16èmes Journées de l'Hydrodynamique

27-29 novembre 2018 - Marseille

CENTRALE MARSEILLE

irphé Institut de Recherche sur les Phénomènes Hors Equilibre

Impact of Hydrokinetic turbines on the near sedimentary bed by using an Euler Two-phase modelling CFD approach.

F. Khaled[1], S. Guillou[1], Y. Mear[1,2] , F. Hadri[3]

fatima.khaled@unicaen.fr ; sylvain.guillou@unicaen.fr ; yann.mear@lecnam.net *; ferhat.hadri@uvsq.fr*

[1] Normandie Université, UNICAEN LUSAC, EA 4253, 60 rue Max Pol Fouchet, CS 20082, 50130 Cherbourg

[2] INTECHMER, Bd de Collignon, 50110, Tourlaville

[3] LISV, Université de Versailles, 78140 Vélizy

Résumé

La nécessité de déterminer l'impact sur l'environnement de la turbine hydro-cinétique a conduit à plusieurs études. Cependant, peu d'entre elles ont pris en compte les effets locaux sur le lit de sédiment mobile. L'objectif de la présente contribution est de modéliser l'impact de la présence d'une turbine à axe horizontal avec l'écoulement et un lit de sédiment mobile. L'approche de modélisation Euler-Euler à deux phases est utilisée en association avec l'approche de représentation de la turbine des éléments de pales (BEMT). Les résultats fournissent l'impact sur la morphologie du lit dû à la présence de la turbine. L'érosion est accentuée sous la turbine en raison de l'accélération de l'écoulement et de l'augmentation de la contrainte de cisaillement locale sur les sédiments.

Summary

The need of determining the environmental impact of hydrokinetic turbine has led to several studies. However, few of them have taken into account the local effects on mobile sediment bed. The aim of the present contribution is to model the impact of the presence of horizontal-axis turbine with the flow and a mobile sediment bed. A modelling framework is derived to predict the scour induced by the turbine installed on fluvial erodible sandy bed surfaces, such as the Eulerian multiphase model for sediments and blade element momentum theory for turbine. Results provide the impact on the bed morphology due to the presence of the turbine, scouring capabilities are well enhanced below the turbine due to the acceleration of the flow and increasing of local shear stress of sediments.

I. Introduction

Interest in electricity generation from low carbon technologies and renewable energy has become recently the most discussed and the forefront concern, due to the need to reduce the excessive use of fossil fuels known to be responsible for climate change. For that, the need of sustainable, predictable energy generation has led in recent years to the development of Stream Turbines (ST) and has become a near reality, to reach 20% from the total energy production in France on 2020. Stream turbines will be exploited in regions of high currents either marine or river resources, as Alderney Race, Race of Barfleur, The Rhône, … HydroQuest is one of the riverine ST industries, it aims to install a farm of 39 turbines in The Rhône river and produces 6700 MGh per year. The future potential energy of riverine turbines by 2025 should be 3000 MW [1].

Determining the impact of ST on their environment is becoming a significant interest nowadays. Regional modeling with a representation of the turbines' array with bed friction or momentum sink approach were proposed to quantify the impacts far away from the turbines. Sanchez and al. [2] have applied a 3D numerical model in Ria de Ortigueira to study the potential flow changes due to the operation of a steam turbines' farm. It was found that the resulting transient flow modifications were concentrated in the area occupied by and next to the farm, with nearly negligible effects outside the inner Ria. Fairley et al. [3] studied the cumulative impact of ST on sediment transport in the Pentland Firth and have concluded that the array implementation of ST only has minimal effect on the baseline morphodynamics of the large sandbanks in the region.

One of the less discussed and most important impact is the effect of turbines on the sediment transport and the bed evolution close to the turbines. The energy extracted by the turbine will alter the hydrodynamics of the stream by increasing or reducing the friction in areas around the turbines. Close to the turbine, scour phenomena and deposit zones could appear as shown by a 3D full scale modelling of one turbine. This effect could be enhanced by a turbines' farm. Hill et al. [4] have shown experimentally that the presence of the turbine and the rotation of the blades impact the bed morphology. Therefore, the interaction between ST and the bed morphology deserves more studies.

The present contribution investigates the impact of the deployment of riverine turbines on the evolution of the sandy bed close to the turbine by CFD. As fully resolved CFD (Computational Fluid Dynamics) simulation of ST farm is time consuming, a simpler approach is proposed.

The section II presents the methods, following by some elements of validation in section 3. Then the case of the interaction between a turbine and a mobile bed is considered in section 4.

II. Modelling Methodology

The modelling is based on the combination of the 3D CFD model able to simulate the sediment transport and the morphology evolution in one hand and on the introduction of the effect of the turbine in the other hand. The model implemented is based on the use of a two-phase flow Euler-Euler model to simulate the flow and sediment motion (Chauchat and Guillou [5] , Bonamy et al.[6], Zhen Cheng et al. [7]) and the development of a methodology to represent the turbine in the model based on the use of the Blade Element Momentum Theory (BEMT) (Malki et al.[8], Shives et al.[9]). The CFD open source library OpenFoam has been used with the SedFoam code [6] which is based on twoPhaseEulerFoam code, and modified to introduce the momentum source code of the turbine.

II.1. <u>Hydrokinetic turbine modelling</u>

The momentum theory is often used to predict the power output, the thrust and the efficiency of a hydrokinetic turbine. The model is a hybrid analytical, BEM, and CFD computational model. First, the hybrid analytical 3D BEM CFD models the turbine to compute the effect of energy extracted from turbine to fluid in the near wake.

The blade element theory is considered as a more advanced analysis of the hydrodynamic behavior of hydrokinetic turbine in addition to the analysis of energy extraction process examined by the actuator disk model. It allows considering the effects of rotor geometry characteristics like chord and twist distributions of the blade airfoil. The blade is separated into radial sections since each of the blade elements has a different rotational speed and geometric characteristics hence experiencing a slightly different flow (Figure 1). Moreover, the forces and moments are determined in each element so that total forces and moments are calculated by integrating the individual forces and moments on each element. In the present study, the Blade Element Momentum (BEM) theory is adopted as the main computation method and it is applied over a disc that represents the turbine with radius r, thickness e and number of blades n. The method is a combination of momentum theory and blade element theory.

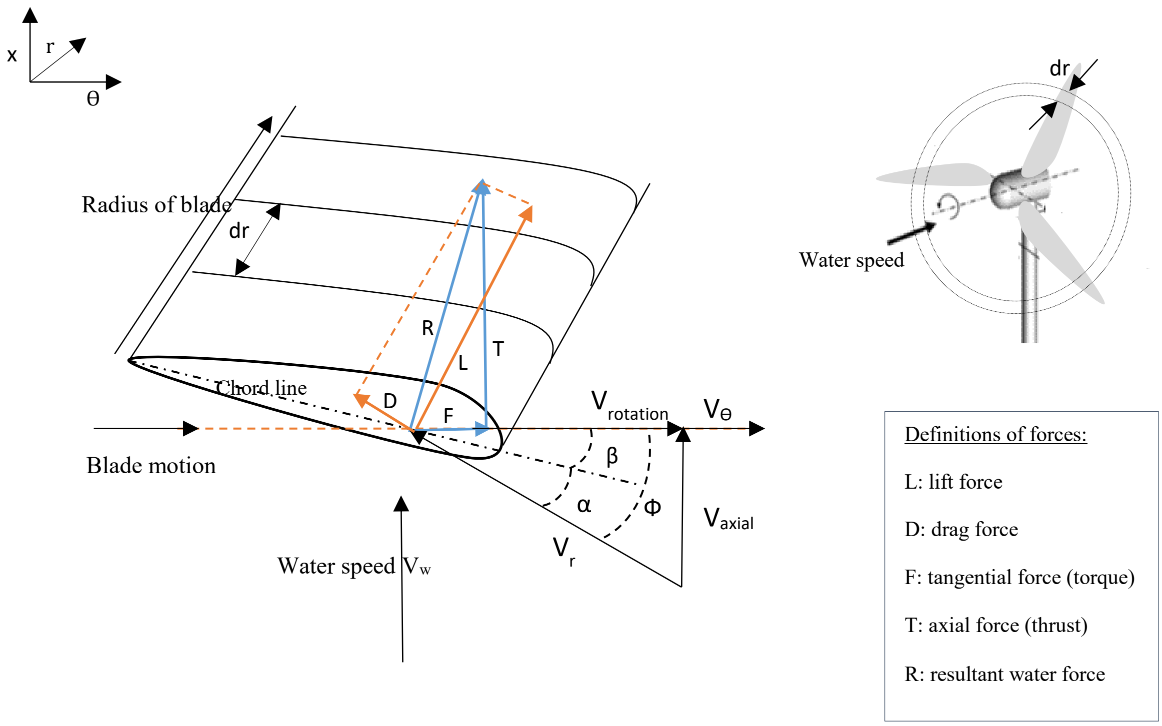

Figure 1:Flow velocities and forces acting on the blade by the blade element momentum theory. V_{axial}, V_{θ} are the axial and tangential velocity components, β is the blade twist angle, α the angle of attack, Φ the inflow angle

The blade lift and drag forces are imposed on the fluid using the momentum source term S_i in equation (7). The lift force acts perpendicular to the relative velocity, while the drag acts parallel:

$$L = \frac{1}{2}\rho c V_{rel}^2 C_L F_{tip}$$
$$D = \frac{1}{2}\rho c V_{rel}^2 C_D \quad (1)$$

where c is the chord of the blade which varies in function of radius, C_L and C_D are lift and drag coefficient respectively, their values depend on the geometry of the blade (angle of attack), V_{rel} is the relative velocity of the water flow such as:

$$V_{rel} = \sqrt{V_{axial}^2 + (r\omega - V_\theta)^2} \quad (2)$$

The correction F_{tip} Prandtl tip-loss factor is necessary because the simple approach does not resolve the individual blades and associated tip-vortices and thus tends to over-predict the lift near the blade tips.

$$F_{tip} = \frac{2}{\pi}\cos^{-1}(\exp(-\frac{n(R-r)}{2r\sin\emptyset})) \quad (3)$$

The lift and drag forces are rotated into the rotor's cylindrical coordinate system to obtain axial and tangential force components:

$$f_x = L\cos\emptyset + D\sin\emptyset$$
$$f_\theta = -D\cos\emptyset + L\sin\emptyset \quad (4)$$

The momentum sources are computed based on the time-averaged blade forces (per unit cell volume) imparted by the blade onto the fluid as it traverses through the water.The term source S_i is projected with reference to cylindrical coordinate system, S_x and S_Θ, such as:

$$S_x = \frac{nf_x}{2\pi re}$$
$$S_\theta = \frac{nf_\theta}{2\pi re} \quad (5)$$

The final source Si in the Cartesian coordinate system is obtained by a projection of the cylindrical source terms. The total thrust T and power P is calculated by numerical integration of the sources over the disc region.

II.2. Transport of sediments

The mathematical formulations of the Eulerian two-phase flow model is obtained by averaging the local and instantaneous mass and momentum conservation equations for fluid and particles phases [7, 10, and 11]. In the present model, the turbulence time averaged Eulerian two-phase flow equations described in Chauchat et al. [5] and implemented in the open source finite volume CFD library.

This model show encouraging results concerning the suspended-load transport mainly by integrating the influence of the sediment particles on the fluid turbulence and the collisions between particles.

The mass conservation equations for the particle phase and fluid phase are written as:

$$\frac{\partial \emptyset}{\partial t} + \frac{\partial \emptyset u_i^s}{\partial x_i} = 0$$

$$\frac{\partial (1-\emptyset)}{\partial t} + \frac{\partial (1-\emptyset) u_i^f}{\partial x_i} = 0, \quad (6)$$

where $\emptyset$, and 1-$\emptyset$ are the particle and fluid volume fractions, u_i^s, u_i^f are the sediment and fluid phase velocities, and the index i=1, 2, 3 represents the streamwise, spanwise and vertical component, respectively. The momentum equations are written for each phase. The index *k* designs the phase (‘f ‘for the fluid phase, ‘s’ for the solid phase)

$$\frac{\partial \rho^s \phi u_i^s}{\partial t} + \frac{\partial \rho^s \phi u_j^s u_i^s}{\partial x_j} = -\phi \frac{\partial p}{\partial x_i} + \phi f_i - \frac{\partial p^s}{\partial x_i} + \frac{\partial \tau_{ij}^s}{\partial x} + \phi \rho^s g_i + \phi(1-\phi) K\left(u_i^f - u_i^s\right) - \frac{1}{S_c}(1-\phi) K v_t^f \frac{\partial \phi}{\partial x_i} + S_i$$

$$\frac{\partial \rho^f (1-\phi) u_i^f}{\partial t} + \frac{\partial \rho^f (1-\phi) u_j^f u_i^f}{\partial x_j} = -(1-\phi) \frac{\partial p}{\partial x_i} + (1-\phi) f_i + \frac{\partial \tau_{ij}^f}{\partial x_j} + (1-\phi) \rho^f g_i - \phi(1-\phi) K\left(u_i^f - u_i^s\right) + \frac{1}{S_c}(1-\phi) K v_t^f \frac{\partial \phi}{\partial x_i} + S_i \quad (7)$$

Where ρ^s, ρ^f are the particle and the fluid density, g_i is the gravitational acceleration and p is the fluid pressure. τ_{ij}^f is the fluid stress (viscous stress and fluid Reynolds stresses), p^s is the normal stress of particles and τ_{ij}^s their shear stress. The terms K and S_c are the drag parameter and Schmidt number respectively of particles. S_i represents the turbine’s effects as described previously.

The turbulence model k-ε described by Cheng et al. [7] from Hsu et al. [11] has been used.

Validation of the models

a- **Transport of sediments modelling**

The model is implemented numerically and it is validated by an experience realized in LMSGC (Pham Van Bang et al. [12]) consisting of suspension of polystyrene beads (d = 290 µm, ρs = 1050 kg/m3) of volume fraction 0.48, in silicone oil (ρ_f = 950 kg/m3). The numerical results of sedimentation process during the time is validated with experimental measurements, such as the time of formation of interfaces between particles and fluid phases.

b- Turbine modelling

Since this study is interested on the local effect of the turbine on the sediment bed, so it is important to validate the BEMT in the near wake in terms of efforts and energy extraction source terms. The experimental measurements of Mycek et al. [13] has been used. The experiments were performed in the IFREMER's experimental flume tank, which has a length of L=18m, a width of l = 4m and a depth of h=2m. Figure shows the tidal turbine used in the experiments. The rotor of diameter d = 0.7m is connected to a 0.72 m long cylindrical hub of diameter 0.092 m. The chosen case has the following characteristics showed in figure 2:

Description: IFREMER - LOMC

- Profile : NACA 63418 ;
- Flow Velocity : $U = 0.8\ m.s^{-1}$;
- Tip speed ratio : $TSR = \omega d/2U = 3.67$;
- Reynolds number: $Re = Ud/2\nu = 280\ 000$;
- Turbulence intensity: I = 3%;
- Pitch (set angle) : 0°;
- Sense of rotation: counter-clockwise.

Figure 2: Picture of the tidal turbine used during the experiments (Mycek et al. [13])

The geometry and mesh are created using ICEM CFD software. Structured hexahedral cells were used to mesh the entire domain, which comprises of approximately 1.3 million cells. The rotor domain mesh was of structured hexahedral mesh used for CFD-BEMT with diameter d=0.7m. The computational domain is a 4m x 18m x 2m block. The disc is located at 11D downstream the inlet boundary to avoid interactions with this inlet. This lets 14D downstream to observe the development of the wake. As in the experimental case, an ambient turbulence intensity rate I_{∞}=3% is adopted. The mesh of areas in interest is refined to concentrate accuracy. The hub is represented as a solid cylinder in the center of the disc with radius 0.046m. The blades are modelled using 23 discrete NACA 63418 elements along their span in accordance with the detailed blade profile description given in [13]. The averaged value of the incident flow velocity $0.8\ m.s^{-1}$ is considered.

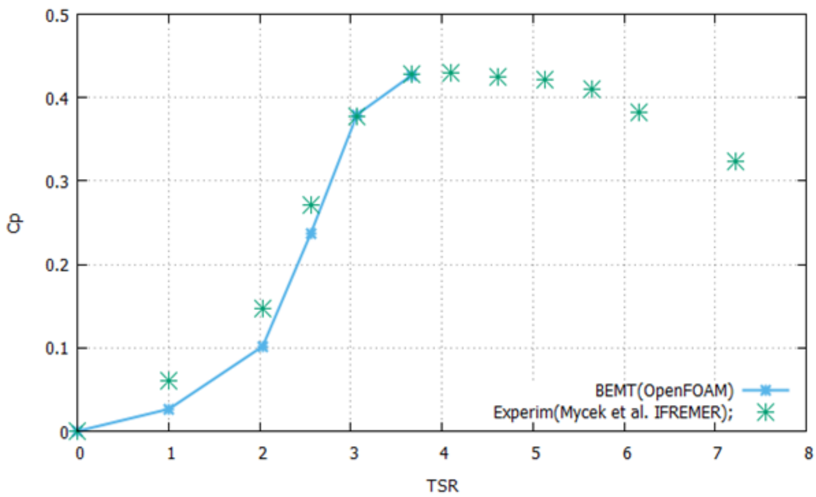

Figure 3: Variation of Power coefficient in function of TSR factor

According to figure (3), a good agreement is observed on the C_p evolution for TSR values from zero to 3.67. A better agreement is obtained as the TSR increases until 3.67. This could be explained regarding the tidal turbine geometry (no mast modelling) so the vortices generated on the tip of blades and on the hub are taken into account. Since we consider from the beginning a simple correction on the tip, while in literature there exist other corrections for BEMT that might be considered. For TSR=3.67, the absolute errors according to experiment measurements are 0.3% for C_P.

III. Applications Case Study

An attempt has been made to simulate the effect of stream turbine on the transport of contouring sediments in a river; a first approach consists on considering a unique type of sediments.

a- Mesh configuration

The mesh is created using OpenFOAM package, it is configured as 11700000 hexahedral mesh, which captures a rectangular domain with, dimensions 1m x 0.35m x 0.1m, in x, y and z directions. The mesh is refined on the bottom of the domain where a sediment sheet of thickness equal to 0.05m, and volume fraction of 0.61, is imposed.

b- Initial and boundary conditions

The initial conditions and inlet condition for velocity is set to a non-uniform logarithmic profile. Neumann boundary condition is applied to the bottom and hydrostatic pressure for outlet boundary. The top and sides boundaries are set to symmetry planes. The turbulent kinetic energy initial condition and inlet is set to non-uniform logarithmic profile, while the turbulent kinetic energy dissipation rate inlet is set to 0.00397 $m^2.s^{-3}$ for this case. The turbulent kinetic energy is set such as the turbulence intensity is 3% at the inlet.

c- Turbulence and transport properties of sediments

The model is calibrated with medium-diameter of sands ($d = 0.25$ mm, $\rho^S = 2100$ $kg.m^{-3}$) in the sheet flow condition. The kinetic theory factors (equation) are $e = 0.8$, $Fr = 0.05$ and $p = 5$. The applied model on the particles for viscosity and conductivity is Syamlal [11], for the granular pressure is Lun [11]. The limit volume fraction is set to 0.635. The used drag model for both faces is Gidaspow Schiller Naumann [11]. The turbulent suspension of particles is taken into consideration, since the unsteady turbulent RAS model is used. For that purpose, a very small step of iterations is imposed to reach the convergence.

Since this work aims to illustrate the effect of the turbine, so the same geometrical configuration of the last case is created for the incoming simulation in terms of dimensions and directions of the entire domain. Thereby, what makes the difference in the mesh configuration is the presence of a disc that represents the tidal turbine. The same mesh properties is applied in this case with refer to section above.

The angular velocity of the turbine is 400 rpm, the blades are modelled using 23 discrete NACA 63418 elements along their span in accordance with the detailed blade profile description given in [13].

d- Primary Results and discussions

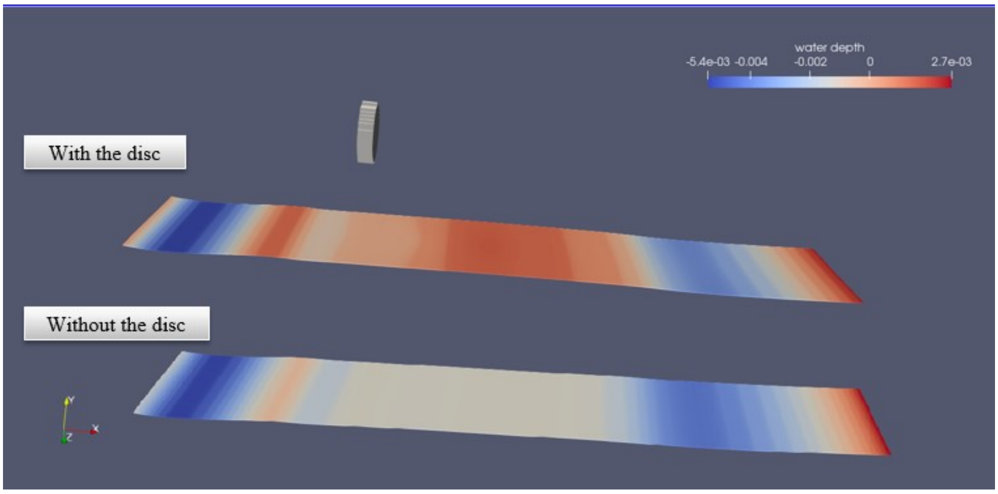

Figure 4: Representation of the bathymetry of the bottom, contouring of volume fraction of sediments (0.58)

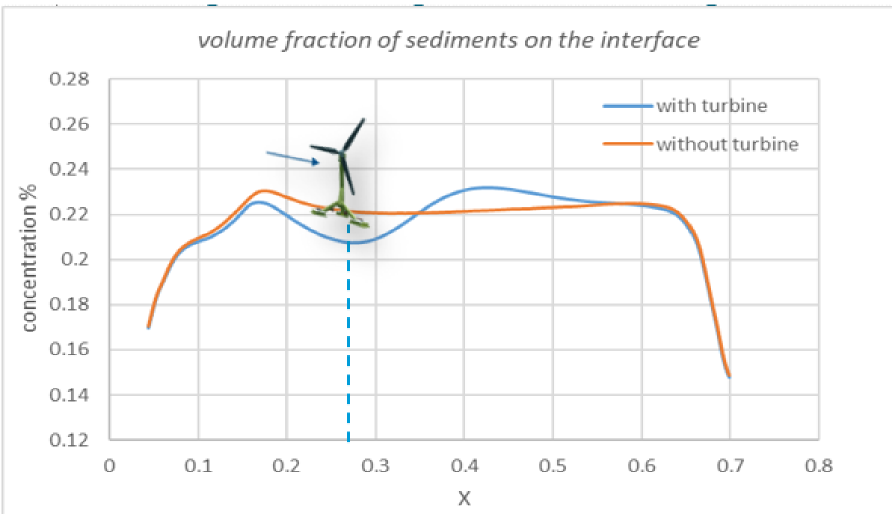

Figure 4: Evolution of volume fraction of sediments downstream the turbine

Figures 4 and 5 show the changes of the sediment bed at the streamwise channel centerline throughout the simulation, with and without the rotor. Several changes in bed morphology have been found in the presence of the rotor. The clearest feature is the scour in the near wake area. Scouring occurs immediately after the rotor. An increase in bed shear stress just below the turbine is observed, which explain that the presence of turbine increases the local shear stress directly downstream of its location and greatly enhances scour below the turbine and deposition downstream. The experiments of Hill et al. also showed similar scour.

The difference in concentration values is due to the local flow acceleration resulting from flow constriction between the rotor bottom tip and the approaching dune crest. This acceleration increases local shear stresses and transport capacity, thereby decreasing the dune crest elevation below the turbine.

In particular, the deposition of the sediments in the far wake area shown in figures, and found by [4] would be an important factor for the installation of downstream turbine arrays.

IV. Conclusion and perspectives

The research program carried out during this work aim to couple the 3D Euler multi-phase model CFD approach to a BEMT model of an axial-flow three-bladed hydrokinetic turbine to study the interactions between axial-flow hydrokinetic turbines and sediment transport of sandy bed. A validation of the different part of the model has been done. Finally, a first study of the erosion of a bottom substrate has been conducted. It shows the interest of the method to study such a case. More precisely the turbine amplifies the scour phenomenon especially when compared to non-rotor case.

The future work will provide parameter study to quantify the scour phenomenon created downstream the turbine.

Acknowledgments

The authors acknowledge CRIANN (Centre Régional Informatique et d’Application Numériques de Normandie) co-financed by the Normandy Region, the State and the European Union, for the access to the Computation means and to the Manche Country Concil for their founding.

References

[1] http://www.hydroquest.net/

[2] Sánchez, Marcos & Carballo, R & Ramos, V & Iglesias, Gregorio. (2014). Tidal stream energy impact on the transient and residual flow in an estuary: A 3D analysis. Applied Energy. 116. 167–177. 10.1016/j.apenergy.2013.11.052.

[3] Fairley I., Masters I., Karunarathna H. (2015), The cumulative impact of tidal stream turbine arrays on sediment transport in the Pentland Firth, Renewable Energy, Volume 80, Pages 755-769.

[4] Craig Hill, Mirko Musa, Michele Guala, Interaction between instream axial flow hydrokinetic turbines and uni-directional flow bedforms, Renewable Energy, Volume 86, 2016, Pages 409-421

[5] Chauchat, J., and S. Guillou (2008), On turbulence closures for two-phase sediment-laden flow models, J. Geophys. Res., 113, C11017, doi: 10.1029/2007JC004708.

[6] Bonamy, Cyrille & Julien, Chauchat & Cheng, Zhen & Nagel, Tim. (2017). SEDFOAM, A OPENFOAM SOLVER FOR SEDIMENT TRANSPORT.

[7] Cheng, Zhen & Hsu, Tian-Jian & Calantoni, Joseph. (2017). SedFoam: A multi-dimensional Eulerian two-phase model for sediment transport and its application to momentary bed failure. Coastal Engineering. 119. 32-50. 10.1016/j.coastaleng.2016.08.007.

[8] R. Malki, A.J. Williams, T.N. Croft, M. Togneri, I. Masters, A coupled blade element momentum – Computational fluid dynamics model for evaluating tidal stream turbine performance, Applied Mathematical Modelling, Volume 37, Issue 5, 2013, Pages 3006-3020, ISSN 0307-904X.

[9] Shives, Michael & Crawford, Curran, 2016."Adapted two-equation turbulence closures for actuator disk RANS simulations of wind & tidal turbine wakes," Renewable Energy, Elsevier, vol. 92(C), pages 273-292.

[10] Barbry N., Guillou S. et Nguyen K.D., Une approche diphasique pour le calcul du transport sédimentaire en milieux estuariens, Comptes Rendus de l'Académie des Sciences, Série II.b, 328 (11), 793-799 (2000).

[11] Hsu, T.-J., J. T. Jenkins, and P. L.-F. Liu (2003), On two-phase sediment transport: Dilute flow, J. Geophys. Res., 108, 3057, doi: 10.1029/2001JC001276, C3.

[12] Pham van bang, Damien & Lefrançois, Emmanuel & Sergent, Philippe & Bertrand, F. (2008). MRI experimental and finite elements modelling of the sedimentation-consolidation of mud. La Houille Blanche. 3.

[13] Mycek Paul, Gaurier Benoit, Germain Gregory, Pinon Gregory, Rivoalen Elie (2014).Experimental study of the turbulence intensity effects on marine current turbines behaviour. Part I: One single turbine. Renewable Energy, 66, 729-746.