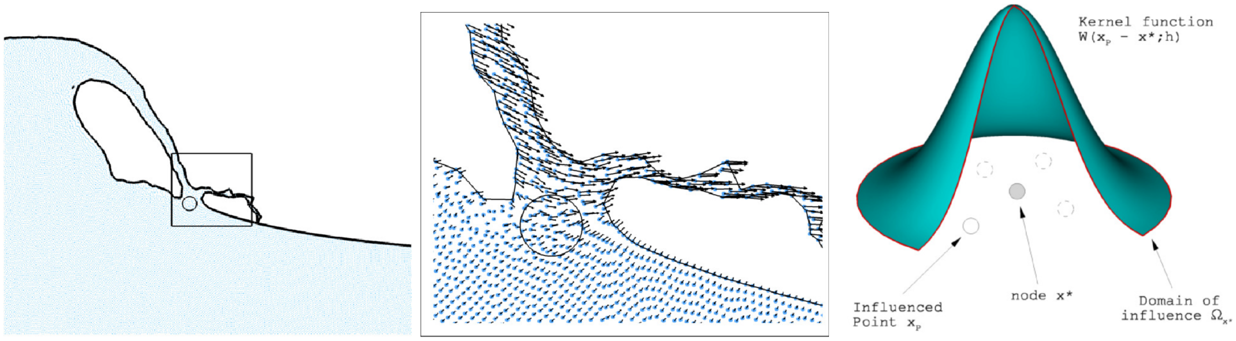

Figure 3: Sketch of the "smoothing" procedure. The fluid domain is discretized in particles interacting each other through a bell shaped kernel function.

In the present work the δ-SPH scheme by Antuono et al. (2010) [2] has been adopted. This model was further inspected in Marrone et al. (2011) [17] and in Antuono et al. (2011) [1]. In such a model an artificial diffusive term is added into the continuity equation in order to remove the spurious numerical high-frequency oscillations in the pressure field.

Discretizing the governing equation (1) the δ-SPH scheme reads:

$$\begin{cases} \dfrac{D\rho_i}{Dt} = -\rho_i \displaystyle\sum_j (\boldsymbol{u}_j - \boldsymbol{u}_i) \cdot \nabla_i W_{ij} \, V_j + \delta \, c_0 \, h \displaystyle\sum_j \boldsymbol{\psi}_{ij} \cdot \nabla_i W_{ij} \, V_j \, ; \quad p_i = c_0^2 \, (\rho_i - \rho_0) \\ \rho_i \, \dfrac{D\boldsymbol{u}_i}{Dt} = -\displaystyle\sum_j (p_j + p_i) \, \nabla_i W_{ij} \, V_j \; + \; \rho_i \, \boldsymbol{g} \\ \dfrac{D\boldsymbol{r}_i}{Dt} = \boldsymbol{u}_i \, , \end{cases} \tag{3}$$

Symbol ∇_i indicates the differentiation with respect to the position of the i-th particle. The argument of the diffusive term is:

$$\boldsymbol{\psi}_{ij} = 2 \, (\rho_j - \rho_i) \, \frac{(\boldsymbol{r}_j - \boldsymbol{r}_i)}{|\boldsymbol{r}_j - \boldsymbol{r}_i|^2} - \left[\langle \nabla\rho \rangle_i^L + \langle \nabla\rho \rangle_j^L \right]$$

Symbol $\langle \nabla\rho \rangle_i^L$ indicate the renormalized density gradient (see Randles & Libersky (1996) [19] for more details). The parameter δ governs the action of the numerical diffusive term and is set equal to $\delta = 0.25$. With this high value of δ it is not necessary to use any artificial viscous term in the momentum equation, further, the numerical scheme (3) is marched in time through a 4-th order Runge-Kutta scheme with a CFL factor equal to 1 (for details see *e.g* [3]).

To reduce the computational effort, it is a common practice in the weakly-compressible SPH solvers to use a sound velocity much smaller than the physical one, generally, one order of magnitude larger than the maximum expected velocity of the fluid. This corresponds to put $c_0 \geq 10 \, \max_i |\boldsymbol{u}_i|$. In this way the maximum Mach number allowed is 0.1, therefore the compressible effects can be considered negligible.

The system (3) preserves the global mass and both the linear and angular momenta. Finally, the diffusive term goes to zero as the spatial resolutions increases (that is, when h goes to zero) and, in this limit, the consistency with the Euler equations is recovered. Note that in the SPH scheme both dynamic and kinematic free-surface boundary conditions are intrinsically satisfied as proved by Colagrossi et al. (2009) [6]. Consequently, there is no need to detect the particles belonging to the free-surface. Indeed, the latter operation can be very complex and not unique for violent fragmented 3D flows like the one studied in this work (see *e.g* [16]).

For what concerns the boundary treatment, the problem of flow around a ship in constant forward motion requires the definition of suitable inflow/outflow boundary conditions to model the flow current. This is achieved through an extension to 3D of the algorithm proposed by Federico et al. [10] where buffer zones are defined to reproduce in/out-flow conditions. Moreover, in order to enforce solid boundary conditions along the snorkel mast, the vertical side and bottom walls, a robust and simple method is needed. To this purpose, the technique presented by [9] has been adopted.

The numerical effort to simulate the complete evolution of the flow around the snorkel mast with sufficient resolution is considerable. Consequently, an ad hoc hybrid MPI-OpenMP parallelization has been developed to achieve simulations of several millions of particles running on a computer cluster for about three weeks.

4. NUMERICAL RESULTS

In the numerical tests, the snorkel mast is towed at a given velocity of $6m/s$ in the calm water. Specifically, the problem is solved in the frame of reference of the mast which is, therefore, fixed in the coordinate system adopted. The fluid domain needs to be bounded on the bottom and on the vertical sides in order to create a channel with in/outflow and reproduce the flow current. The mast is connected directly to the bottom. A sketch of the numerical domain is depicted in figure 4.

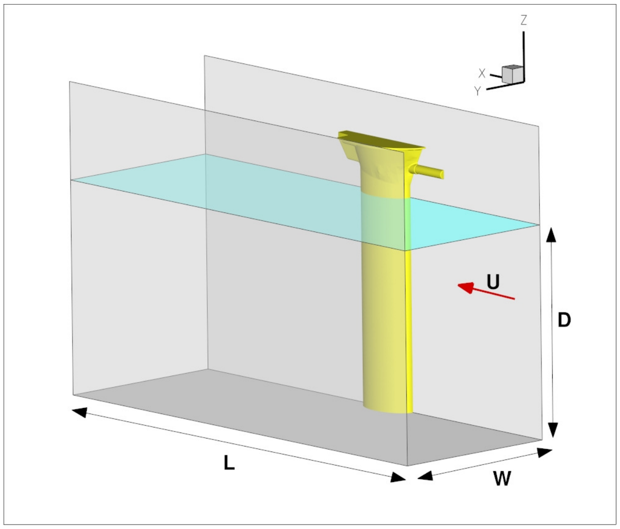

Figure 4: Sketch of the fluid domain adopted for the numerical model.

The SPH numerical solver is not able to manage simulations with particles characterized by a large variation of size. Consequently, the use of a uniform spatial resolution limits the maximum size domain, $L \times D \times W$, since the CPU time cost increases with a N^4 law, where N is the particle number. Hereinafter, all the spatial dimensions will be referred to the beam, B, of the underwater mast which is a simple elliptical cylinder. The center of the ellipse has coordinates $x = 0$, $y = 0$ while the plane $z = 0$ coincides with top part of the deplumer. In this way it is easier to highlight portions of water with positive z which can cause water entries in the mast. In this frame of reference the water level is set equal to $z = -1.46B$.

Before running the simulation with the finest resolution, a comparison is performed between the flow obtained in a channel with depth $D = 7.3B$ and the one obtained in a channel with depth $D = 4.4B$. Indeed, it is necessary to assess that the smaller domain is able to correctly reproduce the flow around the snorkel mast, since the bottom may influence the flow. For these simulations the domain is discretized with particles of size $\Delta x = B/10$. This spatial resolution is insufficient to fully resolves the main flow features and, in particular, it is not possible to capture the jet outgoing the deplumer crown. However, these preliminary rough simulations allowed defining the final domain size, ensuring that the depth $D = 4.4B$ is sufficient to neglect the bottom effects.

A simulation has been performed with a finer resolution, namely $\Delta x = B/40$. The results are shown in figure 5. With this resolution the deplumer jet is well described and the flow follows regularly the profile of the deplumer (right plot of figure 5). The extension of the water dome is about 2.5B. In the front part of the domain, the flow rises along the mast, impacts the horizontal cylinder, moves along the deplumer side wall and, incidentally, exits from the bevelled edge of the deplumer crown. No water intake in the mast is observed during this simulation. Indeed, as displayed in the left plot of figure 5, all the water levels are negative. However, the resolution adopted is still not sufficient to assess the shape influence of the deplumer crown profile. To perform this simulation, 15 Milions of particles have been used to simulate a physical time of 5 seconds. The code has run on 16 Opteron nodes (2-way quad-core 2.1GHz each) for 160 hours.

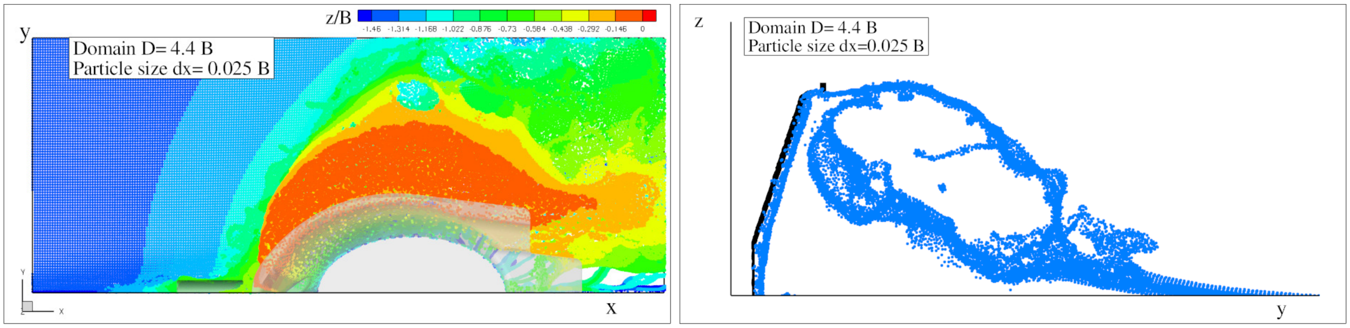

Figure 5: Left: top view of the SPH wave elevation around the deplumer ($\Delta x = B/40$). Right: transversal slice of the fluid domain ($-0.24B \leq x \leq 0.24B$, $\Delta x = B/40$).

To further reduce the particles size, a multi-resolution scheme is needed. The multi-resolution technique for SPH model has been developed in the astrophysics context by using the so called h-variable formulation (see e.g. Hernquist and Katz [12]). This formulation permits the use of particles of different sizes, even if an efficient parallelization of this solver is quite complex. A possible simple solution has been proposed by Landrini et al. (2007), where the smoothing length is maintained constant using different particles sizes. This leads to change the interacting neighbour particles. On the bottom part of the fluid domain, the latter have been fixed to a minimum value which is increased close to the free-surface region. This procedure allows only a particles size ratio of 1.5. A detailed analysis of the new solution has been done to check possible spurious numerical influence on the flow that could degrade the results.

4.1 Further results using a multi-resolutions scheme

The left plot of figure 6 displays the contour levels of the heights of the water dome around the deplumer with the multi-resolution scheme. In particular, the fluid is discretized with $\Delta x = B/62$ in the top layer (from the free surface up to $z = -B$), and $\Delta x = B/40$ for the rest of the domain. In this way the higher layer of particles, involved in the flow of interest, has a finer discretization and it is possible to better resolve the jet flow in the top part of the snorkel mast characterized by high curvature of the deplumer grooved crown. On the right plot of the same figure, a detail of the jet outgoing from the deplumer edge is reported. The jet exits with a speed close to the inflow velocity, that is $U = 6m/s$. Further, the portion of the jet showed in this figure has the x-component of the velocity which is almost zero. This means that the incoming fluid is deflected of 90 degree in the deplumer jet without any kinetic loss.

Plots in figure 7 show respectively the horizontal cuts near the water surface and close to the top part of the deplumer. These figures allow a deeper understanding of the flow trajectory after past the deplumer profile. At $z = -0.49B$ the flow slows down in front of the mast and, then, accelerates downstream along the mast, mainly in the streamwise direction. At

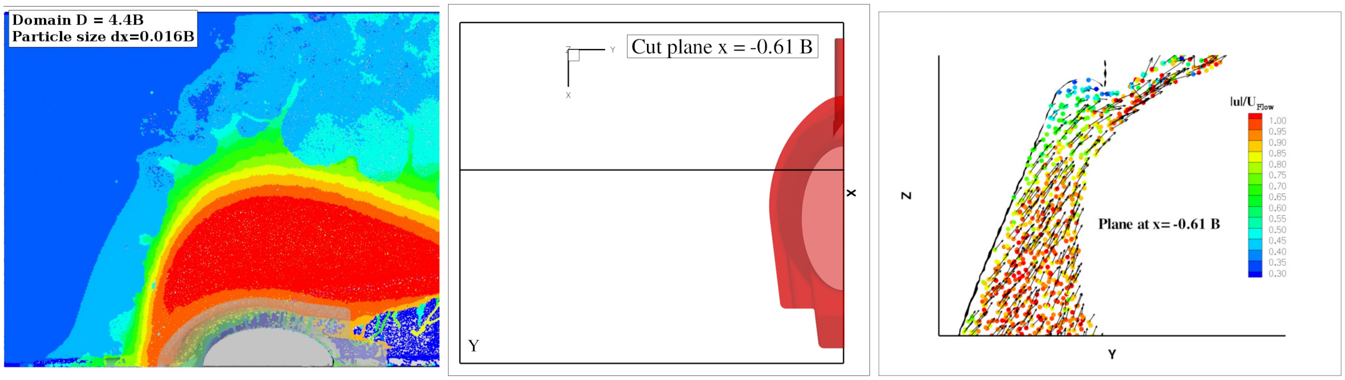

Figure 6: Left: SPH heights of the water dome around the deplumer using a multi-resolution scheme. The free-surface particles have a size equal to $\Delta x = B/62$. Middle: transversal slice used for the jet visualization. Right: details of the deplumer jet on the transversal slice at $x = -0.61B$.

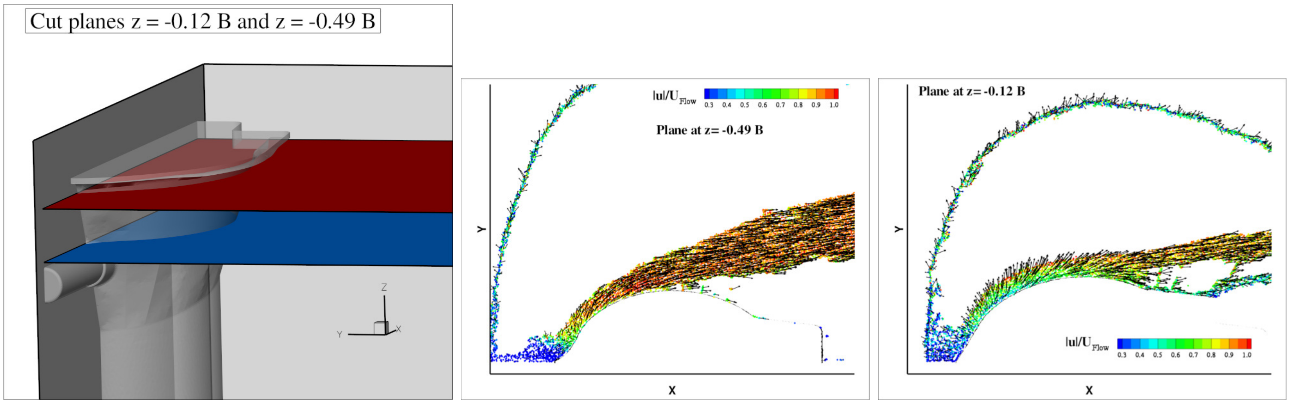

Figure 7: Left: horizontal cuts for flow visualization. Middle: horizontal slice of the flow at $z = -0.49B$. Right: horizontal slice of the flow at $z = -0.05B$. Particles are colored with the intensity of their velocity

$z = -0.12B$, that is, closer to the deplumer crown, the flow is slower and the velocity vectors present a large component orthogonal to the body profile, the flow being deflected by the upper part of the deplumer. In the experiments, the water surrounds the whole mast profile while, in the numerical results, the flow separates from the mast in the rear region. In any case, this differences do not modify the conclusions about water intakes, since the rear part of the mast is not critical for this aspect.

A 3D view of the water dome obtained with the multi-resolution scheme is depicted in left plot of figure 8. To run this simulation, 25 millions of particles have been used on a 128 opteron cluster machine. The simulation of 2.6 seconds of physical time required 8 days of computing time. The results presented in the right plot of the same figure show that the flow in the aft part of domain completely surrounds the cylinder which, in practice, has no influence in deflecting the jets of water released by the deplumer crown profile. Conversely, from the experiments (see picture 9) it seems that the flow impacts violently on the cylinder, producing a large fragmentation. As a consequence, drops of water fall inside the mast. The main features of the flow are well captured by the numerical simulation, in particular in the front part. The main differences are given by the presence of the air-water mixture and droplets in the experiments due to large turbulent phenomena which cannot be easily resolved by numerical solvers. In the mathematical model adopted, the presence of air is not taken into account, as well as the surface tension effects which cause the formation of the small drops. Consequently, even with a higher spatial resolution, it would not be possible to describe the small drops and jets that blur the experimental

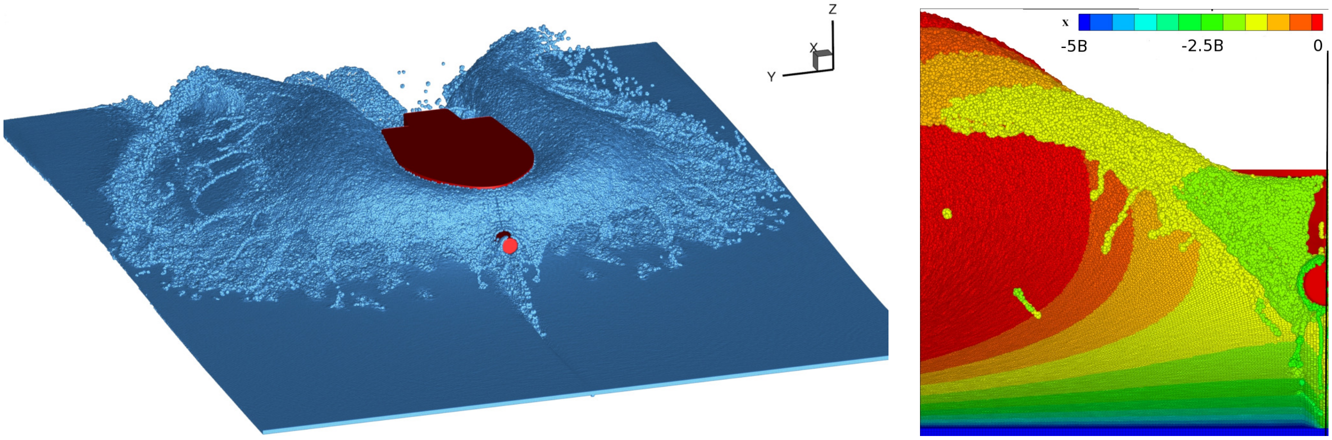

Figure 8: Left: 3D view of the SPH water dome around the deplumer using a multi-resolution scheme. Right: detail of the flow around the snorkel mast, front view.

pictures. Nonetheless, with the highest resolution, the deplumer jet is well represented, as shown by the figure 10 where a detail of the flow around the deplumer edge is depicted. This permits a correct evaluation of the size of the water dome, even if, as commented above, the local flow features highlighted in the experiments cannot be captured in the simulations.

Figure 9: Front view of the flow from experiments (the water depth is the same used in the numerical simulations). Left: low velocity condition ($U = 6$ kn). Right: high velocity condition ($U = 12$ kn).

6. CONCLUSION

The present work describes an experimental and numerical investigation of the flow field around a submarine snorkel mast. The analysis is focused on the determination of the pattern and extension of the outgoing jet flow around the deplumer. The experimental activities have been conducted on full model scale using the INSEAN towing tank $n°1$ facility. The numerical solution of such a kind of flow is challenging because of the complexity of the free surface behaviour. Numerical simulations performed with a Smoothed Particle Hydrodynamics solver have been used as a complement of the experiments. To overcome the limit of the standard SPH solver, an innovative and promising SPH model has been adopted.

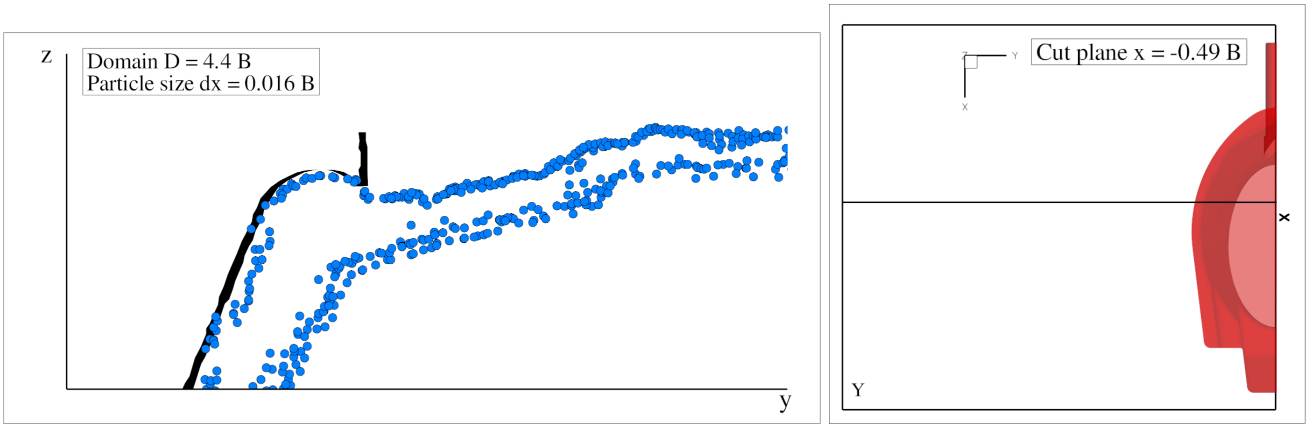

Figure 10: Left: detail of the jet exiting from the deplumer profile at $x = -0.49B$. Rigth: the plane used for the SPH jet visualization.

ACKOWLEDGEMENTS

The numerical computations presented here have been performed on the parallel machines of CASPUR Supercomputing Center (Rome) under the Standard HPC Grant 2012 within the project "SPH simulations of bow waves dynamics", their support is gratefully acknowledged.

The research leading to these results has received funding from the European Community's Seventh Framework Programme (FP7/2007-2013) under grant agreement n. 225967 "NextMuSE".

References

[1] Antuono, M., Colagrossi, A., Marrone, S., Lugni, C., (2011), "Propagation of gravity waves through an SPH scheme with numerical diffusive terms", Computer Physics Communications, 182, 866877.

[2] Antuono, M., Colagrossi,A. ,Marrone, S., Molteni, D., (2010), "Free-surface flows solved by means of SPH schemes with numerical diffusive terms", Computer Physics Communications, 181 (3), 532549.

[3] Antuono, M., Colagrossi,A. ,Marrone, S., (2012), "On the use of numerical diffusive terms in weakly-compressible SPH schemes" 7th International SPHERIC workshop, 28-30 May, Prato (Italy).

[4] Cleary, P., Prakash, M., Ha, J., Stokes, N., and Scott, C. (2007), "Smoothed particle hydrodynamics: Status and future potential", Progress in Computational Fluid Dynamics, 7, pp. 70–90.

[5] Colagrossi, A., (2005), "A Meshless Lagrangian Method for Free-Surface and Interface Flows with Fragmentation", Ph.D.Thesis, Department of Mechanical Engineering, University of Rome, LaSapienza, http://padis.uniroma1.it

[6] Colagrossi, A., Antuono, M., Le Touzé , D. , (2009), "Theoretical considerations on the free surface role in the SPH model", Physical Review E 79/5,056701:1-13.

[7] Colagrossi, A., Landrini, M., (2003), "Numerical simulation of interfacial flows by smoothed particle hydrodynamics", Journal of Computational Physics, 191/2, 448475.

[8] Colagrossi, A., Landrini,M. ,Tulin, M.P., (2001), "Numerical studies of breaking bow waves compared to experimental observations", Proceedings of 4th Numerical Towing Tank Symposium, Hamburg, Germany.

[9] M. De Leffe, D. Le Touz, B. Alessandrini, (2009) Normal flux method at the boundary for SPH, Proc. 4th International SPHERIC Workshop, 26-29 May, Nantes.

[10] Federico I., Marrone S., Colagrossi A., Aristodemo F., Antuono M., (2012) "Simulating 2D open-channel flows through a SPH model", Journal of Mechanics - B/Fluids, 34: European pp. 35–46.

[11] Gingold, R.A., Monaghan, J.J., (1977), "Smoothed particle hydrodynamics: theory and application to non-spherical stars", Monthly Notices of the Royal Astronomical Society, 181, 375389.

[12] Hernquist, L. and N. Katz, (1989), "TreeSPH: A Unification of SPH with the Hierarchical Tree Method", Proc. Astrophysical Journal Supplement, 70, pp. 419446.

[13] Landrini M., Colagrossi, A., Greco, M., Tulin, M.P., (2007), "Gridless simulations of splashing processes and near-shore bore propagation", Journal of Fluid Mechanics, 591, 183-213.

[14] Liu, M. B. and Liu, G. R. (2010), "Smoothed particle hydrodynamics (SPH): an overview and recent developments", Arch. Comput. Meth. Eng. 17, 2576.

[15] L.B. Lucy, (1977), A numerical approach to the testing of the fission hypothesis, Astron. J., Vol 82, pp. 10131024, 1977.

[16] Marrone, S., Colagrossi, A., Le Touzé, D., Graziani, G., (2010), "Fast free-surface detection and level-set function definition in SPH solvers", Journal of Computational Physics ,229, 36523663.

[17] Marrone S., Antuono M., Colagrossi A., Colicchio G., Le Touzé D., Graziani G., (2011), "δ-SPH model for simulating violent impact flows", Comput. Methods Appl. Mech. Engrg, 200: pp. 1526-1542.

[18] Monaghan J.J.,(2005), "Smoothed Particle Hydrodynamics", *Rep. Prog. Phys.*, 68: 1703-1759.

[19] Randles P.W. & Libersky L.D., (1996), "Smoothed Particle Hydrodynamics: Some recent improvements and applications" Comput. Methods Appl. Mech. Engng., 139, pp. 375–408.

[20] Violeau, D., (2012), "Fluid Mechanics and the SPH Method, Theory and Applications", Oxford University Press.