Tidal currents characterization with Large Eddy Simulation

Adrien Bourgoin*, Sylvain Guillou*, Riadh Ata†
Jérôme Thiébot* and Sofiane Benhamadouche†
*Normandie Université, UNICAEN LUSAC, EA 4253, 60 rue Max Pol Fouchet, CS 20082, 50130 Cherbourg-en-Cotentin, France
E-mail: adrien.bourgoin@unicaen.fr, sylvain.guillou@unicaen.fr, jerome.thiebot@unicaen.fr
†LNHE/MFEE, EDF R&D, 6 quai Watier, 78400 Chatou, France
E-mail: adrien-externe.bourgoin@edf.fr, riadh.ata@edf.fr, sofiane.benhamadouche@edf.fr

***Abstract*—The hydrodynamics of tidal energy sites is generally characterized by rough seabed, as shown in Figure 1. Such configuration favours the formation of powerful turbulent flow structures that strongly affect both the performance, and the lifespan of the turbine installations. To optimize the placement of the turbines, the effect of turbulent flows has to be understood, thus requiring a detailed description of the physical processes of turbulence. As part of the THYMOTE project, a Large-Eddy-Simulation (LES) approach is being developed which aims at computing the unsteady aspect of turbulent flows. LES approaches dedicated to environmental free surface flows are nowadays gaining popularity thank to the increasing power of supercomputers. This paper presents the implementation of a LES approach into the TELEMAC-3D solver, a hydrodynamic free surface flow model included in the open TELEMAC-MASCARET suite of numerical solvers. The TELEMAC-3D model [1] originally includes a Reynolds Averaged Navier-Stokes (RANS) solver dedicated to the simulation of environmental flows. The implementaion of the LES method in TELEMAC-3D consists of adding turbulence models [2] to mimic the effect of smallest motion scales as well as including suitable schemes to reduce numerical dissipation. The implementation of the LES approach to TELEMAC-3D is validated using experimental measurements of a flow above dunes [3]. Future applications of the LES approach will deal with the characterization of currents at the tidal energy site named Raz Blanchard (which is also refered as Alderney Race). We expect that the LES will enable more accurate representation of the real world turbulent phenomena, including inherent turbulence and complex processes of vortex shedding.**

***Index Terms*—Turbulence, numerical modelling, Large-Eddy-Simulation, tidal currents, TELEMAC**

I. Introduction

Tidal currents are nowadays seen as a promising source of renewable energy. The tidal energy sites are often characterized by rocky bottom and/or a rugged bathymetry, as in the figure 1. The conjunction of rough seabed and high current velocities induces the emergence of large and powerful swirling structures. Several measurements, such as those realized in [4] show that the turbulent intensity can reach 24%. These vortexes can strongly influence the performances or the lifetime of the turbines [5]. Understanding thoroughly the hydrodynamics of these sites is thus a prerequisite to the deployment of turbines. By using, inter alia, numerical modelling, the 'THYMOTE' project aims at characterizing the turbulent processes with a Large-Eddy-Simulation (LES) approach. The LES method enables simulating the random aspect of turbulence, which plays an important role in transport phenomena [6]. It consists in solving the Navier-Stokes equations to simulate the greatest scales and in introducing a subgrid model in order to mimick the influence of the smallest motion scales. Although not often used for the simulation of environmental flows, the improvement of computation resources nowadays permits using LES for such applications. In this paper, we present several developments related to the implementation of a LES method in the code TELEMAC-3D [1], which was originally designed for RANS simulations. The developments includes the implementation of a subgrid models such as the Smagorinsky model [2] and of particular boundary conditions generating velocity fluctuations at the inlet of the computation domain. Here, we use the Synthetic-Eddy-Method [7]. The developments are tested using the validation case [3] [8], which represents an open channel flow over two-dimensional dunes.

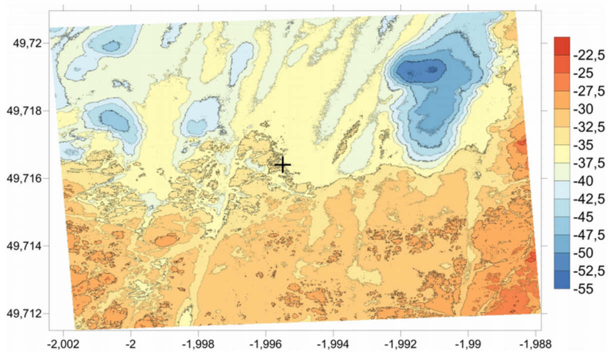

Fig. 1. Bathymetry of the Raz-Blanchard site.

II. TELEMAC-3D

TELEMAC-3D is a CFD code developed by EDF [1] for the simulation of environmental flows. It is able to simulate free surface flows including several natural phenomena such as sediment transport, waves, tide, rain, or wind. The numerical method relies on the finite element method $P1$. In a computational domain Ω, the model solves the non-hydrostatic tridimensional Navier-Stokes equations for an incompressible fluid using prismatic elements (see figure 2):

$$\begin{cases} \nabla.\boldsymbol{u} = 0 \quad \text{in} \quad \Omega \\ \frac{\partial \boldsymbol{u}}{\partial t} + (\boldsymbol{u}.\nabla)\boldsymbol{u} = -\frac{1}{\rho}\nabla p + \nu \Delta \boldsymbol{u} + \boldsymbol{g} + \boldsymbol{F} \quad \text{in} \quad \Omega \end{cases} \tag{1}$$

where $\boldsymbol{u} = (u, v, w)$ are respectively the velocities in the directions (x, y, z), p is the pressure, ν is the kinematic viscosity and F is the source terms.
The moving mesh is discretized vertically using a sigma transformation [9]. Based on a change of variables for the space coordinates, this method allows to simplify the calculation on a fixed grid.

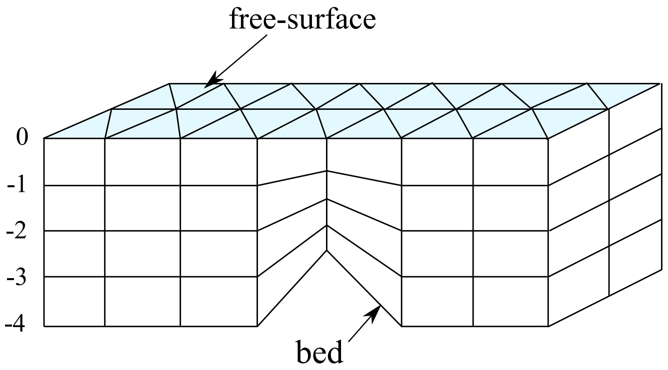

Fig. 2. Classical mesh used with TELEMAC-3D. Source: TELEMAC-3D theory guide.

Using a too dissipative numerical scheme can inhibit the subgrid modelling. High order and non-dissipative schemes are thus required (at least at the second order in space and in time) in order to transport efficiently the flow fluctuations. Several approximations (such as mass-lumped formulations) used in the original version of TELEMAC-3D have been removed so that the fluctuations propagate in the flow. In addition to those efforts in reducing the numerical dissipation, the convection schemes provided in TELEMAC-3D are too dissipative and not suitable for Large-Eddy-Simulation, since they prioritize a conservative and positive behaviour. Thus a centered finite element formulation has been implemented, based on a second order Adams-Bashforth time integration scheme for the convection term and a Crank-Nicholson scheme for the diffusion term.

In the next, the computational domain is defined as Ω, and the free surface and the bottom boundaries are respectively noted Γ_s and Γ_b. The main algorithm for the Navier-Stokes resolution is described below. From a velocity field $\mathbf{u^n}$ and a free surface elevation η^n, the first stage aims at solving the advective and diffusive parts of the Navier-Stokes equations to get the velocity field written $\mathbf{u^d}$.

$$\frac{\mathbf{u^d} - \mathbf{u^n}}{\Delta t} + (\mathbf{u^n}.\nabla)\,\mathbf{u^d} - \nu \Delta \mathbf{u^d} = 0 \quad \text{in} \quad \Omega \tag{2}$$

Then the horizontal velocities are coupled with the free surface elevation and the horizontal velocities is solved to get $(\mathbf{u^e}, \eta^{n+1})$, defined by:

$$\begin{cases} \dfrac{\mathbf{u^e_H} - \mathbf{u^d_H}}{\Delta t} = -g\nabla_H \eta^{n+1} + \mathbf{F_H} \quad \text{in} \quad \Omega \\ \dfrac{\eta^{n+1} - \eta^n}{\Delta t} + \nabla.\displaystyle\int_{b^n}^{\eta^n} \mathbf{u^e_H} dz = 0 \quad \text{on} \quad \Gamma_s \end{cases} \tag{3}$$

where p_h is the hydrostatic part of the pressure, g is the gravity and b^n and η^n are respectively the bottom and the free surface elevation at the time t^n. At least, the correction and continuity step aims at computing the dynamic pressure p_d^{n+1} and the final velocity fields $\mathbf{u^{n+1}}$ by solving

$$\begin{cases} \Delta p_d^{n+1} = -\dfrac{\rho}{\Delta t}\nabla.\mathbf{u^e} \quad \text{in} \quad \Omega \\ \mathbf{u^{n+1}} = \mathbf{u^e} + \dfrac{\Delta t}{\rho}\nabla p_d^{n+1} \quad \text{in} \quad \Omega \end{cases} \tag{4}$$

As regards the bottom friction, TELEMAC-3D uses a wall model to avoid an unaffordable mesh refinement near the bottom. The friction stress is expressed with the formulation:

$$\begin{cases} \tau_{xz} = \mu\dfrac{\partial u}{\partial n} = -\dfrac{u}{2}\rho C_f\sqrt{u^2+v^2} \quad \text{on} \quad \Gamma_b \\ \tau_{yz} = \mu\dfrac{\partial v}{\partial n} = -\dfrac{v}{2}\rho C_f\sqrt{u^2+v^2} \quad \text{on} \quad \Gamma_b \end{cases} \tag{5}$$

where ρ is the density, C_f is a dimensionless friction coefficient and (u, v) are the horizontal flow velocities. Several formulations for the friction coefficient are proposed in TELEMAC-3D. Here, we use a Nikuradse law consisting in computing the coefficient from the bottom roughness size k_s [10] with the formulation:

$$C_f = 2\left(\frac{\kappa}{\log(30\frac{z_{p1}}{k_s})}\right)^2 \tag{6}$$

where z_{p1} is the altitude of the first grid plane and κ is the Von Karman constant.

III. Large-Eddy-Simulation

The concept of Large-Eddy-Simulation is to divide the energy spectrum of the flow in two parts by using a numerical filter. LES permits to compute differently the small and the large scales. Considering that the small turbulent structures have a universal behaviour, they are simulated with a simple model. Conversely, the more complex behaviour of the largest scales is computed by solving the Navier-Stokes equations for the filtered quantities (the largest scales). The interactions between the large and the small scales are simulated by using a subgrid model. Moreover specific boundary conditions have to be prescribed in order to increase the numerical schemes accuracy and reduce the computational costs.

A. Subgrid modelling

The filtered incompressible Navier-Stokes equations read:

$$\left\{ \begin{array}{l} \frac{\partial \tilde{u}_i}{\partial x_i} = 0 \\ \frac{\partial \tilde{u}_u}{\partial t} + \frac{\partial \widetilde{u_i u_j}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \tilde{p}}{\partial x_i} + \nu \frac{\partial}{\partial x_j}\left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i}\right) - \frac{\partial \tau_{ij}}{\partial x_j} \end{array} \right. \tag{7}$$

where ν is the molecular viscosity. The term τ_{ij} is the subgrid tensor, which characterizes the interactions between the resolved scales and the subgrid scales (which is unknown and has to be modeled). The main LES approach, named functional modelling [6], assumes that the interactions between the scales can be described by an energetic process. The anistropic part of subgrid tensor τ_{ij}^a can be written according to this new quantity and the filtered velocity gradients take a Boussinesq assumption-like formulation [11], whereas its istropic components are added in the pressure definition P with

$$\left\{ \begin{array}{l} \tau_{ij}^a = \tau_{ij} - \frac{1}{3}\tau_{kk}\delta_{ij} \\ P = \tilde{p} + \frac{1}{3}\tau_{kk} \end{array} \right. \tag{8}$$

The Smagorinsky model [2] can be considered as the first subgrid model [12]. It consists of evaluating simply the ansitropic subgrid tensor thanks to a subgrid viscosity ν_t and the strain rate tensor norm from the filtered scales, with

$$\tau_{ij}^a = -2\nu_t \tilde{S}_{ij} \quad \text{where} \quad \tilde{S}_{ij} = \frac{1}{2}\left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i}\right) \tag{9}$$

Based on a physical definition, the subgrid viscosity as the product of a characteristic legnth scale l and a characteristic velocity v. The length scale is evaluated directly with the filter width $\tilde{\Delta}$ through a constant C_s (called Smagorinsky constant). Then the velocity is expressed with the Prandlt formulation [13]. It defines the velocity scale v as the strain rate tensor norm $|\tilde{S}|$ times a length scale times a length, which is l is this case. At least the Smagorinsky model reads:

$$\nu_T = (C_s\tilde{\Delta})^2|\tilde{S}| \tag{10}$$

The value of the Smagorinsky constante does not have a unique value in litterature. Its varies between 0.065 and 0.2 depending on the numerical features of Navier-Stokes solvers, as well as the modelled flow.

B. Artificial turbulence

Experimentally and numerically, full development of turbulence can be a prolonged process. To accelerate this transitional phase, artifical turbulence methods allow to introduce flow fluctuations at the inlet of the computational domain.

The Synthetic Eddy Method (SEM) [7] consists in introducing artificial velocity fluctuations (computed from the Reynolds tensor) at the inlet of the computation domain in order to get quickly a fully developed turbulence. N virtual vortex are created in a tridimensional box around the inflow surface having the following dimensions:

$$\left\{ \begin{array}{lcl} x_{j,min} & = & \min\limits_{x\in S, i\in 1,2,3}(x_j - \sigma(x)) \\ x_{j,max} & = & \max\limits_{x\in S, i\in 1,2,3}(x_j + \sigma(x)) \\ \Delta x_j & = & x_{j,max} - x_{j,min} \end{array} \right. \tag{11}$$

where σ is a length scale for the vortexes, defined by:

$$\sigma = \max(\min(\frac{k^{3/2}}{\epsilon}, \kappa\delta), \tilde{\Delta}) \tag{12}$$

with k the turbulent kinetic energy, ϵ the turbulent dissipation rate, κ the Von Karman constant, δ the half water depth and $\tilde{\Delta}$ the filter width of the LES. At the beginning, each artificial vortex is defined by a random position in the virtual box as well as a direction of rotation (in the three dimensions). Then, at each time step, the velocity fluctuations at the inlet are computed from the prescribed Reynolds tensor and the positions of the vortex using the expression:

$$u_i' = \frac{1}{\sqrt{N}}\sum_{k=1}^{N} c_i^k f_\sigma(\boldsymbol{x} - \boldsymbol{x}^k) \tag{13}$$

where f is a shape function defined by:

$$f_\sigma(\boldsymbol{x} - \boldsymbol{x}^k) = \prod_{j=1}^{3}\sqrt{\Delta x_j}\sqrt{\frac{3}{2\sigma}}\left(1 - \frac{|x_j - x_j^k|}{\sigma}\right) \tag{14}$$

and c_i^k is amplitude of the vortex, such as:

$$c_i^k = a_{ij}\epsilon_j^k \tag{15}$$

where $\epsilon_j^k \in \{-1, 1\}$ is the orientation of the kth vortex in the jth dimension. and a_{ij} is the Cholesky decomposition of the prescribed Reynolds stresses tensor $\mathbf{R}$, given by:

$$\begin{pmatrix} \sqrt{R_{11}} & 0 & 0 \\ R_{21}/a_{11} & \sqrt{R_{22} - a_{21}^2} & 0 \\ R_{31}/a_{11} & (R_{32} - a_{21}a_{31})/a_{22} & \sqrt{R_{33} - a_{31}^2 - a_{32}^2} \end{pmatrix} \tag{16}$$

The R_{ij} are the Reynolds stress components, which are the input parameters of this method. The Synthetic Eddy Method has the advantage of introducing a fluctuation field based on a user-defined Reynolds tensor that can be adapted to each flow configuration. At each time step, each vortex is transported in its generation zone with the averaged flow. Of course after a while, these turbulent structures end up leaving their domain. In this case, they are reintroduced upstream with new lateral and vertical coordinates, as well as new random orientations. This method gives very good results provided that a Reynolds tensor is prescribed in accordance with the desired flow. For this, RANS-type modelling can be performed beforehand.

IV. Validation

In this section, the results obtained with TELEMAC-3D are compared with experimental data. Our model is intended to simulate tidal flows, that is why flows over complex bed forms are investigated.

A. *Flow over a dune*

The flow presented here describes a turbulent open channel flow over two-dimensional dunes [3], [8], [14] of height H (see the figure 3). This case has been studied experimentally in [3] in which a train of two-dimensional dunes have been studied with laser Doppler velocimetry measurements. The dune height is $H = 8cm$ and its length is $L = 1.6m$. The maximum flow (at the foot of the dune) depth is $h = 0.294m$, which give a Reynolds number based on the bulk velocity U_b and the maximum flow depth h of approximately 1.15×10^5.

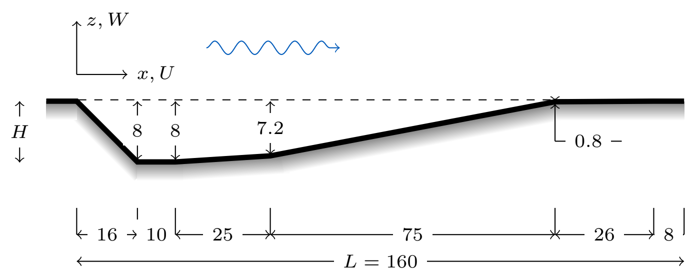

Fig. 3. Concrete dune profile (cm).

A one-dune and $1m$ wide space domain has been modeled with Telemac3D with three grids. The mesh used to discretize this computational domain is composed of about 1.2×10^6 points is used. In wall units the dimensionless grid spacings are $\Delta x^+ = 160$, $\Delta y^+ = 120$ and $\Delta z^+ \approx 100$, which corresponds to suitable scales according to [15] (in which the recommended LES grid spacings are $\Delta x^+ = 100 - 600$, $\Delta y^+ = 100 - 300$ and $\Delta z^+ = 50 - 150$ with the use of a wall model).
. For these configurations the time step is as to have a CFL number of the order of 0.15

As inlet boundary condition, in order to take into account of the previous dunes in the experimental configuration, the mean velocity components of the downstreal vertical profiles (corresponding to the position $x = 1.580m$) are used to define the inflow boundary condition of the numerical model. To prescribe the inflow turbulence, the Synthetic Eddy Method is used, for which the Reynolds stresses components are given by the experiments at this location. Thanks to this method, both the mean flow and the turbulence incoming from the previous dunes of the experimental set up are reproduced at the inlet of our computational domain. Furthermore it requires also the knowledge of the turbulent dissipation rate which is not provided by the experiments. It is evaluated by using a theoritical law [10], which is:

$$\epsilon^+ = \frac{1}{\kappa} \frac{1}{(z^{+4} + 15^4)^{1/4}} \qquad (17)$$

where ϵ^+ and z^+ are respectively the normalized turbulent dissipation rate and the normalized distance to the solid walls, and $\kappa = 0.41$ is the Von Karman constant. Thanks to this method, both the mean flow and the turbulence incoming from the previous dunes of the experimental set up are reproduced at the inlet of our computational domain.

In [3], some sand is placed above the dune with a roughness size of $k_s = 1.6mm$. Thanks to the Nikuradse law provided by TELEMAC-3D, an affordable and suitable boundary condition can be prescribed at the bottom. Indeed using a no slip boundary condition would require a considerable mesh refinement over the vertical near the bottom ($\Delta z^+ < 1$ according to [15]). Large-Eddy-Simulation results are compared to these experiments at four positions of measurements, defined by the locations $x = 6cm$, $x = 21cm$, $x = 43cm$ and $x = 127cm$ and which we will denote respectively L_i hereafter.

The averaged streamwise and vertical velocities(normalized with the bulk velocity) obtained with TELEMAC-3D is compared to the experimental data from [3] in gigure 4. The agreement with experiments is satisfactory despite a sligh overestimation of the vertical velocity at the location $L2$.

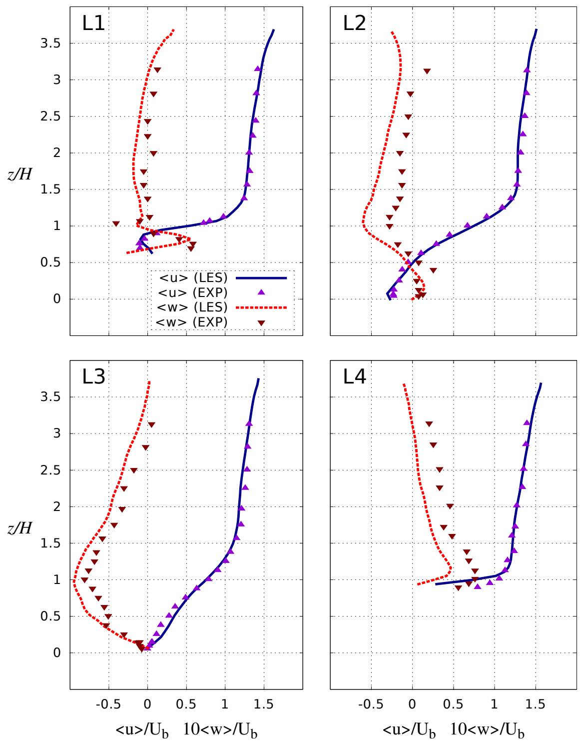

Fig. 4. Comparison of averaged streamwise velicity and vertical velocity obtained with LES and the experiments from [3].

Figure 5 presents the comparison of the vertical profiles of the turbulent kinetic energy k as well as the Reynolds stress $< u'^2 >$, which have approximately the same magnitude. The agreements are here excellent since both the altitude of the peak of energy and its amplitude are well reproduced. Profiles of the Reynolds stresses $< w'^2 >$ and $< u'w' >$ are shown

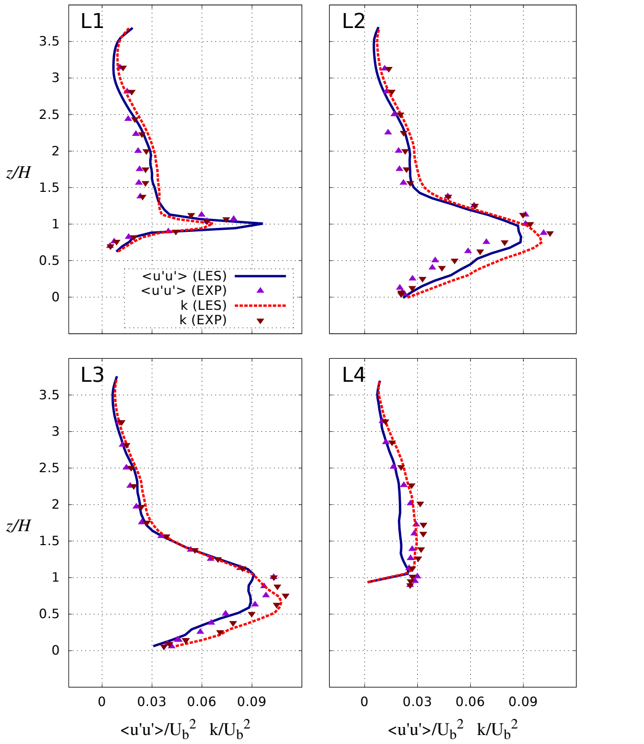

Fig. 5. Comparison of vertical profiles of the turbulence kinetic energy k and the Reynolds stress $< u'^2 >$ obtained with LES and the experiments from [3].

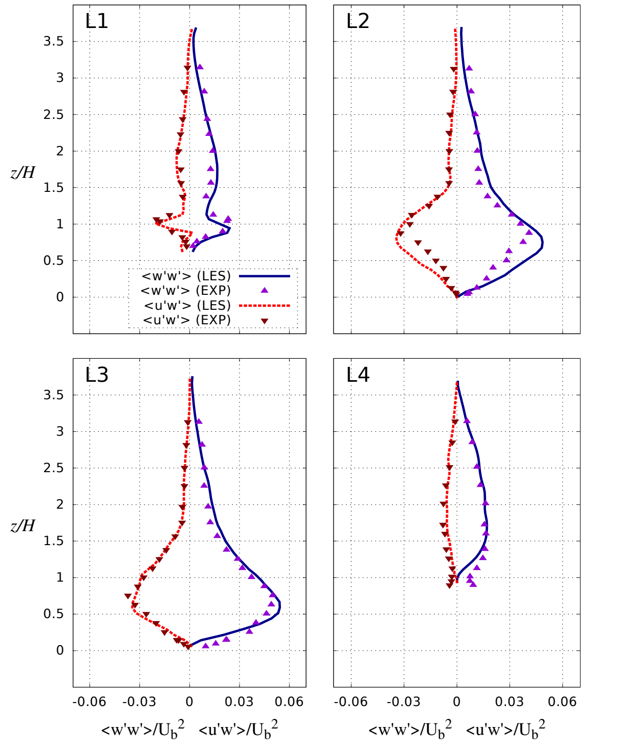

Fig. 6. Comparison of vertical profiles of the Reynolds stresses $< w'^2 >$ and $< u'w' >$ obtained with LES and the experiments from [3].

in figure 6. Here again the agreement of LES results and experimental data is very good at the four vertical axis.

The good prediction of the flow statistics allows to investigate its instantaneous behaviour. Figure 7 represents the iso-surfaces of the λ_2 criterium [16] which allows to identify the turbulence structures. This figure highlights the birth of Hairpin vortexes [17], which are transported in the flow with an inclined shape. It triggers the development of spanwise vortices in the separated shear layer that ascend up to the free surface. The overall agreement between the model results and the measurements allows to validate the developments. Using LES now permits to analyse the dynamics of the turbulent structures. Methods presented in this paper will be used for modelling tidal domains. Such application has already been simulated with a RANS approach in Telemac3D [18]. However, for applying LES, smaller time steps and much greater numbers of points will be required to preserve the filtering concept. This kind of simulation will require high performance simulations on supercomputers as in [19].

V. Prospective application

Before using these methods to simulate real tidal flow, a channel flow with the same characteristics is investigated. Therefore, a computational domain of length $5km$, of width $1km$ a maximum water depth of $H = 40m$ and a bulk velocity of $U_b = 3ms^{-1}$ has been defined, with a bottom composed of dunes with the same shape of the validation test case described previsouly. These dunes have a length of $L_d = 100m$ and a height of $h_d = 5m$, located all along the channel. This flow has been discretized over 40 horizontal planes, each one composed of $5m$ triangles, resulting in about 20×10^6 points. Considering the Reynolds number (which is $Re \approx 10^8$ based on the bulk velocity and the water depth), the recommanded grid spacings in [15] are obvisouly

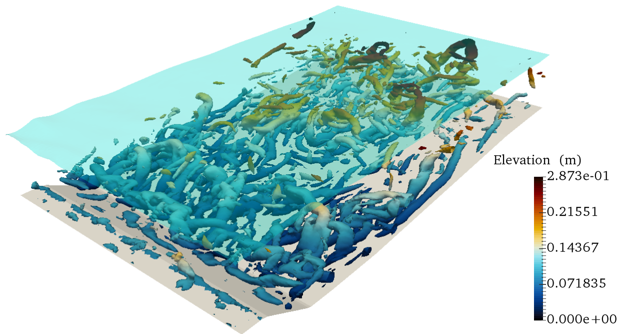

Fig. 7. Isosurfaces of λ_2 criterium.

not affordable. As boundary condition, the Synthetic-Eddy-Method is used as the inlet, and wall models based on the Nikuradse law ($k_s = 0.3$) are prescribed both at the bottom and the lateral boundaries.
A snapshot of the velocity magnitude respectively at the top of the dune, at the middle of the water column and at the free surface is shown in figure 8. It shows the turbulence development along the channel, and its effect on the free surface. The first boils are observable from the first kilometer of the flow. Downstream the turbulent structures are so many that they are not distinguishable.

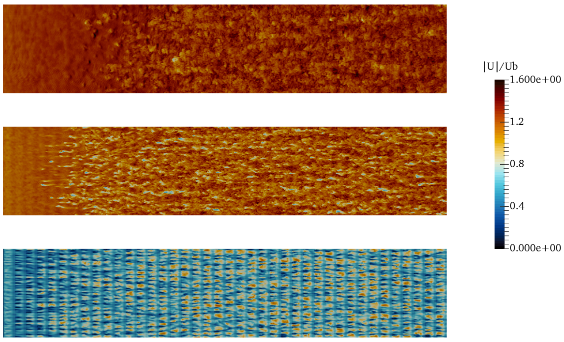

Fig. 8. Snapshot of instantaneous velocity at different (x, y) planes. At the top $z/H = 1$, at the middle $z/H = 0.5$ and at the bottom $z/H = 0.1$ (top of the dune).

Figure 9 presents lateral slices of several quantities, picked in the downstream part of the channel. At the top, the instataneous velocity magnitude is shown. As expected with Large-Eddy-Simulations, it involves high fluctuations. These fluctuations lead to investigate the turbulent kinetic energy, shown in the middle picture of figure 9. Its distribution is satisfactory but its amplitude is lower than expected. However using LES methods with this configuration allows to identify turbulence structures, as shown at the bottom on the figure, which displays the λ_2 criterium, for which high values characterizes vortex centers. Although the Reynolds stresses and the turbulent kinetic energy can not be described accurately due to the coarse discretization, this model allows to observe vortexes of varying sizes which could be interesting to discriminate. According to [5], these structures have a different impact on turbines. Those with the size of machines are more susceptible to affect their power, whereas the smaller structures are responsible for the fatigue of turbines. According to these preliminary results, these turbulent structures are transported by the flow in all the water column, and mainly they can quantified.

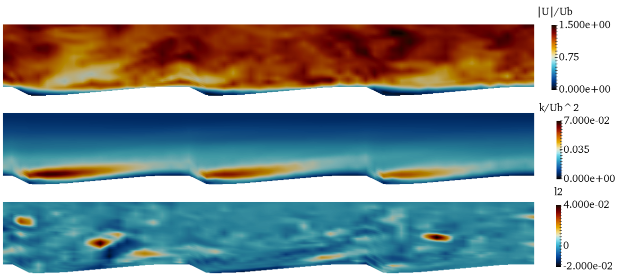

Fig. 9. At the top, velocity magnitude normalized with the bulk velocity. At the middle, normalized turbulent kinetic energy. At the bottom, λ_2 criterium.

As part of the 'THYMOTE' project, coupled ADCP measurements methods (performed in Brest-France in [20]) are going to be realised in the Alderney-Race area, which is a very interesting for tidal energy. These methods allow to evaluate five Reynolds stresses out of six accurately, it will be an opportunity to compare results obtained with TELEMAC-3D. To compensate the coarse grids (in terms of Large-Eddy-Simulations) used to model very high Reynolds number flows, additional subgrid methods are investigated [21] [22].

VI. Conclusion

In order to characterize thoroughly the turbulence of environmental flows an particularly in tidal sites, a Large-Eddy-Simulation approach is developed in the code TELEMAC-3D. Subgrid models have been implemented, as well boundary conditions such as the Synthetic Eddy Method which aims at introducing velocity fluctuations at the inlet of the computational domain. The development of the LES method also requires reducing the numerical dissipation of TELEMAC-3D (which was originally designed for RANS simulations) by adding a new advection scheme in the solver.
The model validation with measurements acquired in an open-channel over two dimensional dunes demonstrates the good model performance. Both the averaged and the fluctuating quantities computed by the model are in good agreement with the experiments. It allows moreover to indentify efficiently turbulent structures moving in the flow, up to the free surface where they are easily observable.
A prospective application is realised, characterizing a flow with the characteristics of tidal flows ($Re \approx 10^8$). Despite the coarse discretization, it allow to better understand the turbulent processes of these configurations and to observe the behaviour of vortexes over complex bed forms. The final objective is to perform a regional simulation tidal currents and particularly the Alderney Race area (Raz-Blanchard in French) which represents a very promising tidal energy site [23]. We foresee that Large-Eddy Simulations will enable to understand thoroughly the turbulence and give reliable input data for the design of the turbines and for the optimisation of the turbine layout.

Acknowledgment

This work has benefited from a State support managed by the National Research Agency under the Investments for the Future program bearing the reference ANR-10-IEED-0006-11.

REFERENCES

[1] J.-M. Hervouet, *Hydrodynamics of free surface flows: modelling with the finite element method*. John Wiley & Sons, 2007.

[2] J. Smagorinsky, "General circulation experiments with the primitive equations: I. the basic experiment*," *Monthly weather review*, vol. 91, no. 3, pp. 99–164, 1963.

[3] M. Van Mierlo and J. De Ruiter, "Turbulence measurements above artificial dunes," *Q0789*, 1988.

[4] M. Togneri and I. Masters, "Micrositing variability and mean flow scaling for marine turbulence in ramsey sound," *Journal of Ocean Engineering and Marine Energy*, vol. 2, no. 1, pp. 35–46, 2016.

[5] T. Clark, "Turbulence in marine environments (time): A framework for understanding turbulence and its effects on tidal devices." *Proceedings of the 11th European Wave and Tidal Energy Conference, Nantes, 08B4-3-1.*, 2015.

[6] P. Sagaut, *Large eddy simulation for incompressible flows: an introduction*. Springer Science & Business Media, 2006.

[7] N. Jarrin, S. Benhamadouche, D. Laurence, and R. Prosser, "A synthetic-eddy-method for generating inflow conditions for large-eddy simulations," *International Journal of Heat and Fluid Flow*, vol. 27, no. 4, pp. 585–593, 2006.

[8] T. Stoesser, C. Braun, M. Garcia-Villalba, and W. Rodi, "Turbulence structures in flow over two-dimensional dunes," *Journal of Hydraulic Engineering*, vol. 134, no. 1, pp. 42–55, 2008.

[9] N. A. Phillips, "A coordinate system having some special advantages for numerical forecasting," *Journal of Meteorology*, vol. 14, no. 2, pp. 184–185, 1957.

[10] P.-L. Viollet, J.-P. Chabard, and P. Esposito, *Mécanique des fluides appliquée: écoulements incompressibles dans les circuits, canaux et rivières, autour des structures et dans l'environnement*. Presses des Ponts, 2003.

[11] J. Boussinesq, *Essai sur la théorie des eaux courantes*. Impr. nationale, 1877.

[12] W. Rodi, "Large eddy simulation of river flows," in *Proceedings of the International Conference on Fluvial Hydraulics, Bundesanstalt fur Wasserbau, Karlsruhe, Germany*, 2010, pp. 23–32.

[13] L. Prandtl, "Bericht uber untersuchungen zur ausgebildeten turbulenz," *Zs. angew. Math. Mech.*, vol. 5, pp. 136–139, 1925.

[14] C. Polatel, *Large-scale roughness effect on free-surface and bulk flow characteristics in open-channel flows*. ProQuest, 2006.

[15] U. Piomelli and J. R. Chasnov, "Large-eddy simulations: theory and applications," *Turbulence and transition modelling*, vol. 2, pp. 269–336, 1996.

[16] J. Jeong and F. Hussain, "On the identification of a vortex," *Journal of fluid mechanics*, vol. 285, pp. 69–94, 1995.

[17] J. Best, "The fluid dynamics of river dunes: A review and some future research directions," *Journal of Geophysical Research: Earth Surface*, vol. 110, no. F4, 2005.

[18] P. Stansby, N. Chini, and P. Lloyd, "Oscillatory flows around a headland by 3d modelling with hydrostatic pressure and implicit bed shear stress comparing with experiment and depth-averaged modelling," *Coastal Engineering*, vol. 116, pp. 1–14, 2016.

[19] C. Moulinec, C. Denis, C.-T. Pham, D. Rougé, J.-M. Hervouet, E. Razafindrakoto, R. Barber, D. Emerson, and X.-J. Gu, "Telemac: An efficient hydrodynamics suite for massively parallel architectures," *Computers & Fluids*, vol. 51, no. 1, pp. 30–34, 2011.

[20] A. Pieterse, J.-F. Filipot, C. Maisondieu, L. Kilcher, and N. Chaplain, "Coupled adcp measurements for tidal turbulence characterization," in *Proceedings of the 12th European Wave and Tidal Energy Conference*, 2017.

[21] A. Bensoussan and R. Temam, "Equations stochastiques du type navier-stokes," *Journal of Functional Analysis*, vol. 13, no. 2, pp. 195–222, 1973.

[22] U. Schumann, "Stochastic backscatter of turbulence energy and scalar variance by random subgrid-scale fluxes," *Proc. R. Soc. Lond. A*, vol. 451, no. 1941, pp. 293–318, 1995.

[23] D. Coles, L. Blunden, and A. Bahaj, "Assessment of the energy extraction potential at tidal sites around the channel islands," *Energy*, vol. 124, pp. 171–186, 2017.