International Conference on Hydroinformatics

2014

Method Of Use And The Contribution Of LES 3D Modeling To Reproduce Velocity Distribution In Compound Channels

Salma Bellahcen

José Vazquez

Matthieu Dufresne

Robert Mose

Gilles Isenmann

11th International Conference on Hydroinformatics
HIC 2014, New York City, USA

METHOD OF USE AND CONTRIBUTION OF LES 3D MODELING TO REPRODUCE VELOCITY DISTRIBUTION IN COMPOUND CHANNELS

SALMA BELLAHCEN (1), JOSE VAZQUEZ (1,2), MATTHIEU DUFRESNE (1,2), ROBERT MOSE (1,2), GILLES ISENMANN (1)
(1): ICube, Mechanics Department, Fluid Mechanics Team, 2 rue Boussingault 67000 Strasbourg
(2): National School for Water And Environmental Engineering of Strasbourg (ENGEES)

Discharge determination in open channels is of great interest for water engineering. In compound channels, discharge determination raises the issue of the link between the measured velocity and the mean velocity. To do so, Computational Fluid Dynamics is used as a tool to determine the velocity distribution in compound channels. The purpose of this study is to compare through numerical simulations the most common turbulence models such as k-ω and RSM to LES that is rarely used in this field in order to investigate the efficiency of the LES model to reproduce the velocity distribution.

Introduction

Compound open-channel flow which consists of a main channel and a floodplain is often observed in river but also in sewer networks. The main difference between the compound channel in rivers and those in sewer networks is that in sewers, the floodplain is thinner than the main channel because it is only used by staff during their routine maintenance in dry-weather conditions.

In compound channels the characteristics in the cross section such as the distribution of the mean velocity and the pattern of secondary currents are very complicated. To clarify those characteristics, many researchers have investigated them by making use of experiments such as Proust & Rivière [8], Tominaga & Nezu [10] or Nezu [6].

In this study, the distribution of the mean velocity and the secondary currents are evaluated using 3-D numerical simulations. The aim of this paper is to compare different simulations (RSM, K-ω and LES) in order to determine the most suitable one for a good representation of the flow in a compound channel. To do so, Nezu [6] was chosen as the experimental case that will be modeled. Nezu [6] investigated 3-D turbulent structures such as the mean velocity and the secondary currents by making use of a two-component LDA.

A series of simulations have been done using two different softwares: Ansys Fluent® and OpenFoam® and three different numerical simulations: k-ω, RSM and LES.

Theoretical considerations

In order to proceed with the model results and comparison, it is important to really understand the different turbulence model and closure methods used in this study.

Large Eddy Simulation (LES)

There is many ways to solve the transport and the Navier Stokes equations. The easiest one is the RANS approach where the velocity and the pressure are split into a mean value and a fluctuation whose influence is modeled using a so-called turbulence model. The advantage of this method is that it does not need a high computing power but one must keep in mind that it is just a model and some information can be lost or wrongly modeled.

The most efficient way to solve the Navier Stokes equation is the DNS (Direct Numerical Simulation) where the equations are directly solved with a time step relevant to turbulence phenomena. This method is certainly efficient but it requires a high computing power due to the heavy grid that it requires.

Large Eddy simulation is an intermediate approach between DNS and RANS. This method consists in solving directly the large three-dimensional unsteady turbulent motions that are the large scales, and modeling through a sub-grid model the small scales. Filtering the scales is advantageous because the large scales depend on the boundary conditions and dominate the heat and the momentum transfer which is usually what one wants to model. The small scales are modeled as they are isotropic and more universal in nature [7].

However, it is critical to be very accurate with the turbulence behavior while modeling with LES because it is strongly influenced by the boundary conditions. As the turbulence is often wrongly informed, the simulation can take a long time before reaching the boundary conditions and converging to an acceptable solution.

To deal with this problem two methods were found in the literature. First, the synthetized turbulence method was proposed by Castro and al [3] where a random field of velocity and time scale is injected in the flow. The turbulence can then be generated through many technics as the Fourrier method or the POD (Proper Orthogonal Decomposition) (Berkooz [2]).

The second method is the recycling method by Jorgensen [4]. This method consists in projecting turbulence data from a plane located downstream to the inlet. This method is easier than the first one because the projection of the data is done during the same simulation.

In this study, the authors chose the second method.

Reynolds stress model (RSM)

RSM is one of the most elaborate turbulence model provided by Ansys Fluent® [1]. Unlike the (for example) k-epsilon model, RSM does not use the isotropic eddy viscosity hypothesis but solves the transport equations for Reynolds stresses coupled with an equation for the dissipation rate.

This turbulence model has a great potential to give good predictions for complex flows since it takes into account the effect of streamline curvature, swirl and rotation in a better way than the one and two equation models.

RSM in Ansys Fluent® requires boundary conditions for individual Reynolds stresses, ui'uj', and for the turbulence dissipation rate, ε. The near-wall treatment used is the standard wall function.

k-ω model

The k-ω model is widely used and it is based on turbulent kinetic energy equations and dissipation rate. It forecasts consistent results for simple shear flow in particular. Nevertheless it is dependent locally and linearly on Reynolds tensions and on average field. It is poorly suited for complex flows (e.g. recirculation or anisotropy). For this study the model used will be the k-ω sst. It is a new model which presents a real asset. It was developed by Menter [5] to effectively blend the robust and accurate formulation of the k-ω model in the near-wall region with the free-stream independence of the k-ε model in the far field. Indeed it has the same advantages than the k-ω model when close to the wall, and the same than k-ε far from the body.

Experimental case

The experimental case chosen for the modeling is the one investigated by Nezu [6]. It is indeed the study that provided the most complete results concerning the velocity distribution in compound channels.

The objective of this study is to determine the velocity profile and the secondary currents in a transversal section of a compound channel.

The experiments were conducted in a 10 m long, 40 cm wide and 50 cm deep channel (Figure 1). The velocity was measured with a four beam LDA system located at 7 m downstream of the channel entrance. The hydraulic conditions are shown in (Table 1). One should notice that the experiments were carried out at low Reynolds numbers (less than 23,000).

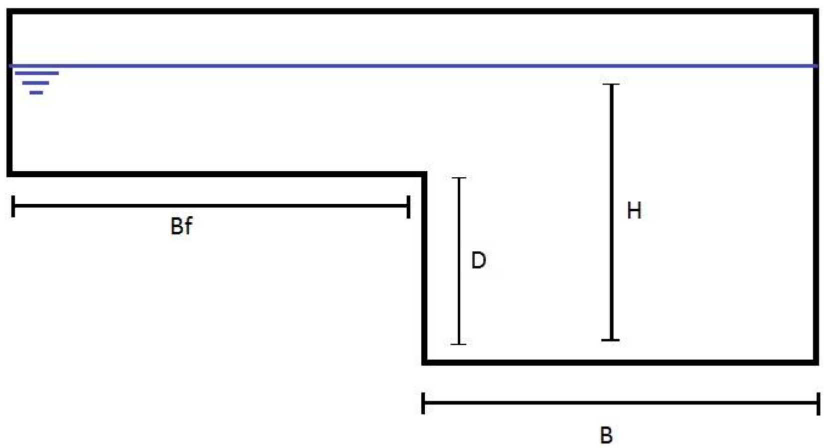

Figure 1. Geometry of the experimental channel

Table 1. Hydraulic conditions of the experimental case

Hydraulic conditions									
case	I	H (cm)	B (cm)	Bf/B	H/D	Q (l/s)	Umax (cm/s)	Fr	Re $x10^4$
H6	0.300%	6	20	0.5	1.2	1.44	13.4	0.2	1.3
H7	0.025%	7	20	0.5	1.4	2.06	14.9	0.2	1.8
H8	0.020%	8	20	0.5	1.6	2.73	15.6	0.2	2.3

Numerical conditions

The numerical simulations were conducted using three turbulence models: RSM, k-ω sst and LES. In order to choose the most accurate grid, a mesh sensitivity analysis has been done using the GCI method by Roache [9]. Three grids M1, M2 and M3 (cited from the coarsest to the finest) were compared. The value of GCI was around 11% between M2 and M1 and near to 0% between M3 and M2. Naturally, the chosen grid was M2 (around 50 000 cells).

Results and comparison

The experimental results show that the velocity bulges from the core towards the corner of the main channel. The isovel lines are very close in the junction between the main channel and the floodplain.

In order to quantify the difference between the experimental and numerical results, it is essential to calculate an error at each measurement point. To do so, a percentage of error is calculated at each point (y,z) and for each case (H6, H7 and H8). (Figure 2) and (Figure 3) represent the calculated errors for the first case simulated with RSM and k-ω sst. Each colored spot represents a percentage of error between the between the experimental result and the numerical one. The author chose 20% as a tolerance limit. Underneath this value, one can consider that the results are similar and above it, they are considered to be significantly different. If the percentage or error is greater than 50%, the figures show a red spot. The colored scale represents the percentage of error from 0% (blue) to 50% (red).

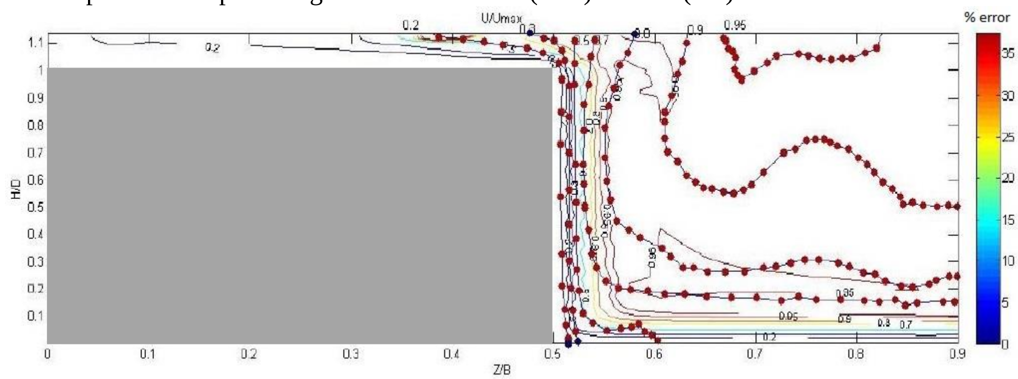

Figure 2. Case H6 – RSM: Percentage of error between the numerical data and the experimental one

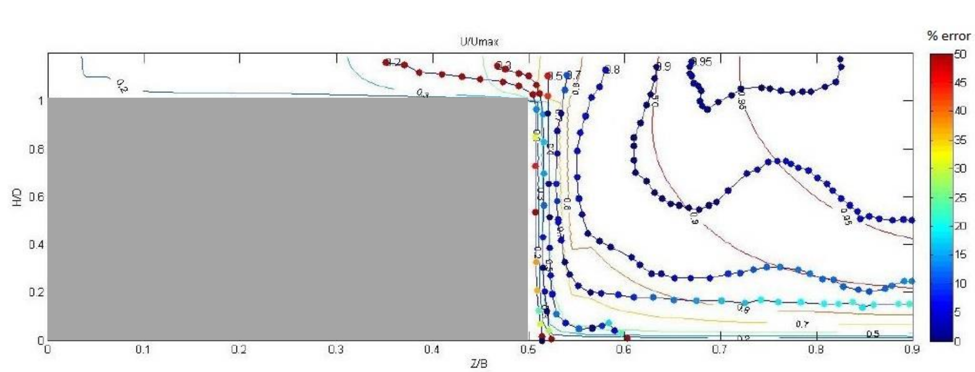

Figure 3. Case H6 – k-ω sst: Percentage of error between the numerical data and the experimental one

The comparison with the RSM shows a high percentage of error. We can say that in this case, RSM does not represent correctly the velocity distribution. In opposition, k-omega sst presents satisfying results. For the isovel-lines located in the main channel, the percentage of error is underneath 20%, which is very satisfying taking into account the uncertainties related to the experimental measure, the sensor used and the digitization of the graphics. In the floodplain, one can notice that the percentage of error is higher than 50%, this model is not efficient in this area.

Another interesting parameter to compare is the secondary currents. This parameter concerns mainly the RSM and LES as the k-omega sst model is a first order model (completely unable in nature to model secondary currents). (Figure 4) shows the velocity vectors of the secondary currents V and W normalized by the maximum mean velocity Umax for the third case of Nezu's experiment modeled using RSM. The pattern of secondary currents is quite similar to what is expected. In the junction between the main channel and the floodplain, the velocity decreases and a recirculation of the secondary currents takes place.

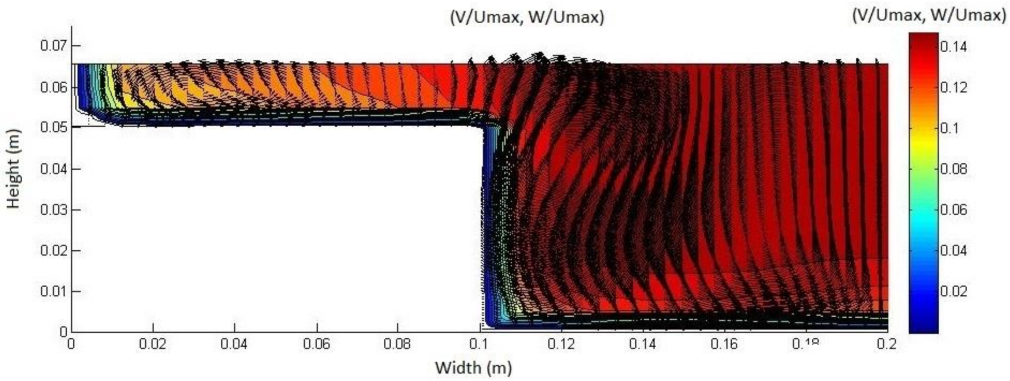

Figure 4. Secondary currents - case H8, RSM

A third series of simulations has been done with LES for the first case only. The interest of using LES is that it provides results for each time step.

The comparison with the experimental results (Figure 5) is done in the same way as the previous ones. As the LES gives the information in each time step, the reader should know that in order to obtain such results, it is necessary to calculate the mean value of each parameter.

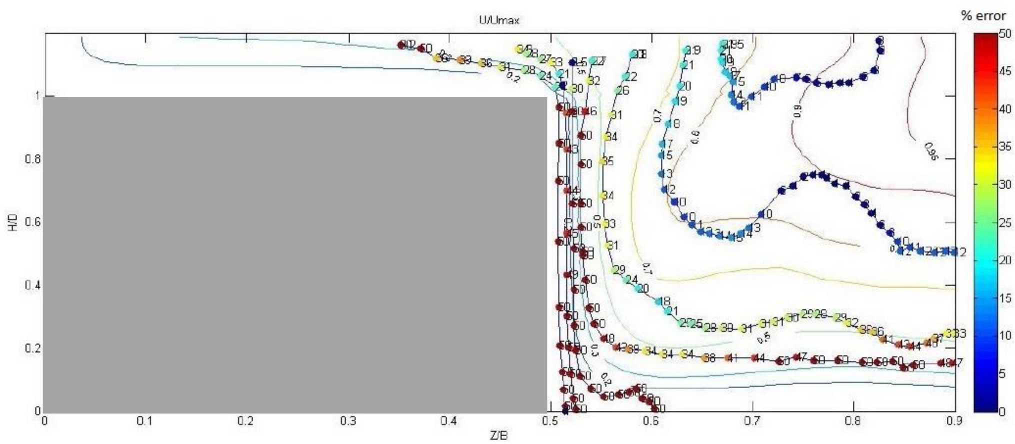

Figure 5. Comparison of the isovel lines, case H6, LES

The results of the LES simulation are very close to the experimental ones. Unlike the other comparisons, one can see that this model gives a low percentage of error in the main channel but also in the floodplain. However, the velocity distribution is erroneous in the region near the main channel wall and at the bottom. This is certainly due to the fact that the model could not handle the development of the boundary layer in this exact area.

In order to see the behavior of the turbulence and the velocity, a LES simulation has been done. (Figure 6) represents the evolution of the velocity and the emergence of turbulence during the time.

The reader should know that the simulations were done for a symmetric compound channel but the measures of the main channel and the floodplain corresponds to the experimental case (Figure 7)

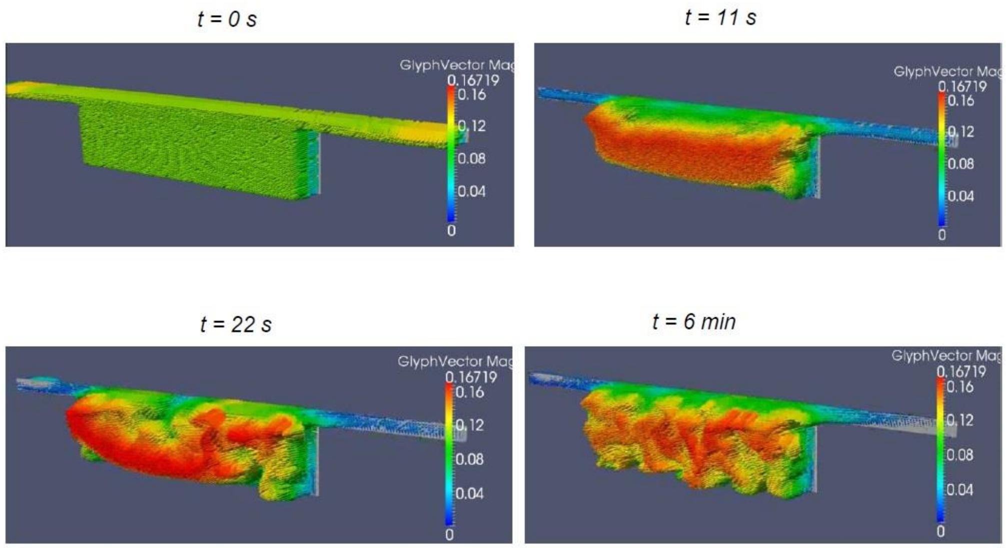

Figure 6. Emergence of turbulence in a compound channel- case H6, LES Smagorinsky

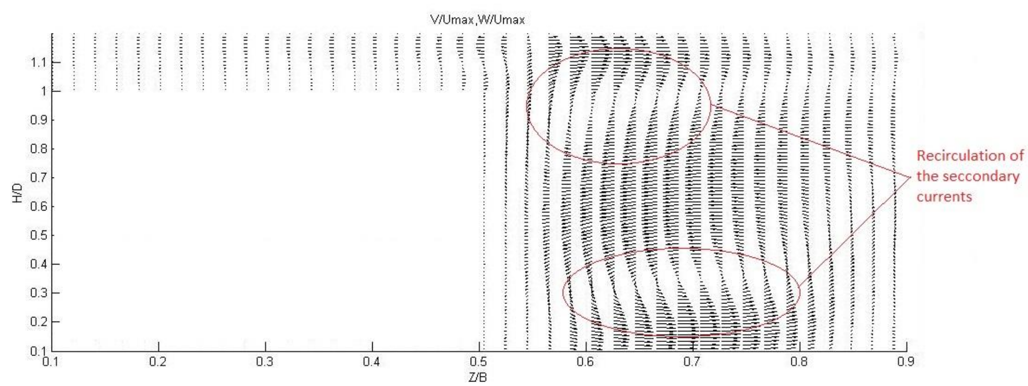

Figure 7. Secondary currents with average velocity- case H6, LES

One can first observe that the velocity distribution fluctuates during the time. The turbulence grows mainly in the junction between the main channel and the floodplain.

The most interesting part of this simulation is that the evolution of the secondary currents during the time does not behave at all like what one saw for other simulations. One does not observe the recirculation but instead the secondary currents are very instable and disordered. After the calculation of the mean value of each parameter, one can observe that a recirculation of the secondary currents appears at exactly the same area than in the experimental case and the simulation with RSM.

Conclusion

Velocity distribution in compound channel was modeled using two softwares and three turbulence models. The study has shown that for such low Reynolds numbers and unlike k-ω sst and LES, RSM is not efficient to reproduce the velocity distribution.

In addition to the good representation of the velocity distribution, LES has the advantage to give information in every time step which reveals an unexpected behavior of the secondary currents. After averaging the velocity the shape of the secondary currents corresponds to what was expected. LES can then be a great tool to model the velocity in a compound channel flow.

REFERENCES

[1] Ansys Fluent Theory Guide, (2011) http://cdlab2.fluid.tuwien.ac.at/LEHRE/TURB/Fluent.Inc/v140/flu_th.pdf

[2] Berkooz G., Holmes P., and Lumley J.L., "The Proper Orthogonal Decomposition in the Analysis of Turbulent Flows", *Annual Review of Fluid Mechanics,*Vol. 25 (1993),pp. 539-575

[3] Castro H., Paz R. and Sonzogni V., "Generation of turbulent inlet velocity conditions for large eddy simulations", *Mecanica computational*, Vol. 30 (2011), pp. 2257-2288.

[4] Jorgensen, N. G. "Implementation of a turbulent inflow boundary condition for LES based on vortex method", *A course at Chalmers University of Technology*, Taught by Hakan Nilsson, (2012)

[5] Menter F.R, "Review of the SST turbulence model exoerience from an industrial perspectice", *International journal of computational fluid dynamics*, Vol 23 (2009), Issue 4.

[6] Nezu I. et al, "Secondary currents and bed shear stress in compound open-channel flows with shallow flood plain", *XXVIII Congress IAHR* (1999).

[7] OpenFoam user guide, http://www.openfoam.org/features/turbulence.php, (2011)

[8] Proust S. et al, "Turbulent non-uniform flows in straight compound open-channels", *Journal of Hydraulic Research*, Vol. 51, No. 6, (2013), pp. 656-667.

[9] Roache, P. J, "A Method for Uniforme Reporting of Grid Refinement Studies". *Journal of Fluids Engineering*, Vol. 116 (1994), pp. 406-413.

[10] Tominaga A., Nezu I., "Turbulent structure in compound open-channel flows", *Journal of hydraulic Engineering*, Vol. 117, No. 1, (1991), pp. 21-41.