Bourgoin, Adrien; Guillou, Sylvain; Thiébot, Jérôme; Ata, Riadh; Benhamadouche, Sofiane

Development of a Large-Eddy-Simulation approach with Telemac-3D

Development of a Large-Eddy-Simulation approach with Telemac-3D

Adrien Bourgoin, Sylvain Guillou and Jérôme Thiébot
Laboratoire Universitaire des Sciences Appliquées de Cherbourg, UNICAEN
60 rue Max-Pol Fouchet 50130 Cherbourg-en-Cotentin France
adrienbourgoin@gmail.com

Riadh Ata and Sofiane Benhamadouche
Laboratoire National d'Hydraulique et Environnement, EDF R&D
Saint-Venant Laboratory for Hydraulics
6 Quai Watier 78400 Chatou France

***Abstract*— Turbulence has a major impact on environmental flows. The complex seabed morphology combined with strong tidal flow gives rise to powerful turbulent structures that affect the different transport processes, such as heat or sediment transport. RANS modelling does not compute the turbulent variables, that is why a Large-Eddy-Simulation (LES) approach is developed in Telemac-3D. This method seeks to calculate the unsteady aspects of flows by modelling the largest turbulent structures. Thanks to the increase of calculation resources, the LES approach is nowadays applicable to simulate a large variety of environmental flows. This paper presents the implementation of new turbulence closures and the adjustment of the numerical schemes in Telemac-3D.**

The development of the method consists in implementing subgrid models and in introducing artificial turbulence in the computational domain with the Synthetic-Eddy-Method (SEM). In the meantime, we undertook efforts to reduce the numerical dissipation of the code. It was a prerequisite to propagate reliably the flow fluctuations.

This paper presents the validation of the developments in Telemac-3D considering simple flow configurations where both experimental and numerical data are available. The first results showed encouraging model behavior. The next step will consist in simulating the hydrodynamics of the Alderney Race (Raz-Blanchard in French).

I. INTRODUCTION

In environmental flows over complex bottom morphology, understanding turbulence is essential for studying processes such as sediment transport or heat transfer. In its original version, Telemac-3D uses a RANS (Reynolds Averaged Navier-Stokes) approach [4] where the averaged turbulent flow is modelled by using for example the famous $k-\epsilon$ model. Although such modelling is the most popular for natural flows, it does not provide accurate information as regards the fluctuating (instantaneous) quantities. The improvement of computational resources nowadays permits using Large-Eddy-Simulation for modelling environmental flows. This approach enables simulating the random aspect of turbulence, which plays an important role in transport phenomena. The method consists in introducing a subgrid model to mimic the smallest motion scales and in simulating the other scales by directly resolving the Navier-Stokes equations. The implementation of LES requires additional treatment, especially for the boundary conditions. In fact, contrary to RANS model, the velocity fluctuations have to be introduced in the computation domain. Moreover, near the solid boundaries, wall models are required to prevent an unaffordable mesh refinement. Finally it requires accurate and non-dissipative numerical schemes.

In this paper, several developments already done or being done in Telemac-3D are described. They are tested using a validation test case [11] representing an open channel flow over two-dimensional dunes.

II. LES METHODS

The concept of Large-Eddy-Simulation (LES) is to divide the energy spectrum [6] of the flow in two parts by using a numerical filter to separate the smallest turbulent length scales from the others. Then, these two parts are treated differently. As the smallest turbulent structures have a more universal behaviour and are hardly independent on the initial conditions, they can be modelled. Conversely, the biggest structures are directly solved by the motion equations. The filtering of the Navier-Stokes equations implies introducing a new unknown tensor called subgrid tensor. This tensor characterizes the interactions between the smallest turbulent scales and the others. It is evaluated by using a subgrid model. The filtered quantities, noted $\tilde{f}$, are computed by solving the filtered Navier-Stokes equations, written as:

$$\begin{cases} \dfrac{\partial \tilde{u}_i}{\partial x_i} = 0 \\ \dfrac{\partial \tilde{u}_i}{\partial t} + \tilde{u}_j \dfrac{\partial \tilde{u}_i}{\partial x_j} = -\dfrac{1}{\rho}\dfrac{\partial \tilde{p}}{\partial x_i} + \dfrac{\partial}{\partial x_j}\left(\nu \dfrac{\partial \tilde{u}_i}{\partial x_j}\right) - \dfrac{\partial \tau_{ij}}{\partial x_j} \end{cases}$$

where ν is the molecular viscosity and τ_{ij} is the subgrid tensor.

A. Subgrid modelling

To model the subgrid tensor, the most popular approach is named functional modelling [13]. It assumes that the interactions between the small and the big turbulent scales can be treated as an energetic process. The action of the tensor is modelled by introducing a subgrid viscosity ν_t linking directly the subgrid tensor to the filtered velocity gradients $\tilde{s}$ with a Boussinesq assumption-like formulation:

$$\tau_{ij} = \frac{2}{3}\tau_{kk}\delta_{ij} - 2\nu_t\tilde{S}_{ij}$$

Then, the subgrid viscosity has to be evaluated with a more or less complex formulation, given by the subgrid models.

Several models have been implemented in Telemac-3D such as the Smagorinsky model [15], the dynamic Smagorinsky model [2] or the WALE model [9]. The latter model, which is retained here, was designed to provide a good asymptotic behaviour near the solid walls where the viscosity varies linearly with z^3, where z is the distance from the wall. The subgrid viscosity is written as:

$$\nu_t = (C_w\tilde{\Delta})\frac{\left(S_{ij}^dS_{ij}^d\right)^{\frac{3}{2}}}{\left(\tilde{S}_{ij}\tilde{S}_{ij}\right)^{\frac{5}{2}} + \left(S_{ij}^dS_{ij}^d\right)^{\frac{5}{4}}}$$

with

$$S_{ij}^d = \frac{1}{2}\left(g_{ij}^2 + g_{ji}^2\right) - \frac{1}{3}\delta_{ij}g_{kk}^2 \ , \ g_{ij} = \frac{\partial\tilde{u}_i}{\partial x_j}$$

and C_w a constant evaluated to 0.325. In the formulation, $\tilde{\Delta}$ is the filter width. It is a length scale directly linked to the grid size. Because of the prismatic shape of the elements used in Telemac-3D, two length scales have been defined to characterize respectively a vertical and a horizontal length.

B. Boundary conditions

The prescription of the LES boundary conditions is crucial as the inflow conditions have a strong influence on the flow characteristics in the calculation domain. In hydraulics, flows are mainly dominated by the advection. Thus, the prescribed values of the velocity must be as realistic as possible. The most popular approach is to prescribe Dirichlet boundary conditions over the inlet area. It is achieved by introducing a mean quantity and a fluctuating part. This technique is possible when the velocity fluctuations are known. For flows over simple bottom morphology, a common approach is to use a periodicity between the outlet and the inlet. However, when the complex geometry of a flow does not allow using periodicity, an artificial turbulence needs to be introduced at the inlet. In this section, two inlet boundary conditions (developed in Telemac-3D) are described, as well as an outlet boundary condition.

The recycling method is also called pseudo-periodicity. It aims at prescribing, at the time t^n, at the inlet $(x = x_0)$ the velocity obtained at the outlet $(x = x_R)$ at the time t^{n-1}. For each component of the velocity, it is written:

$$u_i(x_0, y, z, t^n) = u_i(x_R, y, z, t^{n-1})$$

This method is different from the real periodicity because it is explicit. Both recycling and periodicity are widely used but they have the drawback of introducing a spurious periodicity in the streamwise direction which can trigger instabilities as shown in [17]. To avoid this, a spanwise shift can be introduced at the inlet (a shift with respect to the outlet).

Moreover, for channel flows, the recycling method has the disadvantage of neglecting the friction loss. Near solid walls, the thickness of the boundary layer should be bigger at the outlet δ_R than at the inlet δ_0. This overvaluation at the inlet can be compensated by imposing:

$$u_i(x_0, y, z, t^n) = u_i(x_R, y, z\delta_R/\delta_0, t^{n-1})$$

For free surface flows, a source term has to be added to the streamwise Navier-Stokes equations in order to consider the friction loss of the flow [2]. This term is the mass density of force defined by:

$$F_x = -\frac{u_\tau^2}{h}$$

where u_τ is the friction velocity and h is the water depth.

The Synthetic Eddy Method (SEM) [5] consists in injecting an artificial turbulence in the computation domain. To do that, a virtual box around the inlet is introduced, where artificial eddies are created. The dimensions of the box in each dimension x_j are defined by:

$$\begin{cases} x_{j,min} = \min\limits_{x\in S}(x_j - \sigma(x)) \\ x_{j,max} = \max\limits_{x\in S}(x_j + \sigma(x)) \\ \Delta x_j = x_{j,max} - x_{j,min} \end{cases}$$

where S is the inlet surface and σ is a length scale for the virtual eddies, given by:

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

with k the turbulent kinetic energy, ϵ the turbulent dissipation rate, κ the von Karman constant, δ the half of the water depth and $\tilde{\Delta}$ the filter width.

The SEM consists in creating N virtual turbulent structures in the virtual box. Each of this structure has a random position and a random orientation in the three dimensions of space, noted $\epsilon_j^k \in \{-1,1\}$. Once the structures are created, the fluctuations at the inlet u_i' at the position $\boldsymbol{x}$ are computed from the characteristics of these eddies by using a shape function f_σ, such as:

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

where $\boldsymbol{x_k}$ is the position of the kth eddy, f_σ is the shape function that can be written as:

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

and $c_i^k = a_{ij}\epsilon_j^k$ is the intensity of the k^{th} eddy in the ith direction, depending on the a_{ij} that is the Cholesky decomposition of a prescribed Reynolds tensor R_{ij}, expressed as:

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

At each time step, the eddies are transported by the mean flow in the virtual box. When an eddy leaves the box, it is reintroduced at the inlet of the box with new random spanwise and vertical positions as well as new intensities.

A particular outlet boundary condition is also required to perform Large-Eddy-Simulation. Theoretically, it is based on the prescription of the stress continuity [14] on each side of the domain written as:

$$\begin{cases} -p_{in} + \mu \dfrac{\partial u_n}{\partial n} = -p_{out} + \tau_n^{out} \\ \mu \dfrac{\partial u_t}{\partial n} = \tau_t^{out} \end{cases}$$

Where u_n and u_t are respectively the normal and tangential velocity, μ is the dynamic viscosity, p_{in} and p_{out} are the pressure inside and outside the domain and τ^{out} is the outside boundary stress. Telemac-3D assumes that there is no change of the velocity components across the outlet section, which yields to the set of boundary conditions:

$$\begin{cases} p = p_{out} \\ \dfrac{\partial u_n}{\partial n} = 0 \end{cases}$$

However this is too restrictive for Large-Eddy-Simulation since, for incompressible fluids, the tangential velocity is assumed to be null. Thus, a convective boundary condition has been implemented, written as:

$$\begin{cases} p = p_{out} \\ \dfrac{\partial u_n}{\partial t} + \vec{u}.\dfrac{\partial u_n}{\partial n} = 0 \end{cases}$$

where $\vec{u}$ is the advection velocity.

C. *Numerical dissipation reduction*

LES consists in adding a subgrid viscosity to the diffusion term of the Navier-Stokes equations. The implementation is similar to the adding of turbulent viscosity in a RANS method [7], [16]. However, the magnitude of the subgrid viscosity is much smaller than the turbulent viscosity. Using a too dissipative numerical scheme could therefore inhibit the subgrid modelling. High order and non-dissipative schemes are thus required in order to transport efficiently the flow fluctuations. Furthermore, refined grids and small time steps are also required. In [10], the recommended dimensionless grid sizes in the streamwise, spanwise and vertical dimension are respectively $\Delta x^+ = 50 - 150$, $\Delta y^+ = 15 - 50$ and $\Delta z^+ < 2$. Regarding the vertical discretization, larger cell sizes can be used thanks to a law of the wall.

In addition to the efforts in reducing the numerical dissipation, several approximations used in the original version of Telemac-3D have been corrected so that the fluctuations propagate in the flow. The SUPG advection scheme [1] has been modified to recover the vertical motion, through a redefinition of the vertical advection scheme and the building of finite element arrays without mass-lumping-type and zero vertical velocity approximations. Moreover, the projection step of the resolution does not involve the assumption of velocity constant per element used in Telemac-3D.

III. APPLICATION

A. *Flow over a dune: presentation*

The flow presented here describes a turbulent open channel flow over two-dimensional dunes of height k (see Fig. 1). This case has been studied experimentally by Polatel [10] who measured the flow over a train of 22 dunes using laser Doppler velocimetry. In those experiments, the dune height is $k = 20\,mm$ and its length is $\lambda = 400\,mm$. The maximum flow depth is fixed so that $h = 4k$. The Reynolds number, based on the bulk velocity U_b and the maximum water depth, is approximately 25000.

B. *Computational set up*

This case has been reproduced with Telemac-3D using a computation domain covering a single dune. Indeed the use of the pseudo-periodicity condition (see section II.B) allows to reduce significantly the domain. It has been discretized with $161 \times 151 \times 40$ points, which corresponds respectively to the dimensionless grid spacing of $\Delta x^+ \approx 50$, $\Delta y^+ \approx 30$ and $\Delta z^+ \approx 20$. Those values are in line with the recommended values for LES according to [10], except for the vertical discretization where a Nikuradse wall law is used.

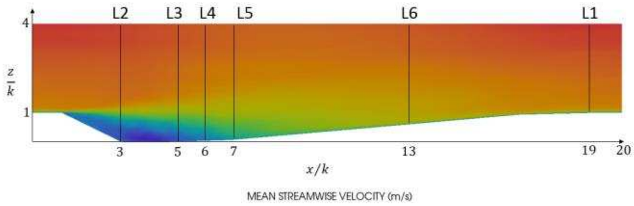

Figure 1. *Morphology of the open-channel, the six positions of measurments and mean streamwise velocity.*

As Telemac-3D clips several quantities (weak formulation), there is a minimum cell size. This limitation has been circumvented by multiplying all lengths by 4 (with respect to the experiments sizes) and by multiplying the bulk velocity by ¼ in order to keep the Reynolds number constant.

As boundary condition, the Synthetic-Eddy-Method is used for the first time steps of the calculation, in order to introduce fluctuations in the flow. An isotropic Reynolds stress tensor is prescribed. Its extra-diagonal components are null and the others are evaluated with:

$$R_{ii} = \frac{2}{3} k^+ u_\tau^2$$

where u_τ is the friction velocity and k^+ is the dimensionless turbulent kinetic energy. The latter is defined with a theoretical law [19] written as:

$$k^+ = 0.07(z^+)^2 e^{-\frac{z^+}{8}} + \frac{4.5\left(1 - e^{-\frac{z^+}{20}}\right)}{1 + \frac{4z^+}{Re_\tau}}$$

where z^+ is the dimensionless distance to the wall and Re_τ is the turbulent Reynolds number ($Re_\tau = \frac{u_\tau h}{\nu}$). Moreover, a mean streamwise velocity profile is prescribed when using the SEM. It is dtermined using a Reichardt law [12], given by:

$$U^+ = \frac{1}{\kappa}\log(1 + \kappa z^+) + 7.8\left(1 - e^{-\frac{z^+}{11}} - \frac{z^+ e^{-0.33 z^+}}{11}\right)$$

An example of streamwise velocity prescribed at the inlet is given in Fig. 2. Once the fluid has passed through the computational domain, the Synthetic-Eddy-Method and the Reichardt law are replaced by the pseudo-periodicity method.

Regarding the numerical configuration of Telemac-3D, the modified SUPG advection scheme (see section II.C) along with Crank-Nicholson time integration scheme with a CFL of a magnitude of 0.3 are used. Finally the WALE model (see section II.A) is used as subgrid model. The velocities and the root-mean-square velocities are averaged over 5000 s, which represents a duration of about 250 flow recirculations. The calculation duration is 28h with 56 threads.

C. Statistic results

The averaged streamwise velocity and three components of the Reynolds stress tensor are extracted at six locations shown in Fig. 1, and compared with the experimental results from [11].

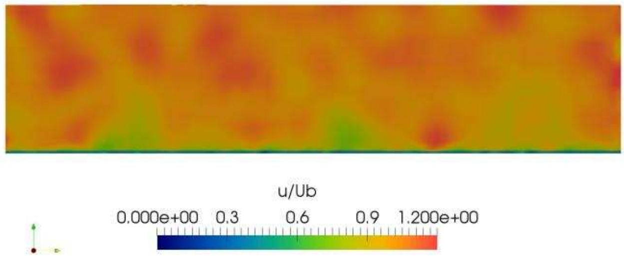

Figure 2. *Streamwise velocity prescribed at the inlet by using the SEM and the Reichardt law.*

The averaged streamwise velocity normalized by the bulk velocity profiles obtained with Telemac-3D are compared to the experimental results of [11] in Fig. 3. The agreement with experiments is good, except at the position L1 and L6 where a deviation is observed in the low part of the flow.

Fig. 4 and 5 show respectively the streamwise and the vertical root-mean square velocity along the six verticals. The agreement between the Telemac-3D results and the measurements are overall good for both of the turbulent intensities, despite a slight overestimation of the streamwise root-mean-square velocity at the location L3 and L4.

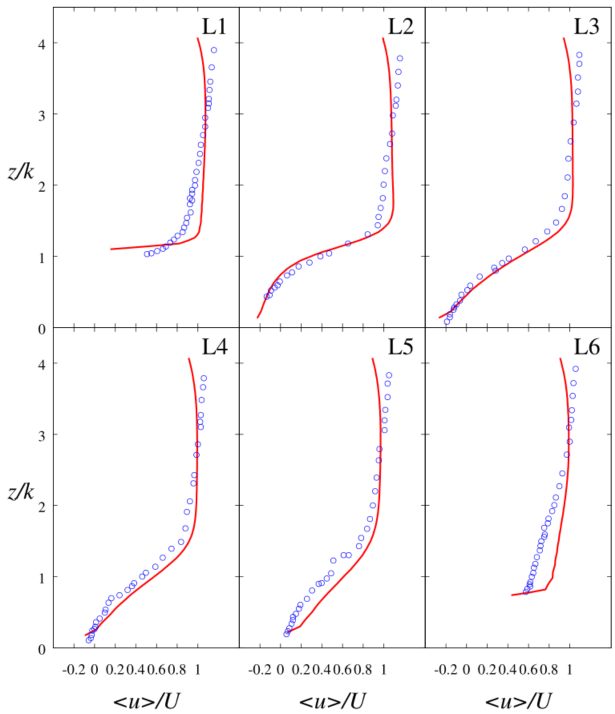

Figure 3. *Averaged streamwise velocity along the six measurement verticals, obtained with Telemac-3D (line) and the experiments of Polatel (symbol) [11].*

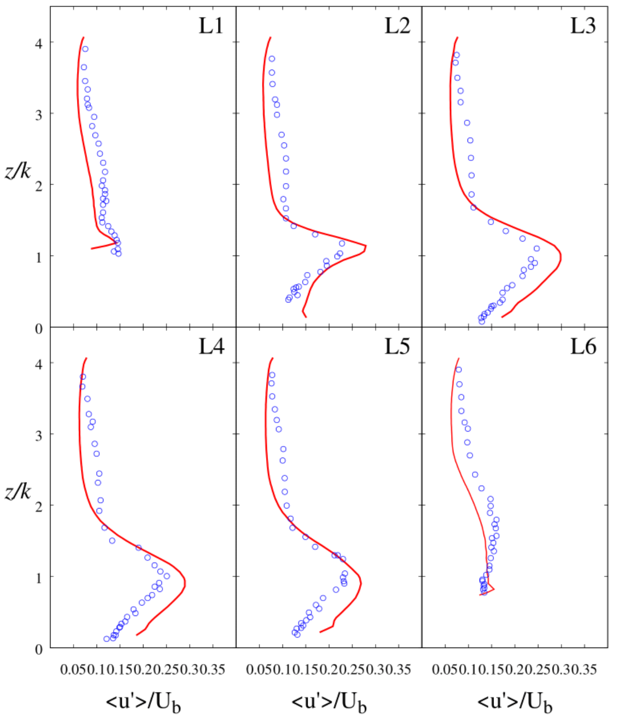

Figure 4. *Streamwise root-mean-square velocity along the six measurement verticals, obtained with Telemac-3D (line) and the experiments of Polatel (symbol) [11].*

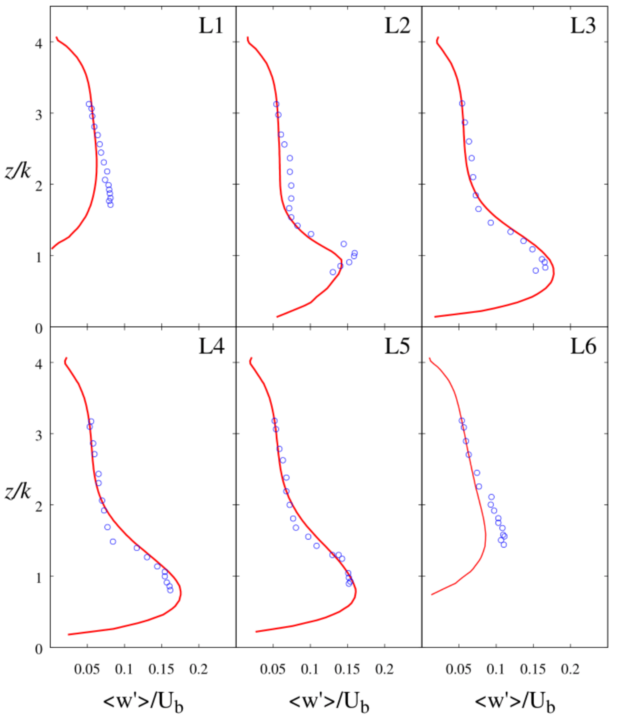

Figure 5. *Vertical root-mean-square velocity along the six measurement verticals, obtained with Telemac-3D (line) and the experiments of Polatel (symbol) [11].*

Fig. 6 shows the Reynolds shear stress profiles. The model-experiment comparison is also satisfactory. The maximum intensity is slightly overestimated at the locations L3 and L4.

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. For instance, Fig. 7 shows some isosurfaces of pressure. This figure highlights the vortex generation at the foot of the dune and the transport of the vortex by the flow with an inclined orientation. The analyse of the instantaneous results indicate that spanwise vortices are generated in the separated shear layer and that they ascend up to the free surface.

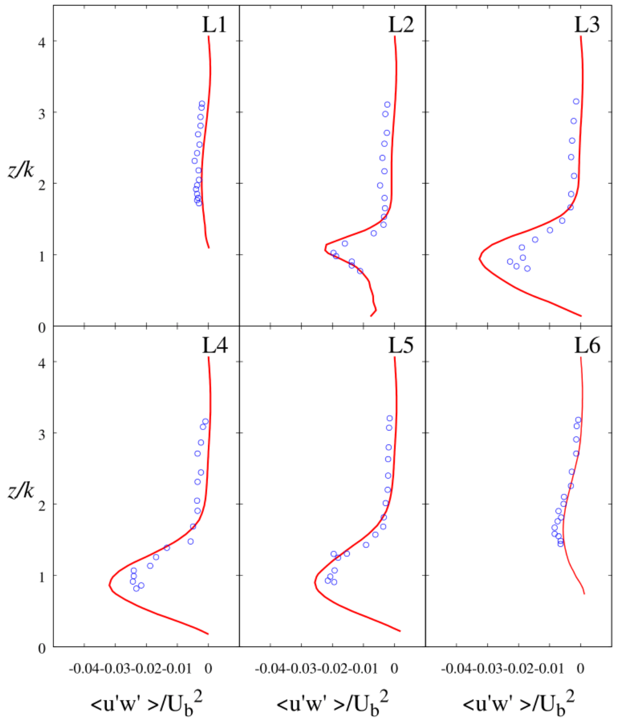

Figure 6. *Reynolds shear stress along the six measurement verticals, obtained with Telemac-3D (line) and the experiments of Polatel (symbol) [11].*

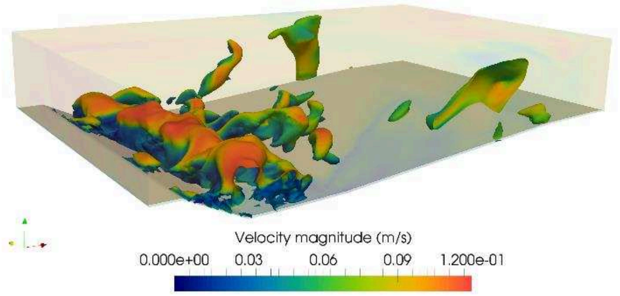

Figure 7. *Isosurfaces pressure colored by the magnitude of velocity*

IV. CONCLUSION

A Large-Eddy-Simulation approach is developed in Telemac-3D [4] for modelling free surface complex flows. After carrying out a state of the art of LES methods in hydraulics, several subgrid models are selected to be implemented. Since this kind of simulation requires specific boundary conditions, the Synthetic Eddy Method (SEM) [5] is used at the inlet boundary for generating the first velocity fluctuations and a recycling method is then used to introduce a realistic turbulence. Additional boundary conditions was also required for the stability of the calculation, such as a convective outflow boundary condition. A special care was at least required to reduce the numerical dissipation of Telemac-3D, mostly due to several assumptions on the vertical velocity in the advection step.

The results obtained by the LES model of Telemac-3D are in good agreement with experimental results of a flow over dunes. The turbulence indicators show satisfactory model performance. Both the averaged velocity and the Reynolds stress fit with the experimental results [11].

A final objective would be to perform regional simulations. Due to the high Reynolds number of environmental flows and the considerable computational cost of Large-Eddy-Simulation, a further investigation is currently in progress on the implementation of a DES method based on the Spalart-Allmaras [16] RANS model, which is a hybrid method between RANS and LES.

ACKNOWLEDGMENT

This work is funded by French Agence Nationale de la Recherche programme under a contract with a reference number ANR-10-IEED-0006-11

REFERENCES

[1] Brooks, A. N., & Hughes, T. J. (1982). Streamline upwind/Petrov-Galerkin formulations for convection dominated flows with particular emphasis on the incompressible Navier-Stokes equations. Computer methods in applied mechanics and engineering, 32(1-3), 199-259.

[2] Germano, M., Piomelli, U., Moin, P. & Cabot, W.H., (1991). A dynamic subgrid-scale eddy viscosity model. *Physics of Fluids A: Fluid Dynamics* (1989-1993), 3(7):1760–1765.

[3] Guillou, S. & Makhloufi, R., (2007). Effect of a shear-thickening rheological behaviour on the friction coefficient in a plane channel flow: a study by direct numerical simulation. *Journal of non-newtonian fluid mechanics*, 144(2):73–86.

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

[5] Jarrin, N., Benhamadouche, S., Laurence, D. & Prosser, R., (2006). A synthetic-eddy-method for generating inflow conditions for large-eddy simulations. *International Journal of Heat and Fluid Flow*, 27(4):585–593.

[6] Kolmogorov, A. N., (1941). Dissipation of energy in the locally isotropic turbulence. *Proceedings: Mathematical and Physical Sciences,* 434(1890):15-17.

[7] Launder, B. E. & Sharma, B. I., (1974). Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc. *Letters in heat and mass transfers,* 1(2):131-137.

[8] Lilly, D.K., (1992). A proposed modification of the germano subgrid-scale closure method. *Physics of Fluids A: Fluid Dynamics* (1989-1993), 4(3):633–635.

[9] Nicoud, F. & Ducros, F.,(1999). Subgrid-scale stress modelling based on the square of the velocity gradient tensor. *Flow, turbulence and Combustion*, 62(3):183–200.

[10] Piomelli, U. & Chasnov, J. R.,(1996). Large-eddy simulations: theory and applications. *Turbulence and transition modelling*, vol. 2, pp. 269-336.

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

[12] Reichardt, H., (1951). Vollständige Darstellung der turbulenten. Geschwindigkeitsverteilung in glatten Leitungen, *ZAMM-J. Appl. Math. Mech./Z. Angew. Math. Mech. 31(7),* 208-219.

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

[14] Sani, R. L. & Gresho, P. M., (1994). Résumé and remarks on the open boundary condition minisymposium. *International Journal for Numerical Methods in Fluids,* 18(10): 983-1008.

[15] Smagorinsky, J., (1963). General circulation experiments with the primitive equations: I. the basic experiment. *Monthly weather review, 91(3):99–164.*

[16] Spalart, P. R. & Allmaras, S. R., (1994). A one-equation turbulence model for aerodynamic flows. *Recherche Aerospatiale,* n/a(1):5-21.

[17] Spalart, P. R., Strelets, M. & Travin, A.,(2006). Direct numerical simulation of large-eddy-break-up devices in a boundary layer. *International Journal of Heat and Fluid Flow*, 27(5) :902–910.

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

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