View metadata, citation and similar papers at core.ac.uk

brought to you by CORE

provided by Hydraulic Engineering Repository

HENRY
Hydraulic Engineering Repository

Ein Service der Bundesanstalt für Wasserbau

Conference Paper, Published Version

Bourgoin, Adrien; Guillou, Sylvain; Thiebot, Jerome; Ata, Riadh; Benhamadouche, Sofiane

Development of a Large-Eddy-Simulation approach for free surface complex flows

Zur Verfügung gestellt in Kooperation mit/Provided in Cooperation with:
TELEMAC-MASCARET Core Group

Verfügbar unter/Available at: https://hdl.handle.net/20.500.11970/104531

Vorgeschlagene Zitierweise/Suggested citation:
Bourgoin, Adrien; Guillou, Sylvain; Thiebot, Jerome; Ata, Riadh; Benhamadouche, Sofiane (2016): Development of a Large-Eddy-Simulation approach for free surface complex flows. In: Bourban, Sébastien (Hg.): Proceedings of the XXIIIrd TELEMAC-MASCARET User Conference 2016, 11 to 13 October 2016, Paris, France. Oxfordshire: HR Wallingford. S. 143-148.

Standardnutzungsbedingungen/Terms of Use:

Die Dokumente in HENRY stehen unter der Creative Commons Lizenz CC BY 4.0, sofern keine abweichenden Nutzungsbedingungen getroffen wurden. Damit ist sowohl die kommerzielle Nutzung als auch das Teilen, die Weiterbearbeitung und Speicherung erlaubt. Das Verwenden und das Bearbeiten stehen unter der Bedingung der Namensnennung. Im Einzelfall kann eine restriktivere Lizenz gelten; dann gelten abweichend von den obigen Nutzungsbedingungen die in der dort genannten Lizenz gewährten Nutzungsrechte.

Documents in HENRY are made available under the Creative Commons License CC BY 4.0, if no other license is applicable. Under CC BY 4.0 commercial use and sharing, remixing, transforming, and building upon the material of the work is permitted. In some cases a different, more restrictive license may apply; if applicable the terms of the restrictive license will be binding.

Development of a Large-Eddy-Simulation approach for free surface complex flows.

Adrien Bourgoin,
Sylvain Guillou,
and Jerome Thiebot.
LUSAC,
60 rue Max Pol Fouchet
50130 Cherbourg-Octeville
Email: adrien-externe.bourgoin@edf.fr

Riadh Ata
and Sofiane Benhamadouche
LNHE/MFEE, EDF R&D,
6 quai Watier,
78400 Chatou
Email: riadh.ata@edf.fr

***Abstract*—For the purpose of improving turbulent process modelling of environmental flows, a LES approach is being developed in TELEMAC-3D [3]. Although still not widely used, LES is increasingly applied for this kind of flows, thanks to the computational resources growth. RANS modelling, such as the one using the standard $k-\epsilon$ model, remains favourable for numerical modelling of natural flows, and is by the way the only procedure currently available in the code for turbulence modelling. Nevertheless, in many cases, this approach cannot provide efficiently enough the intended data, such as the turbulence induced by the bathymetry. The present study is thus dedicated to the development of the model TELEMAC-LES. The different stages involved the implementation of several LES subgrid scale models, such as the standard and the dynamic Smagorinsky [13], [1], [8], and several numerical tools and tests for performing a LES. For example, the turbulent inlet boundary condition is achieved by a Synthetic Eddy Method [4] which produces a fluctuating and coherent boundary condition in order to perform the validation cases. Moreover, as TELEMAC-3D uses prismatic meshes that can be strongly anisotropic, the turbulence model has to be modified, by introducing two filter length scales instead of one. An important part of the developments has been achieved. The chosen validation cases, a flow over periodic hills [9] and in an open channel [6], reveal lots of issues related to this kind of models (numerical scheme order, mesh quality, mesh anisotropy, CPU time, boundary conditions, periodicity,...).**

I. INTRODUCTION

In environmental flows over complex geometry, understanding turbulence is essential for studying other processes, such as sediment transport or heat transfers. A RANS (Reynolds Averaged Navier-Stokes) treatment can be used in TELEMAC-3D [3], that aims to model an averaged turbulent flow, by using for example the famous $k-\epsilon$ model. Although this kind of modelling is mostly used for natural flows, it is sometimes not accurate enough for providing specific information. The improvement of computation ressources 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, which models the smallest motion scales, whereas the other ones are directly simulated by the Navier-Stokes equations. It also requires additional processing, particularly for the boundary conditions treatment. For example, contrary to RANS model, some velocity fluctuations have to be introduced in the computation domain. Morevoer, near solid boundaries, wall models are regularly used for avoiding a considerable mesh refinement.

In this paper, several developments already done or being done in TELEMAC-3D are described. They are tested using a validation test case [2] representing an open channel flow at a low Reyolds number.

II. METHODS

A. Subgrid models

The main idea of Large-Eddy Simulation (LES) is to divide the energy spectrum in two parts, by using a numerical filter. The first part correponds to the smallest scales named subgrid scales, which are modelled ; and the second part is the biggest scales which are directly solved with the Navier-Stokes equations. This operation introduces a new unknow tensor, called subgrid tensor and yields each variable expanded into large-scale and subgrid parts. For its treatment, there are two main approaches, which are the functional and the structural modeling [12]. This first idea is to estimate the effect of this term, and the second one is to reproduce it directly. In order to develop a LES approach in TELEMAC-3D, the selected subgrid models are the standard Smagorinsky [13], its dynamic extension [8] and the WALE model [11]. Indeed, these schemes are the most used in the litterature, and can be used for many configurations of flows with a reasonable computation cost.

1) Smagorinsky model: The Smagorinky model [13] is a subgrid model and can be referred as a functionnal model. It aims to introduce a subgrid viscosity ν_T for modeling the energy transfer processus of the subgrid scales, by using quantities emanating from the resolved scales, with the formulation:

$$\nu_T = (C_s\bar{\Delta})^2|\bar{S}| \tag{1}$$

where C_s is the Smagorinsky constant, $\bar{S}$ is the filtered rate-of-strain tensor and $\bar{\Delta}$ is the filter width. This last viscosity is linked in practice to the grid size.Then this quantity is added to the molecular viscosity and is involved in the diffusion step of the Navier-Stokes solving. However this subgrid model is based on a homogeneous and isotropic turbulence assumption, so it requires some adaptations to apply it to complex configurations. For example in channel flows modelling, the Smagorinsky constant has to be reduced near the walls, by introducing a damping function [16] or by using the dynamic formulation of this constant [1], [8]. This latter approach consists in evaluating the Smagorinsky constant by using powers of the rate of strain tensor, together with a larger implicit filter, of width $\tilde{\Delta}$. Its expression is:

$$C_s^2 = \frac{< L_{ij} M_{ij} >}{< M_{ij} M_{ij} >} \tag{2}$$

where $< . >$ represents a space averaging introduced for the stability of the model and:

$$\begin{cases} L_{ij} & = & \widetilde{\bar{u}_i \bar{u}_j} - \tilde{\bar{u}}_i \tilde{\bar{u}}_j \\ M_{ij} & = & 2\left(\bar{\Delta}^2 (|\widetilde{\bar{S}|\bar{S}}_{ij} - \tilde{\Delta}^2 |\tilde{\bar{S}}| \tilde{\bar{S}}_{ij}\right) \end{cases} \tag{3}$$

where $\bar{S}_{ij}$ is the rate-of-strain tensor for the resolved scale defined by:

$$\bar{S}_{ij} = \frac{1}{2}\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right) \tag{4}$$

Due to the prismatic meshes used with TELEMAC-3D, two filter widths have been defined in order to adjust the scheme to the anisotropic grid. Indeed, TELEMAC uses a vertical and a horizontal viscosities, that depend also respectively on a vertical length scale $\bar{\Delta}_v$ and a horizontal one $\bar{\Delta}_h$. However the explicit filter is defined by a unique length scale.

2) WALE model: The WALE (Wall Adapting Local Eddy) subgrid model [11] is another extension of the Smagorinsky model. It aims also to model a subgrid viscosity by using the square of the velocity gradient tensor, in order to obtain a better near wall behavior. The subgrid viscosity is estimated with:

$$\nu_T = \Delta_s^2 \frac{(S^d_{ij} S^d_{ij})^{3/2}}{(\bar{S}_{ij}\bar{S}_{ij})^{5/2} + (S^d_{ij} S^d_{ij})^{5/4}} \tag{5}$$

where S^d_{ij} is

$$S^d_{ij} = \frac{(\bar{g}^2_{ij} + \bar{g}^2_{ji})}{2} - \delta_{ij}\frac{\bar{g}_{kk}}{3} \tag{6}$$

and

$$\bar{g}_{ij} = \frac{\partial \bar{u}_i}{\partial x_j} \tag{7}$$

The length scale Δ_s involves an other constant, it is assumed to be $\Delta_s = C_w V^{1/3}$ with $C_w = 0.325$.

B. Boundary conditions

1) Synthetic Eddy Method (SEM): In order to solve the partial differential equations in a finite domain, initial and boundary conditions have to be specified. This section exhibits the methods developed for determining suitable conditions for Large-Eddy simulation computations. The inflow has a strong influence on hydrodynamics. Indeed, free surface flows are dominated by the advection phenomena. So the imposed values of the velocity and the pressure have to be as realistic as possible. The most popular approach is to prescribe Dirichlet boundary condition over the inlet area, but it assumes which requires knowing the velocity fluctuations. To overcome this problem, an idea is to generate a synthetic turbulence at the inlet plane [4]. This method consists in introducing a virtual box around the inlet, where artificial eddies will be created. Its dimensions are defined by:

$$\begin{cases} x_{j,min} & = & \min\limits_{x \in S, i \in 1,2,3} (x_j - \sigma_{ij}(x)) \\ x_{j,max} & = & \max\limits_{x \in S, i \in 1,2,3} (x_j + \sigma_{ij}(x)) \\ \Delta x_j & = & x_{j,max} - x_{j,min} \end{cases} \tag{8}$$

where σ_{ij} is the length scale for the ith velocity components in the jth direction, given by:

$$\sigma_{ij} = \max(\frac{k^{3/2}}{\epsilon}, \bar{\Delta}) \tag{9}$$

Then the velocity fluctuations are generated with the action of N synthetic eddies placed randomly in the virtual box.

$$u'_i = \frac{1}{\sqrt{N}} \sum_{k=1}^{N} c_i^k f_{\sigma_{ij}}(\boldsymbol{x} - \boldsymbol{x}^k) \tag{10}$$

where the function f is defined by:

$$f_{\sigma_{ij}}(\boldsymbol{x} - \boldsymbol{x}^k) = \prod_{j=1}^{3} \sqrt{\Delta x_j} \sqrt{\frac{3}{2\sigma_{ij}}} \left(1 - \frac{|x_j - x_j^k|}{\sigma_{ij}}\right) \tag{11}$$

and c_i^k is an amplitude, written as:

$$c_i^k = a_{ij} \epsilon_j^k \tag{12}$$

with $\epsilon_j^k \in \{-1, 1\}$ and a_{ij} is the Cholesky decomposition of the Reynolds stress tensor R_{ij}:

$$\begin{pmatrix} \sqrt{R_{11}} & 0 & 0 \\ R_{21}/a_{11} & \sqrt{R_{22} - a_{21}^2} & 0 \\ R_{31}/a_{11} & (R_{32} - a_{21}a_{31})/a_{22} & \sqrt{R_{33} - a_{31}^2 - a_{32}^2} \end{pmatrix} \tag{13}$$

As the Reynolds stress tensor is unknown, it is assumed here that its extra diagonal components are assumed to be null and the others are evaluated with $R_{ii} = (2/3)k^+ u_\tau^2$. The introduced synthetic turbulence is also isotropic. The turbulent kinetic energy is evaluated by assuming that:

$$k^+ = 0.07(y^+)^2 \exp\left(\frac{-y^+}{8}\right) + \frac{4.5\left(1 - \exp\left(\frac{-y^+}{20}\right)\right)}{1 + \frac{4y^+}{Re_\tau}} \tag{14}$$

where Re_τ is the Reynolds number based on the friction velocity and $y^+ = \frac{y u_\tau}{\nu}$.

Then, at each time step, the synthetic eddies are transported in the box by a mean flow that is evaluated by the Reichard law:

$$U^+ = 2.5 log(1 + \kappa y^+) + 7.8\left(1 - e^{-\frac{y^+}{11}} - \frac{y^+ e^{-0.33 y^+}}{11}\right) \tag{15}$$

Of course, after a certain time, the eddy will leave the virtual box. Then, they will be replicated randomly at the inlet. The synthetic eddy method allows to get a fully developed turbulence quickly, by assuming the presence of an infinite domain upstream of the computation domain.

2) Recycling: A second approach for providing the inflow boundary condition is the pseudo-periodicity, named also recylcing. This method aims to use the velocities got at a specific position $x = x_R$ (often the outlet), by introducing it at the inlet. The classical method is to prescribe for each component i of the velocity:

$$u_i(0, x, y, z, t) = u_i(x_R, y, z, t - \Delta t) \qquad (16)$$

This method gives good results and is less expensive than synthetic turbulence. However, as explained in [15], it introduces a spurious perdiocity in the streamwise direction, which is obviously unphysical and this artificial frequency can be responsible for instabilities if it corresponds to an acoustic mode. Moreover, in a wall bounded flow, the boundary layer is slightly thicker at the outlet than at the inlet. In [14], a spanwise shift of the turbulent boundary layer is added to the velocity profile prescribed at the inlet. Indeed, if δ_0 and δ_R are respectively the boundary layer thickness at the position $x = 0$ and $x = x_R$, the inflow condition is:

$$u_i(x_0, y, z, t) = u_i(x_R, y\delta_R/\delta_0, z + \Delta z, t - \Delta t) \qquad (17)$$

where Δz is the spanwise shift introduced for avoiding complete periodicity, and $t - \Delta t$ corresponds to the previous time.

III. RESULTS

The above mentioned points have been implemented with the trunk version of TELEMAC-3D. The non-hydrostatic version is used and the advection scheme is the MURD scheme [3]. with the predictor-corrector second-order in time option. By using a implicited theta schemes as well, it allows to get closer to a second-order, that is required for performing an efficient LES. Friction laws are set up at the bottom and at the lateral boundaries.

However, the choice of a wall model is still in discussion. For these first results, the inlet boundary condition is the synthetic eddy method. The recycling is not used yet, and the subgrid model is the standard Smagorinsky model with a Van Driest damping function [16].

A. Presentation of the test case

The developments are first used with the reference from [2]. This case is a DNS of fully developed turbulent channel flow at the Reynolds number of $Re = 2340$, that has been selected in order to minimise the effect of subgrid models. Indeed, at the beginning, the global behavior of TELEMAC is examined in terms of turbulence. This case is a free surface shallow water flow in a rectangular channel of dimensions $[4\pi\delta, 3\pi\delta/2, \delta]$. The grid used has $64 \times 48 \times 65$ points. The normal stress profiles are compared to the experiments of Komori [5].

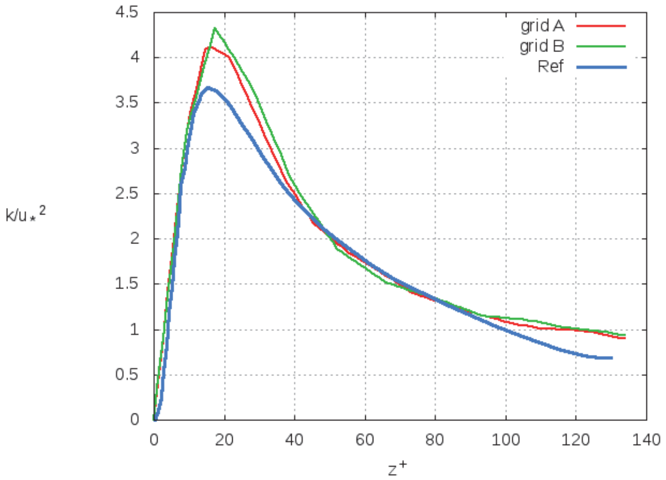

Fig. 1: Turbulent kinetic energy introduced by the SEM at the inlet plane along the centered vertical axis, from the simulations A and B and the results from [5].

In the simulations performed with TELEMAC, this case is reproduced by using two grids which settings are shown in the table I, and based on the water depth $\delta = 1/\pi$. These grids involve a polynomial distribution of the planes, in order to be refined at the boundary layer. For characterizing the friction, a Stricker law is prescribed with a value of $St = 71$, in order to get a Reynolds based on the friction velocity of about $Re_\tau = 134$.

B. Validation of the SEM

The SEM is used for synthetizing velocity fluctuations at the inlet plane, that are also transported with advection. In order to define the distrubution of these fluctuations, the Reynolds stress tensor has to be prescribed. The graphs 2, 3 and 4 show the dimensionless Reynolds stresses for each grid, as well as the turbulent kinetic energy in figure 1.

C. Steady flow

As for the synthetic eddy method, the RMS velocity components are compared to experimental data [10], and the mean streamwise flow to a theoretical log law [7].

IV. DISCUSSION

According to figures 2, 3 and 4, the fluctuations of velocity introduced by the synthetic eddy method are closed to the prescribed profiles, given by the analytical turbulent kinetic energy formulation. This expression also gives very accurate results, shown in figure 1, for the turbulent kinetic energy compared to those from the experiments from [5]. Indeed,

Run	$\Delta x = \Delta y$	$\Delta x^+ = \Delta y^+$	Δz^+_{min}	Δz^+_{max}
A	0.02	8.4	3.6	7.9
B	0.05	21.1	6.7	13.2

TABLE I: Mesh properties

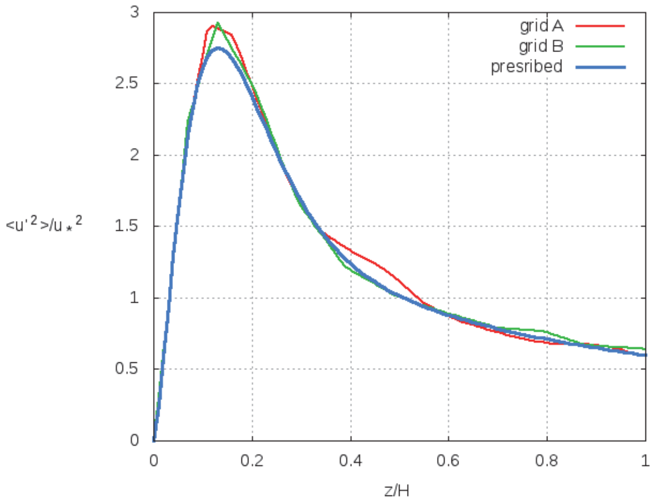

Fig. 2: Streamwise dimensionless component of fluctuacting velocity profile at the inlet plane along the centered vertical axis, from the simulations A and B et the profile prescribed in the SEM.

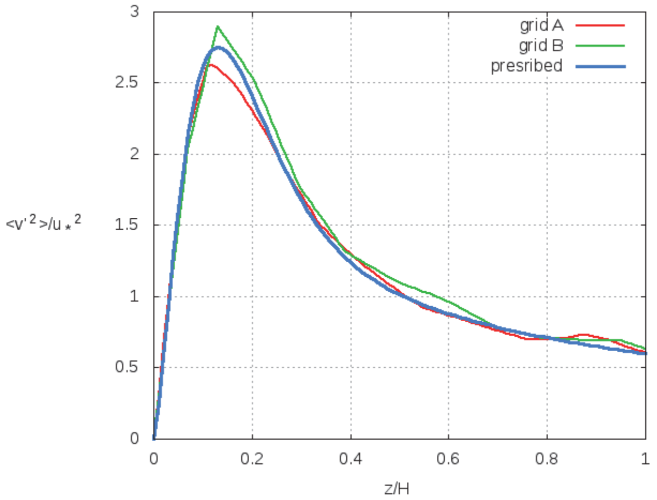

Fig. 3: Spanwise dimensionless component of RMS velocity profile at the inlet plane along the centered vertical axis, from the simulations A and B et the profile prescribed in the SEM.

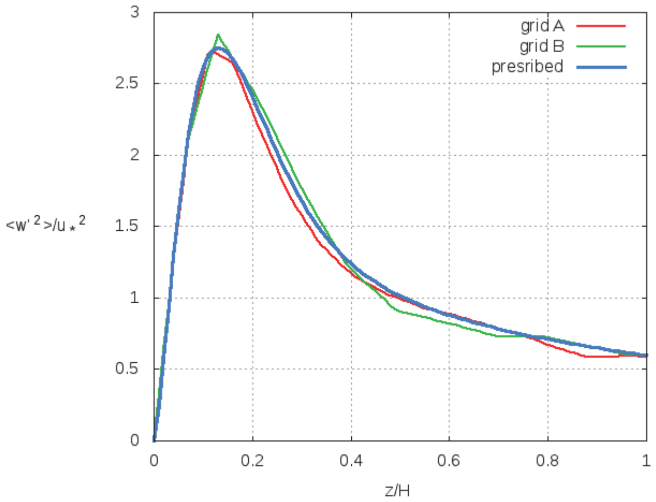

Fig. 4: Vertical dimensionless component of RMS velocity profile at the inlet plane along the centered vertical axis, from the simulations A and B et the profile prescribed in the SEM..

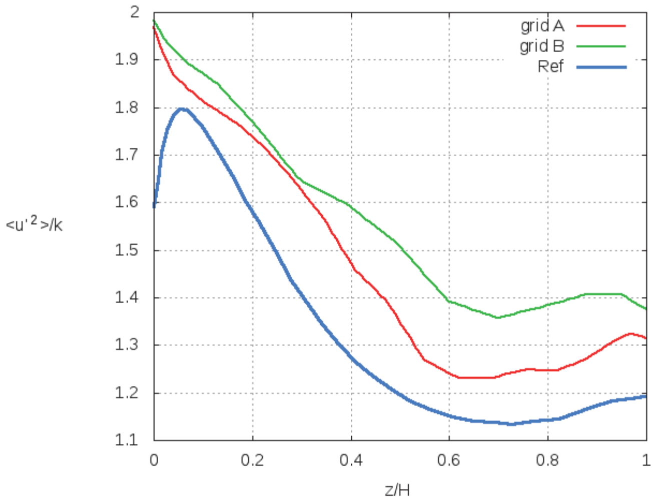

Fig. 5: Streamwise dimensionless component of RMS velocity profile in the middle of the channel along the centered vertical axis, from the simulations A and B and the results from [5].

the prescribed profile of turbulent kinetic energy is consistent with the experimental amplitude and peak at the boundary layer. Even if the turbulence is assumed to be isotropic at the inlet, it is not at all the case for this open channel flow, the anisotropic behavior is recovered very quickly in the channel. As shown in figures 5, 6 and 7, that describe the Reynolds stresses components along the centered vertical line of the computation domain, the global distribution of the fluctuations are reproduced, excepted at the bottom. Indeed, particulary for the streamwise and the spanwise velocities, the variations in the boundary layer are not well modelled, due to the friction treatment.

Furthermore, in figure 8, that is the turbulent kinetic energy profile along the vertical axis, the amplitude of the velocity fluctuations descreases sharply with the fluid progression, particularly for the coarse mesh. At this low Reynolds number, this loss of energy is quite alarming, because the subgrid model has a negligible effect in this case. This dissipation of the kinetic energy is sufficient for yielding the turbulence almost vanished by using pseudo-periodicity instead of SEM. Moreover, this loss of energy seems to be highly dependent on the way of transport of the synthetic eddies in the virtual box. Further investigations will be carried out to quantify the numercal dissipation rate in TELEMAC-3D and to check the effect of the friction treatment on these turbulent fluctuations.

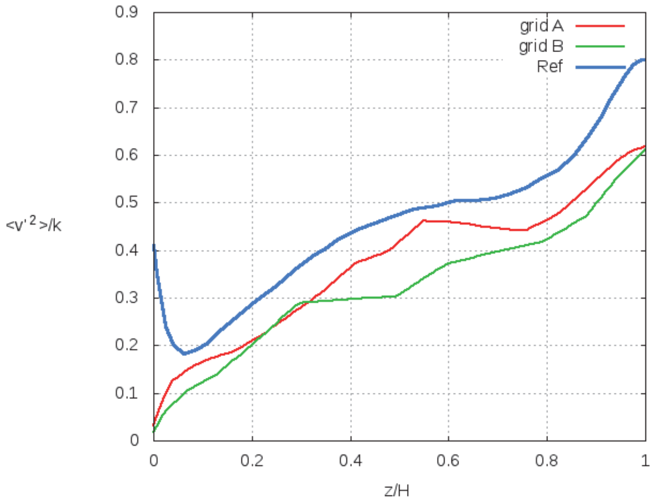

Fig. 6: Spanwise dimensionless component of RMS velocity profile in the middle of the channel along the centered vertical axis, from the simulations A and B and the results from [5].

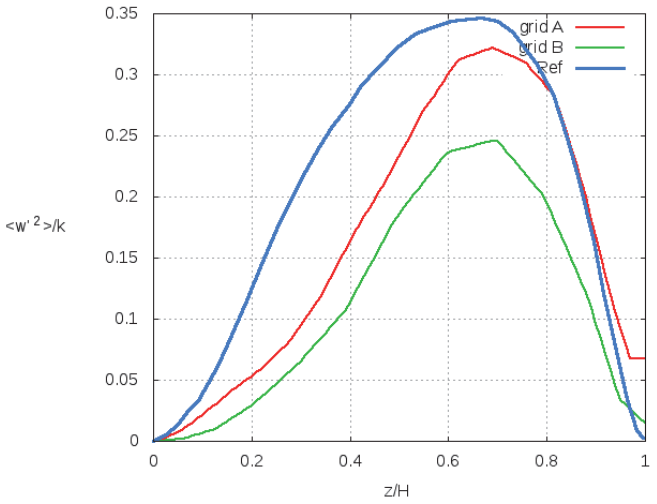

Fig. 7: Vertical dimensionless component of RMS velocity profile in the middle of the channel along the centered vertical axis, from the simulations A and B and the results from [5].

V. CONCLUSION

A Large-Eddy-Simulation (LES) approach is developed in TELEMAC-3D. After carrying out a state of the art of LES methods in hydraulics, two subgrid models are selected to be implemented. Since this kind of simulation requires specific boundary conditions, the Synthetic Eddy Method (SEM) is used at the inlet boundary for generating velocity fluctuations, and a wall model is being discussed.

The first developments allow us to get preliminary results. The SEM is a good alternative to a precursor simulation, since it can introduce accurately velocity fluctuations following a prescribed Reynolds stress tensor with a low computation cost.

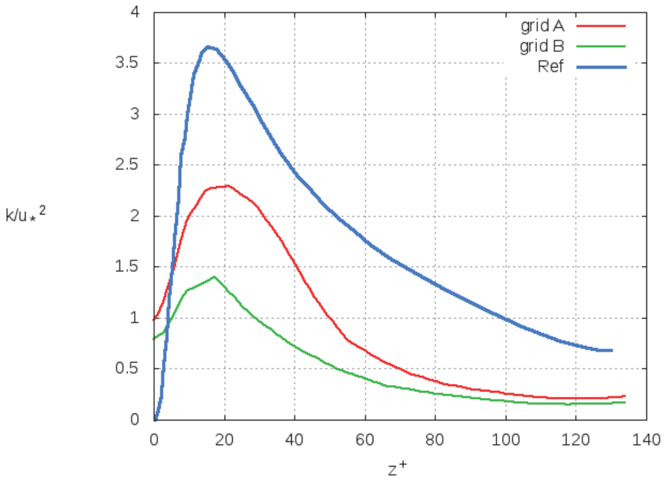

Fig. 8: Turbulent kinetic energy profile in the middle of the channel along the centered vertical axis, from the simulations A and B and the results from [5].

So, by defining a simple analytical turbulence kinetic energy profile, it quickly leads to a fully developed turbulence flow.

The global behavior of our turbulence indicators are satisfactory but the first results show that the turbulent kinetic energy decreases faster than expected with the flow progression. Indeed, a wide part of turbulence intensity introduced by the SEM is lost from the very start of the fluid progression.

REFERENCES

[1] Massimo Germano, Ugo Piomelli, Parviz Moin, and William H Cabot. A dynamic subgrid-scale eddy viscosity model. *Physics of Fluids A: Fluid Dynamics (1989-1993)*, 3(7):1760–1765, 1991.

[2] RA Handler, TF Swean, RI Leighton, and JD Swearingen. Length scales and the energy balance for turbulence near a free surface. *AIAA journal*, 31(11):1998–2007, 1993.

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

[4] Nicolas Jarrin, Sofiane Benhamadouche, Dominique Laurence, and Robert Prosser. A synthetic-eddy-method for generating inflow conditions for large-eddy simulations. *International Journal of Heat and Fluid Flow*, 27(4):585–593, 2006.

[5] Satoru Komori, Hiromasa Ueda, Fumimaru Ogino, and Tokuro Mizushina. Turbulence structure in stably stratified open-channel flow. *Journal of Fluid Mechanics*, 130:13–26, 1983.

[6] Arthur G Kravchenko, Parviz Moin, and Karim Shariff. B-spline method and zonal grids for simulations of complex turbulent flows. *Journal of Computational Physics*, 151(2):757–789, 1999.

[7] CW Li and JH Wang. Large eddy simulation of free surface shallow-water flow. *International journal for numerical methods in fluids*, 34(1):31–46, 2000.

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

[9] C Mellen, J Fröhlich, and W Rodi. Large-eddy simulation of the flow over periodic hills. *16th IMACS world congress*, 2000.

[10] Iehisa Nezu, Hiroji Nakagawa, and Gerhard H Jirka. Turbulence in open-channel flows. *Journal of Hydraulic Engineering*, 120(10):1235–1237, 1994.

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

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

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

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

[15] A Spille-Kohoff and Hans-J Kaltenbach. Generation of turbulent inflow data with a prescribed shear-stress profile. Technical report, DTIC Document, 2001.

[16] Edward R Van Driest. Turbulent boundary layer in compressible fluids. *Journal of spacecraft and rockets*, 40(6):1012–1028, 2003.