CFD ANALYSIS OF DEVICES IN OSCILLATING WATER COLUMN - OWC

VALERIA RUSSO*, DANIELE NICOLINI, TOMMASO CRESCENZI

* ENEA – Italian National Agency for New Technology, Energy and Sustainable Economic Development, Renewable Energy Technical Unit, Casaccia Research Center
Via Anguillarese 301, 00123 S. Maria di Galeria, Rome (Italy)
e-mail <valeria.russo@enea.it>

Key words: Computational Methods, Marine Engineering, Fluid dynamic simulation, 3-D grid model

Abstract. The paper is about the development of a fluid-dynamic model for simulating the behaviour of a device for the production of electric energy based on an oscillating water column (OWC).

First of all it has been developed a 3-D model of the device using a tool in order to create the grid used for the simulation.

The code used for the fluid dynamic simulation is OpenFoam, in particular the solver InterFoam suitable for two-phase isothermal flow characterized by immiscible and incompressible fluids, it uses the VOF (Volume of Fluids) model.

The aim of this work is the optimization of the model from the computational point of view, in particular it has been conducted a sensitivity analysis on the processor's number and on the time step of calculation.

In the first phase of the work were taken into account two forcing, approximated with a sinusoidal profile, corresponding to two types of waves more realistic for the site in which it was made the experimental apparatus.

Subsequently it has been developed an analysis of the boundary condition in order to simulate in the best way the incident wave.

Obviously it is possible to investigate more in detail the model realized in order to optimize the simulation and validate the model with more experimental data.

1 INTRODUCTION

The present paper describes the development of model for the CFD simulation of the U-OWC (Oscillating Water Column) devices.

The CFD simulation allows to optimize the sizing of the device through few experimental tests in order to reduce the cost and the time of development so as to make it economically sustainable activity.

The objective of this paper is to develop a numerical model able to predict the behaviour of both water and air inside the device; in this way it is possible to define the optimal geometry of the device with a little number of experimental tests.

Because of the large amounts of calculations related to this kind of elaborations, it was necessary to perform an optimization from the computational point of view.

2 GENERAL SPECIFICATIONS

The geometrical characteristic of the U-OWC devices is represented in the figure below (Figure 1).

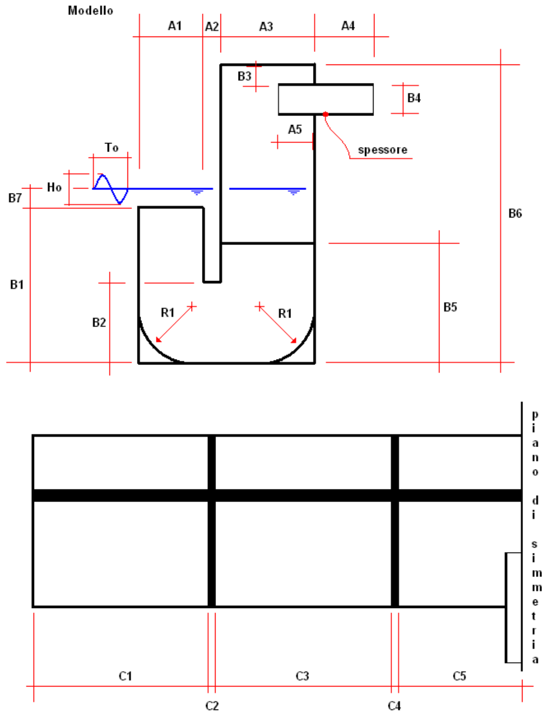

Figure 1: Scheme of the U-OWC device

The system is composed by two volumes, one of water (the oscillating column) and the other of air; the air is pushed from the water below inside the turbine duct.

In order to simulate the system operation it has been used the OpenFoam code; this code is capable to develop thermo-fluid-dynamics simulations in 3D. It is an open source software produced by Open CFD Ltd.

For the numerical simulation it have been taken into account two different types of waves corresponding to two different operating condition of the OWC system:

- wind waves
- long swells

3 CFD MODEL

3.1 Geometrical characteristics of the CFD model

Here (Figure 2) is represented a scheme of the system used for the numerical analysis.

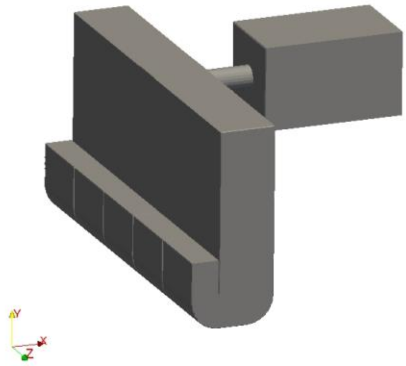

Figure 2: System scheme

In particular it is possible to note that in the model was not taken into account the turbine into the outlet duct; this choice was made because the objective of this paper is to evaluate the capability of the model to simulate the behavior of the water/air system. Moreover also the experimental tests were conducted on the system without the turbine.

3.2 Shape of the waves

For the system simulation have been taken into account two types of forcings corresponding to two types of waves present on the site where the system is located.

These waves were summarized with sinusoidal trends. The characteristics of the first wave are:

- wave height (minimum/maximum) = 0.438 m
- wave period (time between two maxima) = 2.75 s.

The characteristics of the second wave are:

- wave height (minimum/maximum) = 0.232 m
- wave period (time between two maxima) = 7.10 s.

3.3 Fluids

For the CFD simulation were taken into account two fluids: air and sea water at ambient temperature (293K) and pressure (101350 Pa). In these conditions the characteristics of the two fluids are summarized in Table 1.

Table 1: Fluids carhacteristics

	Air	Sea Water
Density (kg/m^3)	1.2	1025
Kinematic viscosity (m^2/s)	1.46e-5	1.11e-6

The surface tension (σ) is equal to 0.073 N/m.

3.4 Physical behavior of the flow and numerical model

In order to develop a CFD simulation it is necessary to define the real condition of the flow within the volume to be analyzed; after that it is possible to choose the best numerical model capable to simulate the phenomena.

In this case the flow is two phase, turbulent and with incompressible fluids.

In order to simulate the system consisting of air in the gaseous phase and sea water in liquid phase it has been used the VOF (Volume Of Fluid) model.

For what concerns the type of flow, if it is laminar or turbulent, it is necessary to evaluate the Reynolds Number.

In order to evaluate the Reynolds Number it was assumed that, during the rising phase of the sea level, the volume of sea water is equal to the volume of air that passes inside of the turbine duct; so in the presence of a wave of the first type it is possible evaluate Re of about 4.5e5; so for the simulations it has been used the Standard k-ε model.

For defining if the fluids are incompressible it is necessary to evaluate the Mach number defined as:

$$Ma=v/c \tag{8}$$

where v is the flow velocity and c is the sound velocity in the considered fluid.

Because of the sound velocity of the air is 330 m/s, in the case of the first wave, the Mach number can be estimated about 0.05, so the fluids can be considered incompressible.

3.5 The mesh

The dimension of the mesh must be little enough to simulate all the physics characteristics of the real flow, but the greater is the number of cells the greater is the computation time. In order to limit the computation time, the mesh must be done and optimized to obtain the necessary accuracy limiting the number of cells.

More in detail, inside the volume of sea water and air far enough from turbine duct the flow is slow, so the cell dimension defines the accuracy in identifying the position of the surface between the air and water; the mesh was been realized with 2.5 cm dimension cell.

Instead in the volume near the duct turbine the cell dimension is related to the flow velocity that is in the order of some tens of m/s. In this case it is important to evaluate the influence of the wall using specific function like as wall functions. In order to obtain acceptable results, this requirement must be satisfied:

$$30 < y^+ < 200 \tag{9}$$

where y^+ is a parameter

$$y^+ = c_t\, y / \nu \tag{10}$$

$$c_t = \sqrt{(\tau_0/\rho)} \tag{11}$$

- ν: kinematic viscosity
- y: distance between the center of the cell nearest to the wall and the wall

To define the cell dimension was following this procedure:

- $y^+=200$
- $f=0.0791Re^{-0.25}$ (Blasius relation valid for turbulent flow inside tube)
- $c_t=|U|\sqrt{(f/2)}$ U average velocity inside the tube
- $\Delta y=2y=2y^+\nu/c_t$

The mesh has been realized dividing the entire model in cubic elements of 25 mm, in a second step a cylindrical zone with a diameter equal to 0.60 m containing the turbine duct and extended longitudinally inside the model and, out of the tube, inside the extra volume, is discretized with a more thick cells (cell dimension 6-7 mm).

The transition zone between the two different mesh has an extension of about 15-20cm.

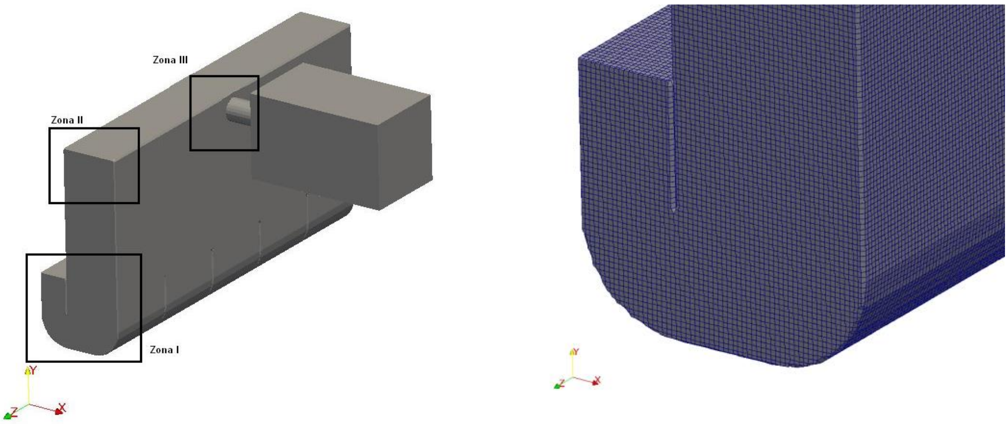

Figure 3: a)Area of interest for the mesh b) Mesh - Area I

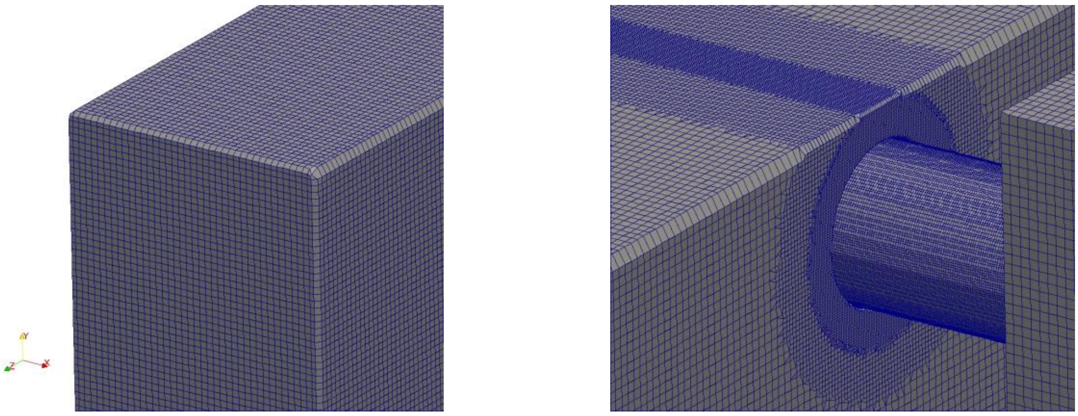

Figure 4: a)Mesh – Area II b) Mesh – Area III

In Figure 3a) is represented the entire model with three relevant zone highlighted, in Figure 3b) there is the first zone, in the Figure 4 there are the other two zone of interest.

The total number of the cells for this model is equal about to 5.1 million.

3.6 Boundary and initial condition

In Figure 5 are represented the model and the boundary condition of the entire external surface.

The external surfaces are all wall (grey one) except the water inlet/outlet (blue one) surface and air inlet/outlet surface (light blue one).

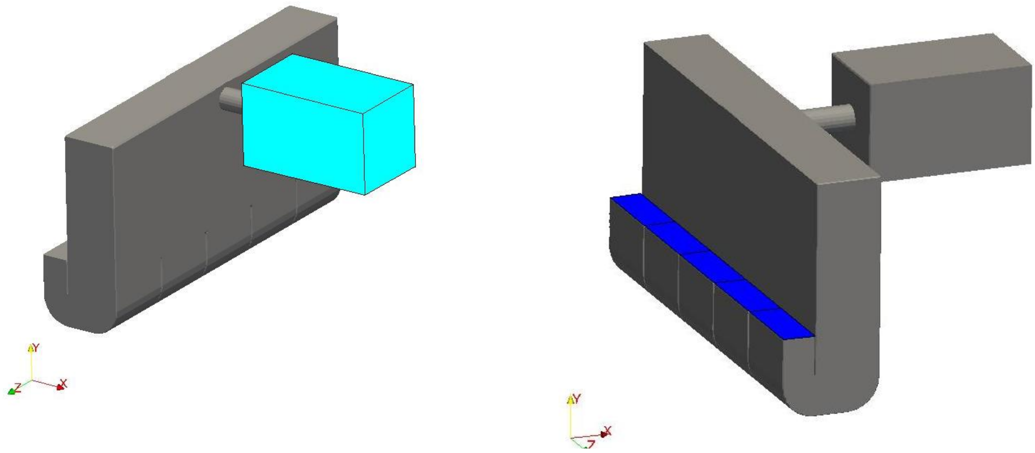

Figure 5: Boundary condition

For what concerns the pressure initial and boundary condition it has been considered the pressure without the hydrostatic component (p-ρgh) instead of the total pressure.

For the air inlet/outlet surface were applied boundary condition on velocity and pressure (p-ρgh) of derived type, this type of condition present in OpenFoam code has the characteristic of stabilizing the calculation.

Boundary condition for U on air inlet/outlet surface (pressure_Inlet_Outlet_Velocity):

- Outlet flow:
  Zero gradient for normal component
  Zero gradient for tangential component
- Inlet flow:
  Zero gradient for tangential component

Boundary condition for p-ρgh on air inlet/outlet surface (total_Pressure)

- Outlet flow:
  Fixed pressure
- Inlet flow:
  Pressure calculated from a fixed value and velocity value

The boundary condition for U on sea water inlet/outlet surface is the same fixed on air inlet/outlet surface (pressure_Inlet_Outlet_Velocity); for the p-ρgh the boundary condition is uniform_Total_Pressure, in this case the pressure can be variable with the time.

The sea motion is simulated as a sinusoidal wave and it has been applied as a variable pressure on the sea water inlet/outlet surface.

In this case the water level varies in a sinusoidal manner around an average value equal to B1+B7 (Figure 1) with an oscillation equal to +/-$H_0/2$ and with a period T_0.

The pressure value (p-ρgh) varies around a mean value equal to

$$p\text{-}\rho gh_{medio}=p_{atm}\text{-}(\rho_{water}g(B1+B7)) \qquad (11)$$

with a variation

$$\Delta(p\text{-}\rho gh)=+/\text{-}(p_{atm}\text{-}(\rho_{water}gH_0/2) \qquad (11)$$

In Figure 6 is represented the pressure (p-ρgh) applied on sea water inlet/outlet surface in the case of the simulation with the first wave type.

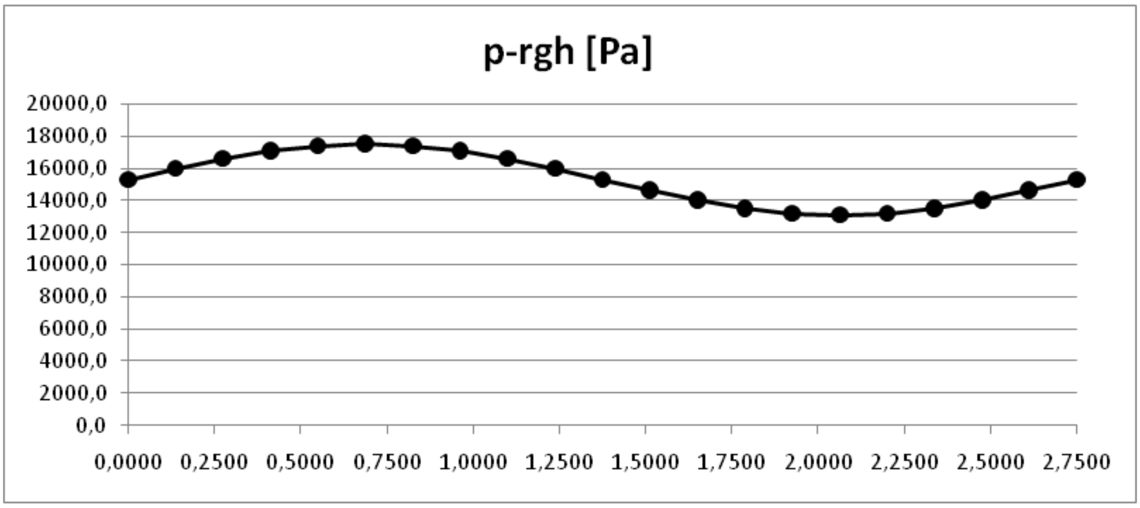

Figure 6: Pressure value for the first wave type

In Figure 7 is represented the pressure (p-ρgh) applied on sea water inlet/outlet surface in the case of the simulation with the second wave type.

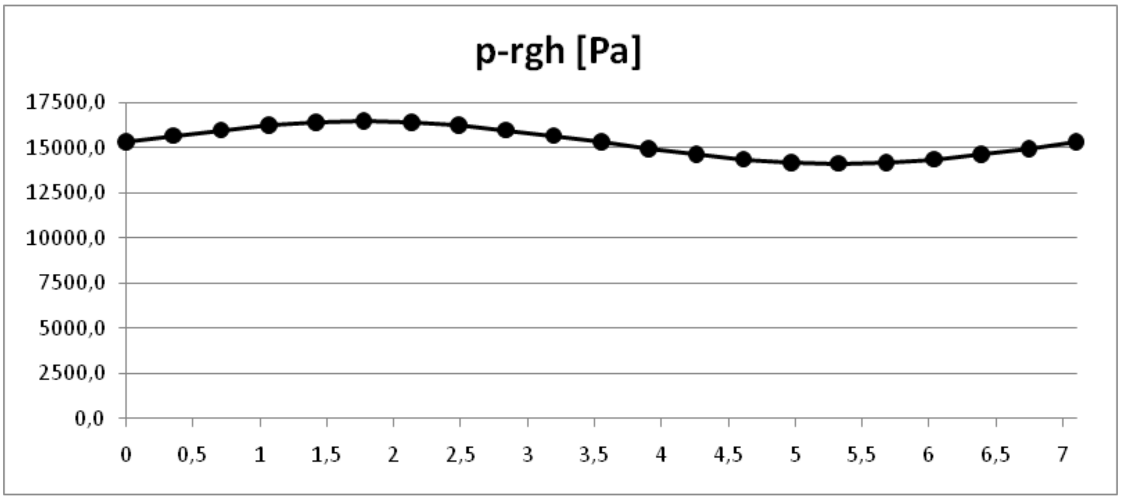

Figure 7: Pressure value for the second wave type

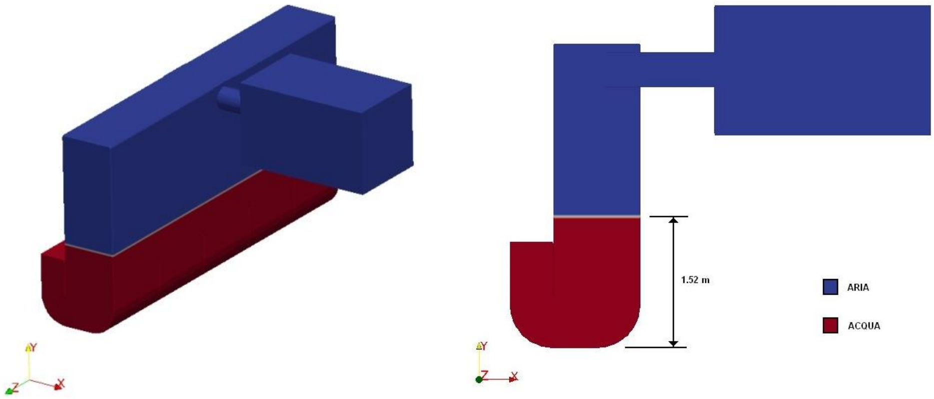

Figure 8: Water level at t=0s

In Figure 8 is represented the water level (1.52m) inside the device at t=0 s.

4 SIMULATION RESULTS

Following are the results obtained from the CFD simulation

4.1 First type of wave

In Figure 9 is represented the velocity at the center of the turbine duct and at 0.5m from the outlet. For the simulation it has been considered the seventh wave period, so from t=16.5 to t=19.25.

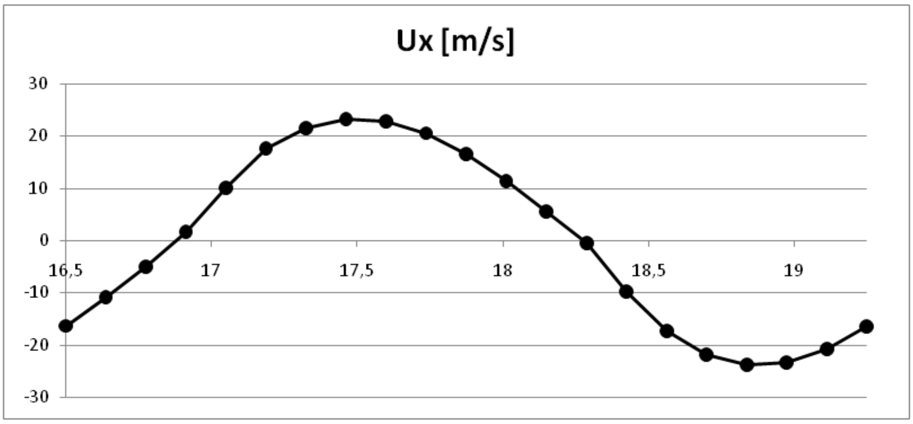

Figure 9: Velocity on the center of turbine duct

In Figure 10a) it is possible to see the water level, in Figure 10b) the water velocity and in Figure 11a) the air velocity in a vertical plane (plane x-y) corresponding to the center of the turbine duct; in Figure 11b)it is possible to note the air velocity in a horizontal plane (plane x-z) corresponding to the center of the turbine duct at time equal to 17.875s.

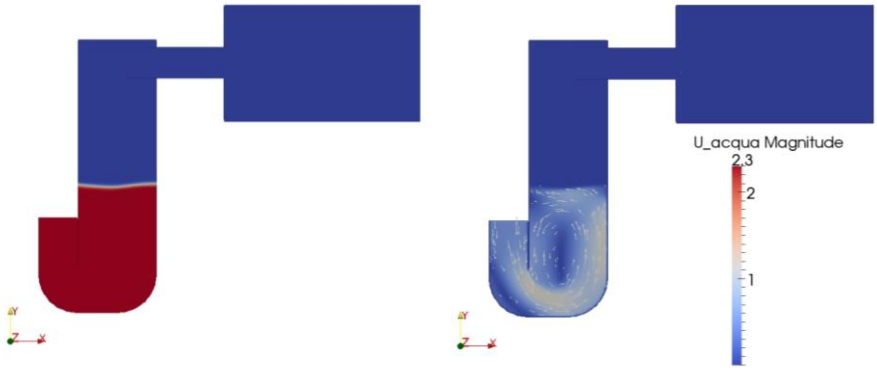

Figure 10: a)Water level at t=17.875 b)water velocity at t=17.875 (Sez.x-y)

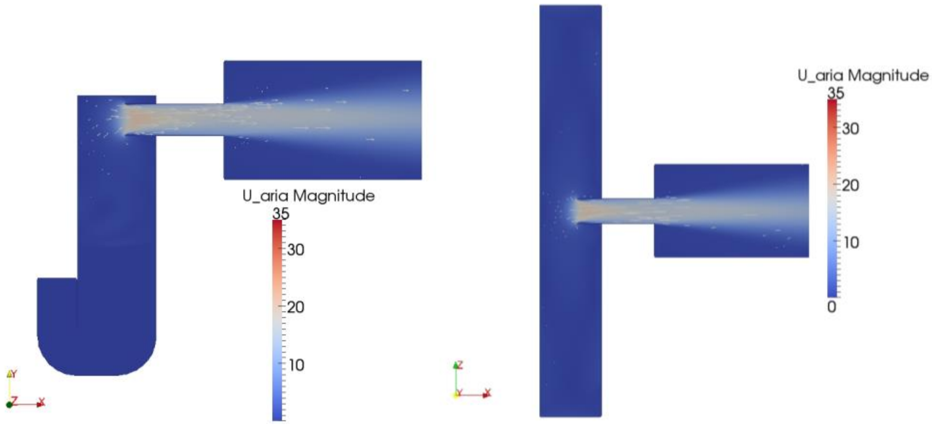

Figure 11: a)Air velocity at t=17.875(sez x-y) b)Air velocity at t=17.875 (Sez.x-z)

4.2 Second type of wave

In Figure 12 is represented the velocity at the center of the turbine duct and at 0.5m from the outlet. For the simulation it has been considered the seventh wave period, so from t=42.6s to t=49.7s.

In Figure 103a) it is possible to see the water level, in Figure 103b) the water velocity and in Figure 114a) the air velocity in a vertical plane (plane x-y) corresponding to the center of the turbine duct; in Figure 114b)it is possible to note the air velocity in a horizontal plane (plane x-z) corresponding to the center of the turbine duct at time equal to 46.15s.

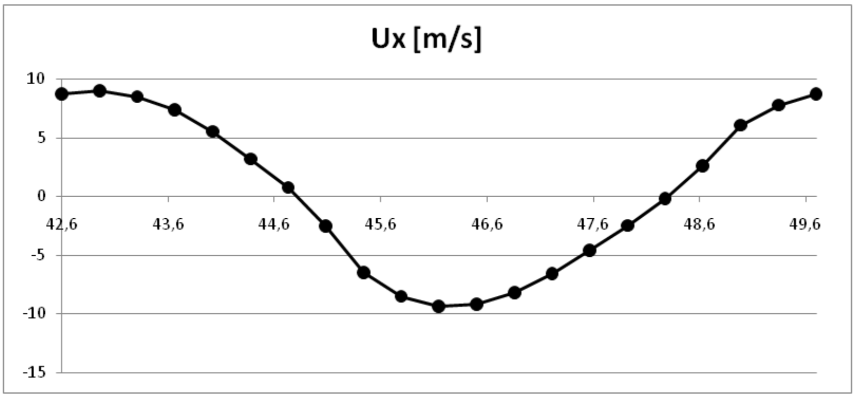

Figure 12: Velocity on the center of turbine duct

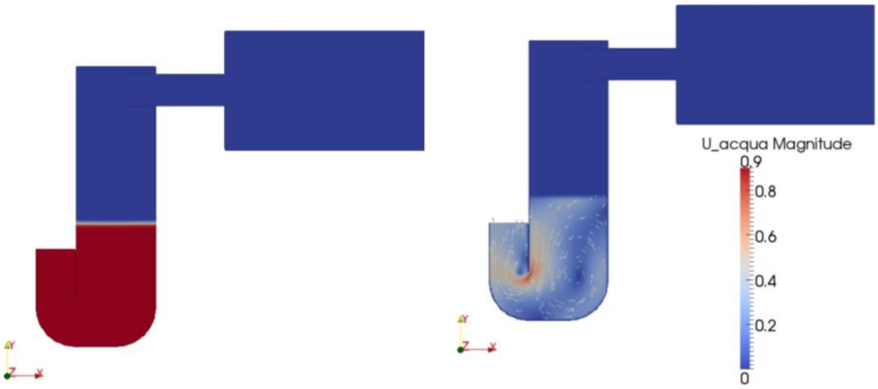

Figure 13: a)Water level at t=46.15s b)water velocity at t=46.15s (Sez.x-y)

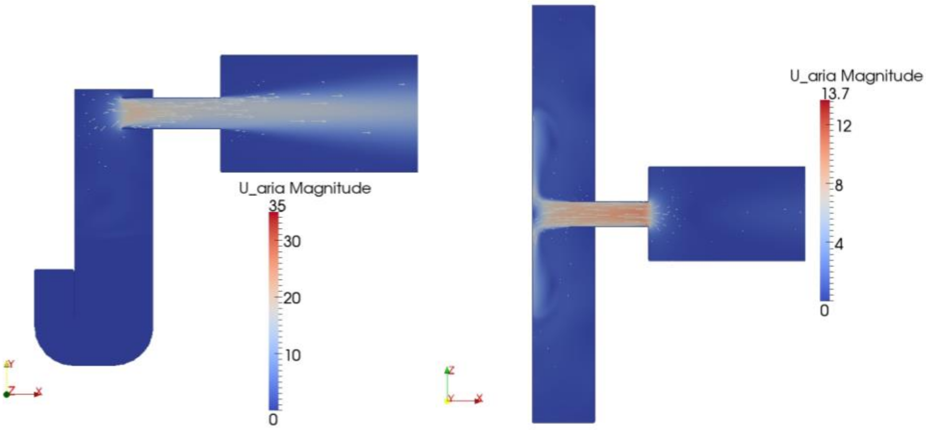

Figure 14: a)Air velocity at t=46.15s (sez x-y) b)Air velocity at t=46.15s (Sez.x-z)

5 CONCLUSION

The realization of the model was quite challenging because of the complexity of the physical phenomena, the device dimension and the flow velocity, so it was necessary to use relevant computing power, in particular were used for each calculation 256 processors.

The computing resources and the related technical support used for this work have been provided by CRESCO/ENEAGRID High Performance Computing infrastructure and its staff; see http://www.cresco.enea.it for information. CRESCO/ENEAGRID High Performance Computing infrastructure is funded by ENEA and by national and European research programs.

It was been developed a simulation using as input a sinusoidal wave and it has been applied as a variable pressure on the sea water inlet/outlet surface.

REFERENCES

[1] Open∇FOAM, The Open Source CFD Toolbox, User Guide, Version 2.2.0, 22nd February 2013.

[2] Open∇FOAM, The Open Source CFD Toolbox, Programmer's Guide, Version 2.2.0, 22nd February 2013.

[3] Open∇FOAM, The open source CFD toolbox, Foundation Training, ESI-OpenCFD, Notes v2.1.1 rev5 12/11/2012

[4] Open∇FOAM, The open source CFD toolbox, Advanced Training, ESI-OpenCFD, Notes v2.1.1 rev5 12/11/2012.

[5] P. Boccotti, "Gli impianti REWEC", Editoriale BIOS - 2004.

[6] P. Boccotti, p. Filianotti, V. Fiamma, F. Arena, "Caisson breakwater embodying an OWC with a small opening – Part II, A small-scale field experiment", Ocean Engineering 34 (2007) (820-841).

[7] R. B. Bird, W. E. Stewart, E. N. Lightfoot, "Fenomeni di trasporto", Casa Editrice Ambrosiana – Milano (1979).

[8] A.A. V.V., "Fondamenti di Termofluidodinamica Computazionale" a cura di Gianni Comini, Servizi Grafici Editoriali – Padova (2004).

[9] T. Crescenzi, D. Nicolini, A. Fontanella, L. Sipione, "Sviluppo di un modello numerico per simulazioni CFD di sistemi di conversione del moto ondoso tipo U-OWC (Oscillating Water Column)", Report Ricerca di Sistema Elettrico, Report RdS/2013/230, Settembre 2013.

[10] D. Nicolini, A. Fontanella, E. Giovannini, "Analisi fluidodinamica CFD su dispositivi a colonna d'acqua oscillante OWC – Fase 1 : Messa a punto del modello CFD sulla base della geometria definitiva dell'apparato U-OWC sperimentale", Report Ricerca di Sistema Elettrico, Report RdS/PAR2013/173, Settembre 2014.

[11] D. Nicolini, A. Fontanella, E. Giovannini, "Analisi fluidodinamica CFD su dispositivi a colonna d'acqua oscillante OWC – Fase 2 : Confronto tra risultati numerici e risultati sperimentali", Report Ricerca di Sistema Elettrico, Report RdS/PAR2013/174, Settembre 2014.