VI International Conference on Computational Methods in Marine Engineering
MARINE 2015
F. Salvatore, R. Broglia and R. Muscari (Eds)

OPTIMIZATION OF THE WAVESAX DEVICE: NUMERICAL MODELLING AND OCEAN WAVE BASIN TESTS

AGATE G.*, AMICARELLI A.*, DANELLI A.* AND PEVIANI M.*

* Department SFE (Sustainable Development and Energy Sources)
Ricerca sul Sistema Energetico - RSE SpA
Via Rubattino, 54 – 20134 Milan, Italy
e-mail: giordano.agate@rse-web.it, web page: http://www.rse-web.it

Key words: Computational Methods, Marine Engineering, Marine Energy, Renewable Energy from Sea, Wave Energy Converter, Oscillating Water Column

Abstract. The Wavesax device has been conceived to be installed in ports and harbours, in the Mediterranean sea. Therefore, two aspects are quite important: flexibility of the device to fit in different structural configurations and replication in a large number of units.

Preliminary numerical modelling of the fixed component of the device has been performed using Computational Fluid Dynamics analysis (RANS-CFD model) and considering four regular wave conditions typical of the Mediterranean sea.

Main issues to be considered in the first stage scale modelling analysis are the effective functionality of the device conception (scale 1:20), the optimization of the design and position of the device in terms of generated velocity gradients in the working section where the turbine blades are installed. The main parameters to be investigated, both with numerical modelling and ocean wave basin tests, are the pressure field in different sections of the device, water levels for different wave conditions and device sinking.

Following the scale model test, the numerical model was calibrated and validated. The paper presents the results of the numerical simulations related to different configurations of the device, under typical Mediterranean wave climates.

1. INTRODUCTION

Recently, several Italian studies in the energy sector, have focused on the evaluation of available marine energy along the Italian coasts. The considered system for the wave energy converter is a modified oscillating water column device, where the working section of the turbine is under the water level.

The present paper shows the results of the studies addressed to verify the potential of a prototype wave energy converter, in different wave conditions; in addition the profile of the device has been optimized studying two different configurations.

The prototype of the device in scale 1:20 was constructed with the aim of verifying the performance of the device under different configurations. Furthermore, the scale model studies have been performed at the hydraulic tank of the HMRC - Hydraulics and Marine Research Centre (Cork, Ireland). Finally the computational fluid dynamics analysis of the fixed components of the reference device scaled (1:20), through the use of a RANS-CFD model (Reynolds Averaged Navier-Stokes - Computational Fluid Dynamics), has been performed with the purpose of calibrating the numerical model and comparing the results with three typical cases chosen from the scale model tests. The validation is necessary for the next phase, not illustrated here, where the numerical analysis of the device with Wells turbine installed at the working section will be performed.

2. PRELIMINARY NUMERICAL MODELING OF THE FIXED COMPONENTS OF THE DEVICE

The considered system for the prototype of a wave energy converter is a modified oscillating water column device, where the working section of the turbine is under the water level. Computational fluid dynamics analysis, of the fixed component of the device, has been performed using a RANS-CFD model and considering four regular wave conditions, representative of the wave climate in the Port of Civitavecchia (Tyrrhenian cost, Italy) where the device can be tested in the near future. The numerical results permitted to estimate the yearly available energy along the working section of the device. This value represents a preliminary (and overestimated) quantification of the producible energy. In addition, the numerical analysis on a reference geometry configuration, allowed quantifying the available energy as a function of the cut-in velocity, which represents a threshold for the fluid velocity, under which no energy is actually produced.

The Figure 1 shows a scheme of the device considered and a sample incoming wave.

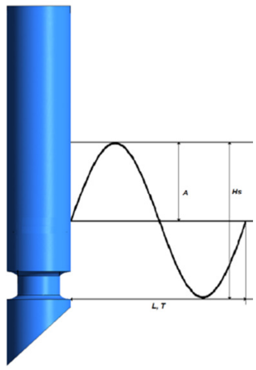


Figure 1: Fixed component of the device. Geometry of reference configuration.

To evaluate the annual available energy along the working section of the fixed component of a device, it is important to define the wave climate conditions that represents the site where the device will be positioned. In order to analyse the performance of the device, four wave classes has been defined in term of (H) height, (T) period and occurrence, as it is reported in Table 1.

The computational fluid dynamics analysis has been performed using a RANS-CFD model for bi-phase flux (water-air, software ANSYS-CFX [6]).

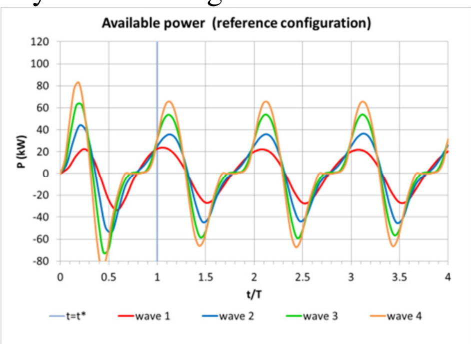
The Figure 2 (left) shows the evolution in time of the available power along four periods, for each one of the selected wave classes. Knowing the occurrences of each wave condition and the calculated power in the working section of the device (Figure 2), the available annual energy is evaluated in about 85MWh, with an equivalent mean power of about 9,7kW (Table 1).

Table 1: Wave characteristics and available annual energy (null cut-in velocity).

Wave type	Hrange	T	Occurrence	Available Power	Annual Energy
	(m)	(s)	(%)	(kW)	(MWh)
0	< 0.5	0.0	44.3	0.0	0.0
1 (H=1m)	0.5 - 1.5	6.0	44.5	15.9	62.0
2 (H=2m)	1.5 - 2.5	7.5	8.7	22.2	16.9
3 (H=3m)	2.5 - 3.5	9.0	2.2	26.9	5.2
4 (H=4m)	> 3.5	10.5	0.3	29.5	0.7
total	-		100.0	-	84.8

The described results are referred to the available theoretical power, without considering yet the efficiency of the turbine and generator to install in the working section.

From the curve of the annual energy as function of the cut-in velocity (Figure 2 right) it is possible to observe that no significant decrease of energy occurs for cut-in velocity less than 1.0m/s. The turbine to use for the reference device must be characterized with a cut-in velocity value not higher than 1.5m/s.



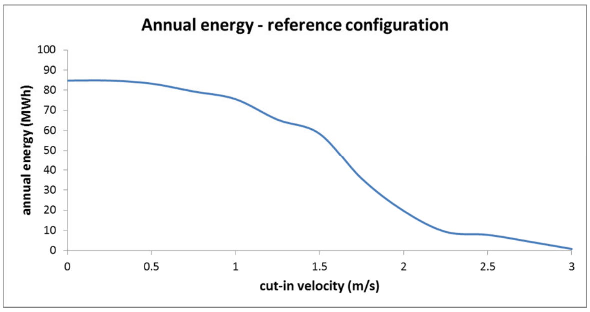


Figure 2: Time evolution of the available power in the reference solution (left) yearly available energy as function of the cut-in velocity (right).

For the optimization phase, starting from the reference solution, two alternative configurations have been analysed with the aim of reducing the loss of energy caused by the turbulence, optimizing the profile of the cylinder in the working section and the mouth at the water inlet section of the device. In Figure 3 the two considered configurations are showed: the configuration 1 (Wavetube, left in figure), close to the reference configuration, presents an optimized hydrodynamic shape and the intake cut with a inclined plane at 45°. While the configuration 2 (Wavesax, right in Figure 3) consider the inlet mouth enlarged with a vertical oval section. This configuration has been thought to facilitate the entrance of water, direct toward the generation section, reducing the energy losses due to the turbulence.

The Figure 4 and the Figure 5 show a time sequence of the pressure and velocity fields related to the wave class 1 for both the configurations.

The velocity module field in the working section is uniform with mean value of about 1.1m/s and maximum value of about 3.9m/s. The vortex formation, that causes local losses due to the irregularity of the tube profile, in the working section detected in the reference configuration, here are considerably reduced.

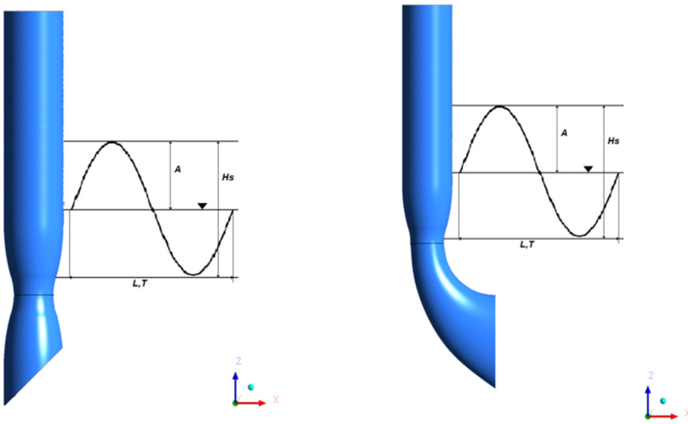


Figure 3: Device optimisation. Configuration 1 (left) and configuration 2 (right).

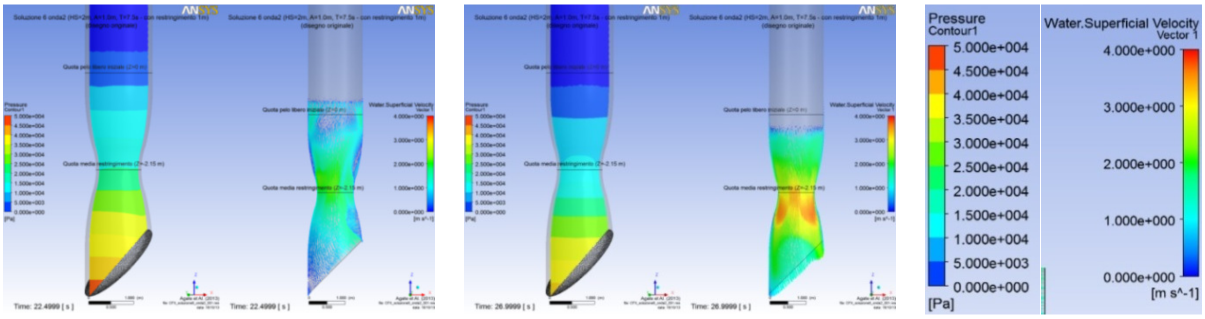


Figure 4: Configuration 1. Pressure field (left on every image) and velocity field (right on every image); vertical sections passing through the axis of symmetry of the inlet section at different time (t=22.5s and 27.0 s).

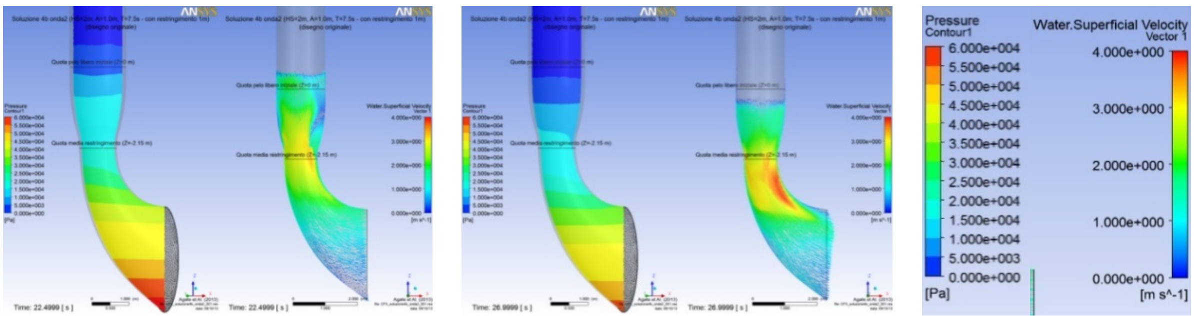


Figure 5: Configuration 2. Pressure field (left on every image) and velocity field (right on every image); vertical sections passing through the axis of symmetry of the inlet section at different time (t=22.5s and 27.0 s).

The time evolution of the available power (red line), of its potential term (blue line) and of its kinetic term (green line) for the two configurations, are shown in Figure 6. The available mean power, considering the cut-in velocity equal to zero, is reported in Table 2.

Following this optimization stage of the fluid dynamics analysis, the configuration 2 allows to maximize the available power in the working section of the device, and at the same time to have higher velocity. This configuration shows also the best performance, considering the cut-in velocity equal to 1m/s and 2 m/s, with respect the other analysed configurations.

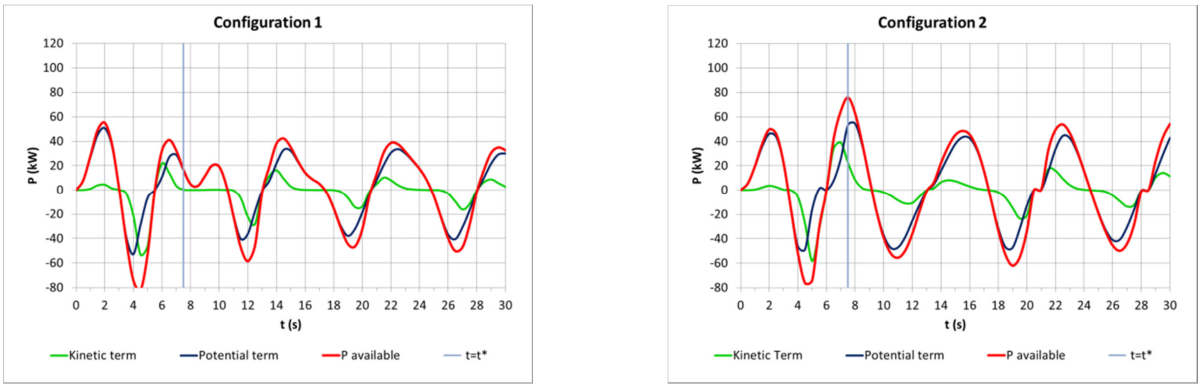


Figure 6: Available power time evolution (red line), the potential term (blue line) and the kinetic term (green line). Configuration 1 (left) and configuration 2 (right).

Table 2: Available power at the working section (cut-in velocity=0 m/s).

Configuration $V_{cut\text{-}in}$=0 m/s	available Power	available Power / ref	kinetic term	potential term	Velocity ave	Velocity max	Power down	Power up
	(kW)	(%)	(kW)	(kW)	(m/s)	(m/s)	(kW)	(kW)
reference	22.2	100.0	3.2	19.0	0.9	2.9	10.5	11.7
1	25.1	113.1	5.2	20.0	1.0	4.1	12.2	12.9
2	34.0	153.1	6.7	27.3	1.1	3.9	16.4	17.6

3. LABORATORY TESTS

In this stage of analysis the major issues to be considered are (a) the effective functionality of the device conception; (b) the optimization of its design in terms of velocity gradients in the working section of the turbine, both under regular and irregular wave conditions.

The present chapter describes the tests of the device (scale 1:20), with regular and irregular wave conditions, carried out at the ocean wave basin of the HMRC (Hydraulics and Marine Research Center) in Cork, Ireland. The studies of this stage will be also interfaced with numerical simulations (RANS-CFD model) of the device under the same wave conditions.

The device and the wave basin are equipped with a series of pressure and level sensors which are able to continuously monitor the performance of the device.

Figure 7 shows the images of the scaled prototype of the device in the configuration 1 (Wavetube) and configuration 2 (Wavesax) and their location in the wave basin during the tests. In total, 37 different tests have been carried out at wave basin, changing the wave climate conditions and/or the device configuration and positioning, taking into account the following parameters: (a) type of mouth for the device (configuration 1 and 2); (b) wave boundary conditions generated in the basin; (c) impounded energy losses at the working section of the turbine; (d) angle between the wave direction and the device mouth; (e) submergence of the device respect to the free surface (m.s.l).

Figure 7: Laboratory tests (scale 1:20): Wavetube (up left) and Wavesax (up right).
Device in the basin during the tests and position of the level and pressure sensors (down).

Starting from the measured data, it was possible to calculate the velocities inside the device, the corresponding available power and the Response Amplitude Operator (RAO). The RAO is defined as the ratio between the water motion amplitude inside the device and the wave amplitude in the basin.

Figure 8 and Figure 9 show two examples of the results obtained for the configuration 2, comparing different sets of tests changing only one parameter at a time. In particular, the Figure 8 show the variation of the RAO and the available power in the working section, for different submergence of the device, considering an incoming wave of height 2m and period 7,5 sec. While the Figure 9 presents the variation of the RAO and the available power in the working section, for different wave periods of incoming 2m height waves.

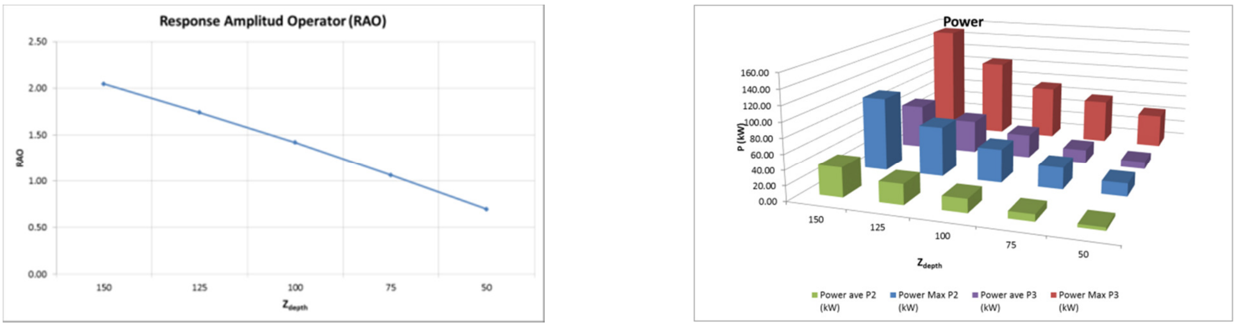


Figure 8: Laboratory tests (scale 1:20):
Response Amplitude Operator (RAO) (left) and available power at the working section (right), considering different submergence, under the same wave conditions (H=1 2m and T=7,5 sec).

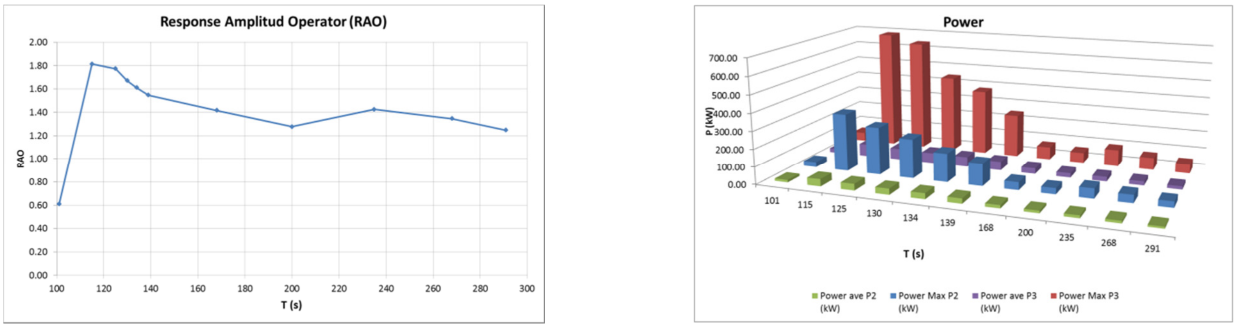


Figure 9: Laboratory tests (scale 1:20):
Response Amplitude Operator (RAO) (left) and available power at the working section (right), considering different wave periods (for a constant H=2m).

4. COMPUTATIONAL FLUID DYNAMICS ANALISYS OF THE DEVICE: CALIBRATION AND VALIDATION

The next step of the study concerns the calibration and validation of the numerical model of the device for the configuration 2 (Wavesax). The activity regards the following aspects:

- scaling (1:20) of the geometry considered in the numerical model in the preliminary optimization analysis (Figure 10, left);
- perform several runs varying the physical and numerical parameters of the model (turbulence model, wall roughness and time integration scheme);
- validation carried out comparing numerical results and measures during the lab tests (the four pressure and the velocity inside the device) (Figure 10, right).

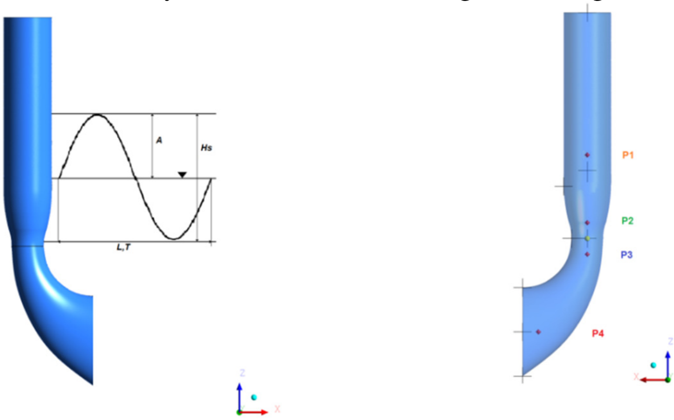


Figure 10: Geometry. Design and wave schematization (left) and monitor point position (right).

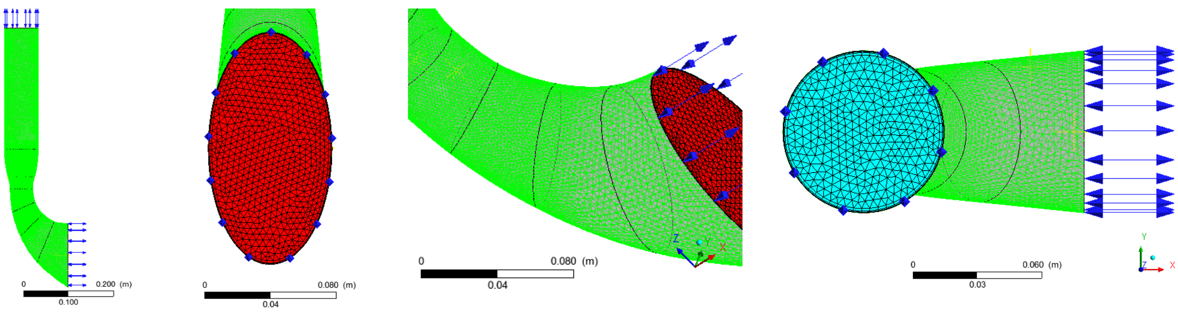


Figure 11: Grid. Lateral view, inlet mouth detail, working section view and top view.

Figure 11 shows some details of the computational grid used for the simulations for the configuration 2.

To perform the calibration and validation, three different wave conditions have been simulated: Run 1 (H=100mm; T=1,68s); Run 2 (H=200mm; T=1,34s); Run 3 (H=150mm; T=2,0s).

The best accuracy has been obtained for the SST turbulence model of Menter, with the same cpu time required. Moreover, a high resolution model of turbulence is used and the imposed wall roughness is equal to ε=0.0005m (0.01/scale). Finally, the second order time integration scheme has been used.

Figure 12 shows the pressure and velocity profiles, where it is possible to see an acceptable agreement, with particular concern to Run 1 and Run 3.

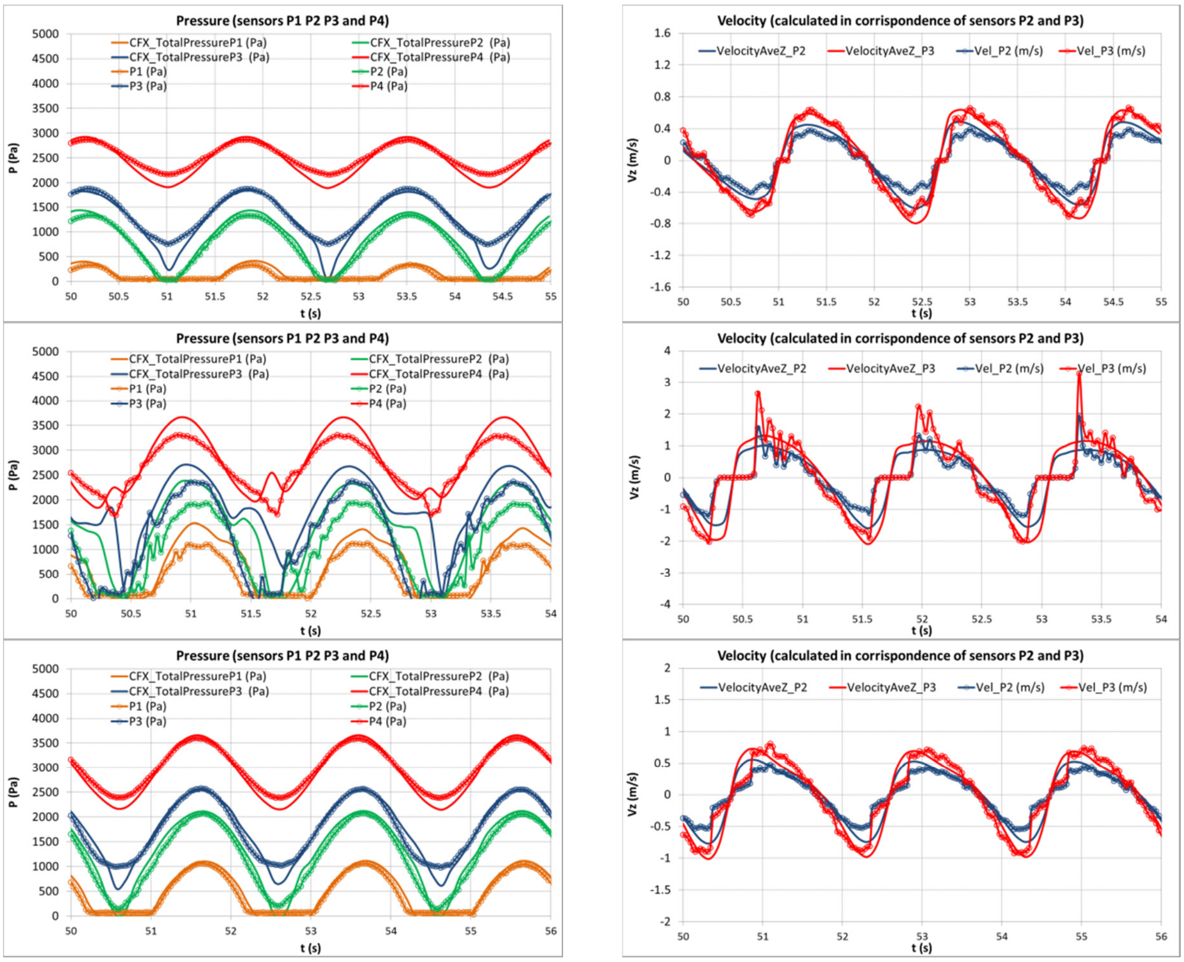


Figure 12: RANS-CFD model calibration and validation: pressure profiles (left) and velocity profiles (right); Run1 (up), Run2 (center) and Run3 (down).

5. CONCLUSIONS

- The paper presents the results of the study performed in order to verify the functionality of a device for converting wave energy into electricity, under different sea conditions. In addition, the profile of the device has been optimized studying two different configurations.

- From the laboratory tests in scale 1:20, the device performance was satisfactory, both in terms of response to incident wave conditions and quality of the measured data. Essential information on key parameters has been clearly understood, with results leading to a quite important step forward in the development of the Wavesax device.
- The computational fluid dynamics modelling of three typical cases from the scale model tests (1:20) has led to calibration and validation of the RANS-CFD model.
- The results of the numerical simulation permitted to assess the profit in terms of available power at the take-off section, considering three different intake configurations. In particular, the configuration 2 (Wavesax) showed to be the best solution, with available power ratio about 1,53 compared to the reference configuration.

REFERENCES

[1] Peviani M., F. Carli, S. Bonamano. *European wave energy and studies for Italy's potential*. International Journal on Hydropower and Dams, Vol 18 - issue 5 (2011).

[2] Peviani M., S. Scanu, F. Carli. *Valutazione per lo studio di fattibilità di un dispositivo di generazione di energia elettrica dal moto ondoso.* RSE Report 12001443 (2012).

[3] Agate G., A. Amicarelli, M. Peviani. *Studi indirizzati allo sviluppo di sistemi innovativi per la generazione di energia dal moto ondoso.* RSE Report 13001033 (2013).

[4] Agate G., A. Amicarelli, M. Peviani. *Analisi fluidodinamica di un prototipo per la conversione di energia da moto ondoso: ottimizzazione della componente fissa e stime preliminari di potenza assorbita con la girante.* RSE Report 14001669 (2014).

[5] Agate G., A. Amicarelli, R. Guandalini, G. Stella. "*Mappe di producibilità energetica dal moto ondoso e dalle correnti marine dei mari italiani*" – Report RSE 12000352 (2012).

[6] ANSYS-CFX ver 14.0 User's Manual. ANSYS, Inc. Southpointe 275 Technology Drive, Canonsburg PA 15317 USA ANSYS-CFX release 14.0 http://www.ansys.com