

Three-dimensional model of detailed hydrodynamics for simulation of subaqueous dunes

M. Abdu Nabi

Hydraulic Engineering Section, Delft University of Technology, 2628 CN Delft, The Netherlands

ABSTRACT: We are developing a numerical model for simulating the development and migration of dunes in rivers in a 3D case. The numerical model consists of three steps: turbulent flow, sediment transport and morphology. We present the first step in which we modelled turbulent flow using state-of-the-art techniques for detailed hydrodynamics. A finite volume method combined with an isotropic unstructured Cartesian grid with local refining is developed for simulating time-dependent incompressible flow. The grid can be refined and adapt to the boundaries. The governing equations are discretized using a staggered grid and advance in time using the fractional step method. The Cartesian grid cells and faces are managed using an unstructured data approach. A ghost-cell immersed-boundary technique has been implemented for the cells which intersect the immersed boundaries. Because of the importance of coherent structures of turbulence on sediment transport, the turbulence regime is modelled by Large Eddy Simulation.

1 INTRODUCTION

The water levels during river floods, and hence the risk of flooding, depend on the hydraulic roughness of the river. One of the components of this roughness is produced by statistically periodic irregularities on the river bed called “dunes”. The development of dunes and the associated hydraulic roughness during a flood is complex. Initially dunes grow higher and make the river bed rougher, but in later stages the dunes grow longer with the opposite effect of making the river bed smoother. Subsequently, in a way still not well understood, new bedforms develop on top of the elongated dunes that make the river bed rougher again.

The sediment transport field over dunes and ripples in open-channel flows is strongly affected by the complex turbulence field caused by flow separation at the dune crest. The three-dimensionality of turbulence and the effect of turbulence on the sediment transport and morphological process form a complex problem which is not completely solved yet. At present, there is still limited knowledge about the effect of dunes on the hydraulic roughness of the rivers. Several researchers proposed methods to predict the dune dimensions based on parameterization methods, empirical relations (Van Rijn 1984, Julien & Klaasen 1995) and theoretical interpretations (Onda & Hosoda 2004). Existing experimental studies are limited to the formation of dunes on steady

state flow regimes (Blom et al. 2003). Wilbers (2004) has shown none of these predictors are able to predict correctly the dune dimensions during several floods in the River Rhine in the Netherlands. He developed an empirical method to predict dune development for unsteady flows. This method is applied successfully to three sections of the River Rhine branches, but it can not be generalized. The method gives limited knowledge about the physical phenomena behind dune development during floods.

Existing numerical studies on the morphology are limited to the approximation of the two-dimensional fluid flow, with two-dimensional dunes development (Shimizu et al. 2001, Giri et al. 2006) or with fixed bed (Yoon et al 1996, Zedler et al. 2001, Yue et al. 2006). The nature of flow over 3D dunes is very different from the flow in many studies that have concerned 2D dunes; to the degree that the application of some of these 2D studies to the field requires careful attention (Best 2005). Field observations suggest an urgent requirement for a fuller analysis of dune three-dimensionality in both laboratory and numerical studies.

We are developing a numerical model for simulating the development and migration of dunes in rivers in a 3D case. The numerical model consists of three steps: turbulent flow, sediment transport and morphology. We present the first step in which we modelled turbulent flow using state-of-the-art techniques for detailed hydrodynamics. The development of dunes is directly influenced by the fluid flow. A

correct prediction of dunes and migration of bedform requires an accurate prediction of fluid flow. The dunes have also a direct effect on the drag and hence on the fluid flow regime. It is important to calculate the fluid in its details close to the bedform. An accurate solution for the dunes can be achieved by a high-resolution grid close to the boundaries. Structured Cartesian grids are not suitable for this because the solution of a fully or partly fine Cartesian grid can be very expensive. Here, a finite-volume method combined with an isotropic unstructured Cartesian grid with local refining is developed for simulating time-dependent incompressible flow. The grid can be locally refined and adapt to the bedform. The governing equations are discretized using a staggered grid and their solution is advanced in time using the fractional step method. The Cartesian grid cells and faces are managed using an unstructured data approach. A ghost-cell immersed-boundary technique is implemented for the cells which intersect the immersed boundaries. Because of the importance of coherent structures of turbulence on sediment transport, the turbulence regime is modelled by Large Eddy Simulation.

We outline the two steps that will be taken subsequently. The second step concerns the modelling of sediment transport which may involve new concepts that are better suited for relatively small spatial and temporal scales. The third step concerns a morphology model for bedform growth, decay and migration. Parameterized relations will be used to determine the pick up and settling of sediment and thus to estimate the deformation and development of dunes.

The model aims at giving better insight into the development of the hydraulic roughness, and hence the flooding risk, during river floods. The good understanding thus obtained will allow the development of parameterized models for larger spatial and temporal scales that can be used in operational models for flood early warning systems and the determination of design water levels.

2 THE GOVERNING EQUATIONS

The governing equations for the fluid are the full three dimensional Navier-Stokes equations for incompressible flow with the Boussinesq approximation invoked. These equations are given below in terms of volume filtered variables.

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{1}{\rho_0} \frac{\partial \bar{P}}{\partial x_i} + \frac{\partial}{\partial x_j} \{2(\nu + \nu_t) S_{ij}\} \quad (2)$$

where x_i are the coordinates, t is the time, \bar{P} the modified pressure, ρ_0 the density, \bar{u}_i the filtered velocity component in x_i -direction, ν and ν_t the mo-

lecular and turbulent viscosities and S_{ij} is the resolved strain rate tensor:

$$S_{ij} = \frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \quad (3)$$

In Large Eddy Simulation, the large eddies is solved directly, ignoring the smaller eddies. The smaller eddies are then modelled. A volume filtering is used in LES, allows filter the eddies which are smaller than the grid cell volume. The filter is defined as

$$\mathbf{f}(\mathbf{x}, t) = \int \mathbf{f}(\mathbf{y}, t) G_\Delta(\mathbf{x} - \mathbf{y}) d\mathbf{y} \quad (4)$$

The effect of the small scales upon the resolved part of turbulence appears in the SGS stress term

$$\tau_{ij} = \overline{u_i u_j} - \bar{u}_i \bar{u}_j \quad (5)$$

which must be modelled.

The SGS effect is modelled using the Smagorinsky's model (Marcel 2005) in which the turbulent viscosity is defined as

$$\nu_t = (C_s \Delta)^2 |\bar{S}| \quad (6)$$

where

$$|\bar{S}| = \sqrt{2 \bar{S}_{ij} \bar{S}_{ij}} \quad (7)$$

$$\Delta = (\Delta x \Delta y \Delta z)^{1/3} \quad (8)$$

The Smagorinsky's coefficient C_s has been chosen 0.1.

3 NUMERICAL METHODS

3.1 Data structure

The conventional structured-grid approach for simulating the flow with complex immersed boundaries is the curvilinear grid method. At this method the governing equations are discretized on a structured body-fitted grid. The advantage of this method is that imposition of boundary conditions is simplified. The boundary conditions can be imposed with their highest accuracy. But on the other side, it has serious drawbacks. Related to the complexity of the immersed boundaries, the generation of a good quality grid can be sometimes impossible without multiblock techniques. Furthermore, the transformation of the governing equations to the curvilinear system yields a complex set of equations, instabilities and convergence problems. Additionally, for the moving boundaries (river bed) a new grid is necessary for every time step. In each time step three transformed Poisson equations must be solved and mapped to the computational space. It is a time consuming process.

A different approach which is gaining popularity in the recent years is the Cartesian grid method. At this method the governing equations are discretized on a Cartesian grid which cannot fit the immersed

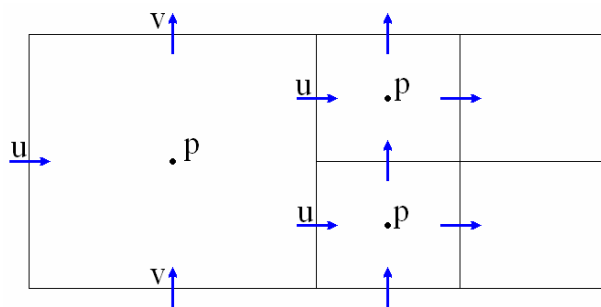


Figure 1. Neighbour cells with different sizes in 2D. The pressure is located in the centres of cells and the velocities on the centres of faces.

boundaries. Cut-cell technique and Ghost-cell methods are the most popular remedies for this problem.

At the cut-cell technique, the intersecting cells are cut, yielding arbitrarily shaped cells, which add complexity to the computational model. The Ghost-cells method adds a force on the immersed boundaries. It is easy to implement and requires less computational efforts than the cut-cells techniques.

A simple structured Cartesian grid requires a large number of cells to capture the small eddies in a turbulent flow. In order to optimize the use of computational resources, we use an adaptive Cartesian mesh with local refinement, in which more grids can be placed in high gradient regions. A multi-level grid is a suitable choice for our purpose. At the multi-level grid, the levels with fine cells are placed close to the boundaries, and the coarse levels are related to the low gradients regions.

An isotropic graded Cartesian grid is used for this purpose. In isotropic Cartesian grids, the refining of each cell occurs in all directions. The refined cell has 8 children in 3D or 4 children in 2D. Corresponding to this refining, any cell has one or four faces on its sides in 3D (one or two faces in 2D). The velocities are defined in the centre of the faces. Figure (1) illustrates a refining cell in the two-dimensional case. Graded Cartesian grid means that two neighbour-cells cannot differ more than one level of refinement.

The set of locally refined Cartesian cells are commonly managed in two ways: the hierarchical tree data structure and the fully unstructured approach. The tree data structure (parent-child tree) requires a tree-traverse approach to determine neighbour connectivity based on logical recursive routines. The calculation time required for determining the neighbours can be considerably larger than the fully unstructured approach. Here we use the fully unstructured approach. In the fully unstructured approach the neighbours are defined by pointers and determined directly.

Finally we point out that the solution of viscous incompressible flow needs considerable attention to avoid the checkerboard effect. A staggered grid is used, where the pressure is defined on the centre of cells and the velocities on the corresponding faces.

Figure (1) shows the pressure and velocities related to two neighbours with different sizes.

3.2 Discretization

The numerical method for the present contribution uses a staggered grid method, with the pressure located in the centre, and the velocities on the faces of cells. The governing Navier-Stokes equations for unsteady incompressible flow are discretized using finite-volume method with second order, linear discretizations for fluxes. Time advancement uses the frictional step method, which decouples the solution of the velocity field from the pressure. The second-order Adams-Bashforth method is used for convection and diffusion terms. The time step is determined by the CFL condition

$$\Delta t = \min_{\Omega} \frac{\beta}{\left| \frac{u}{\Delta x} \right| + \left| \frac{v}{\Delta y} \right| + \left| \frac{w}{\Delta z} \right| + \frac{1}{\text{Re}} \left(\frac{1}{\Delta x^2} + \frac{1}{\Delta y^2} + \frac{1}{\Delta z^2} \right)}$$

where the constant β is set to 0.35 as a safety factor.

The Poisson pressure correction equation is solved by the BiCGStab method using zero-order incomplete LU factorization. Ghost-cell immersed boundary techniques are used to conform the dune geometry (Balaras 2004, Yang 2006).

The accuracy of the numerical models using the unstructured Cartesian grids with local refining is no longer of the second order for the standard discretization on the uniform grid. The accuracy decreases to the first order. This pollution in discretization happens on the interfaces between the different levels and has effect on all the solution. To avoid the pollution in our model, the variables are interpolated with the neighbour cells to ensure a second order accuracy. Figure (2) illustrates a linear interpolation for the pressure with neighbour cells. Interpolations for the velocity component are done straightforwardly.

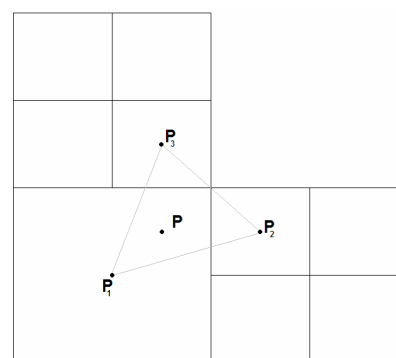


Figure 2. Linear interpolation for the pressure with the neighbouring cells.

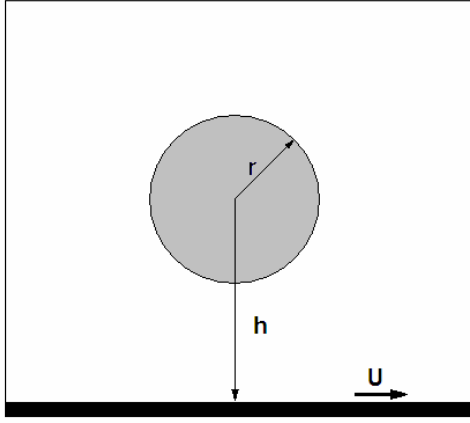


Figure 3 Wannier flow. The computational domain.

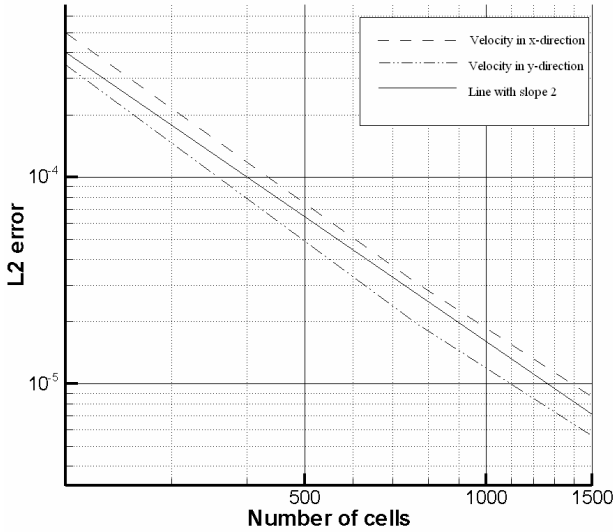


Figure 4 L_2 norm of velocities. The solid line is the line of slope 2.

4 NUMERICAL EXPERIMENTS

4.1 Wannier flow

A straightforward way to verify the second-order spatial accuracy of the present model is to compute a flow which has a curved immersed boundary and for which an analytical solution exist. To demonstrate the accuracy and efficiency of the proposed methodology, two-dimensional Stokes flow past a cylinder in vicinity of a moving wall has been considered. Figure (3) shows the computational domain of this flow. This flow was solved exactly by Wannier (1950). We simulated this flow by our solver on a three-level grid for different numbers of cells. The grids are similar in their shapes. The boundary conditions are given exactly from the existing analytical solution. The L_2 norm of the error is defined as

$$L_2 = \left(\frac{1}{N} \sum_{i=1}^N (u_i^{\text{numerical}} - u_i^{\text{exact}})^2 \right)^{0.5} \quad (9)$$

where N is the number of cells. The norm for the components of velocity vectors is plotted in figure 4

which shows that the norm of the velocities decreases by a slope of 2. It means a second-order accuracy in our discretization.

4.2 Flow over dunes

In this section the results of LES over two-dimensional dunes are presented. The dunes are considered as a sinusoidal wavy form with wave height and length of 2δ and λ respectively. A wave slope $2\delta/\lambda = 0.1$ is considered, that exhibits a large separated region between consecutive crests. The maximum height of the channel is $H = \lambda$ and the size of streamwise and spanwise directions have to be considered as 2λ . The mean channel height is used as the length scale and the bulk velocity as the velocity scale. The bulk velocity is defined as the average velocity in x -direction in a plane parallel to $y-z$ plane at a position with a height of $H - \delta$.

$$U = \frac{1}{(H - \delta)L_z} \int_A \langle u \rangle dA \quad (10)$$

where L_z is the length of the spanwise direction and considered as 2λ . $\langle u \rangle$ is the time average velocity. The Reynolds number has been set 6760 based on the length scale H and velocity scale U . This simulation is similar to the DNS simulation by Maaß & Schumann (1996) and the LES simulation by Balaras (2004).

Periodic boundary conditions have been considered in streamwise and spanwise directions. On the roof, a non-slip boundary is applied. A Ghost-cell immersed boundary method is used to enforce the non-slip condition on the dunes.

The present problem is simulated using a grid with initial cells of $48 \times 22 \times 16$ in streamwise, normal and spanwise direction, with three refining levels. It means the size of the smallest cells in x -direction is

$$\Delta x_{\text{finest}} = \frac{\Delta x_0}{2^{(3-1)}} = \frac{\Delta x_0}{4} \quad (11)$$

Because of isotropy of the grid, the sizes in y - and z -directions are $\Delta y_0/4$ and $\Delta z_0/4$. Δx_0 , Δy_0 and Δz_0 are the sizes of a cell on the initial grid (without refining). The finest cells are located in the vicinity of the non-slip boundaries. Figure (5) illustrates the solution domain and the grid with two levels of refining (three-level grid).

The flow is driven by a pressure gradient in x -direction. After each time step, the bulk velocity is calculated and the pressure gradient is determined in a way that the bulk velocity remains unit.

Figure (6) to (9) show the average velocity profiles in four different intersections. The intersections are parallel to the $y-z$ plane and located at $x/h = 0.2, 0.6, 0.8$ and 1.0 . h is defined as the half-height of the mean channel. The first intersection is located just after the wave crest but outside

the recirculation area. The next two are in the recirculation zone. The last station is just before the wave

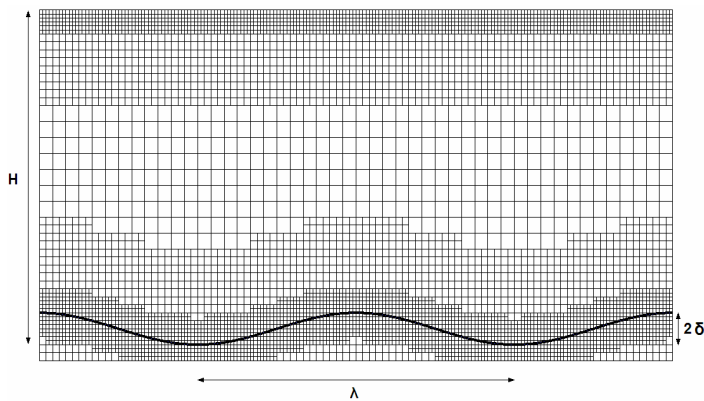


Figure 5 The computational domain and 3-levels grid.

crest at $x/h=1.4$. The length is normalized by λ and the velocities are normalized by bulk velocity U . The results are compared with the DNS simulation of Maaß & Schumann (1996) and the LES of Balaras (2004). From the figures it is clear that the present results have a good agreement with the solution of previous works. There is a small deviation in the boundary layer in figures (7) and (9). This is probably because the difference between the models used for Large Eddy Simulation in the present work and in Balaras's work. Balaras (2004) uses a subgrid dynamics model based on a Lagrangian procedure to determine the coefficient C . At the present work, the classic Smagorinsky's model with constant coefficient is applied.

5 CONCLUSIONS AND FUTURE WORK

We are developing a model to simulate the migration of dunes in rivers. The model has three parts, namely the hydraulics model, sediment transport and morphology. We presented here the first part of the model (hydraulic model). This model is based on a finite-volume method discretized on an unstructured Cartesian grid. The grid can be locally refined which makes the code more flexible and accurate. Additionally, the model takes less computational effort than the simulation on structured grids for the same accuracy. Because of the large time and space scales of real rivers, the problem seems to be impossible to solve by DNS on nowadays computers. LES is an alternative technique which gives us reasonably high accuracy.

To adapt the Cartesian grid to the boundaries, a Ghost-cells technique has been applied. The present Ghost-cell technique has been based on the direct forcing approach that makes it possible to interpolate the velocities on the immersed boundaries and force the pressure with a cheap computational approach.

The model has been verified for Stokes flow around a cylinder in the vicinity of a moving wall (with existing analytical solution) and with a fully turbulent channel flow with sinusoidal dunes. The latter case has been solved intensively in previous papers. The present results show a good agreement with the DNS and LES models by Maaß and Balaras respectively. The model will be developed further to be able to solve the sediment transport and the morphology problems in rivers.

On basis of the results for morphology, the physical phenomena behind the dunes will be study and empirical relations will be derived to be able to use in general and on any parts of the rivers.

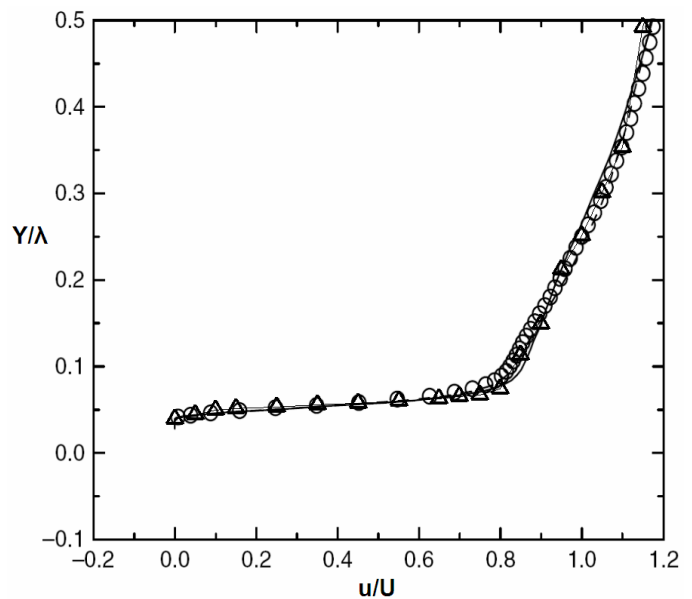


Figure 6 Mean streamwise velocity profile at $x/h=0.2$: (Δ) present work, (\circ) DNS, (---) Balaras with coarse grid LES and (—) Balaras with fine grid LES.

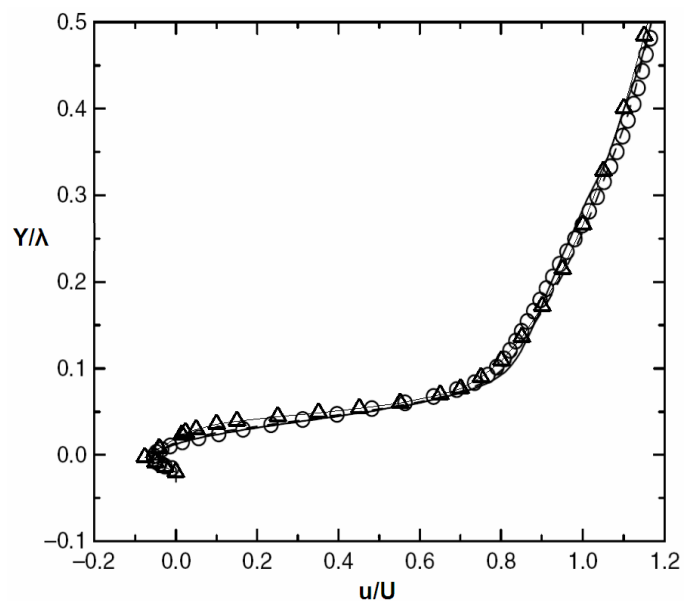


Figure 7 Mean streamwise velocity profile at $x/h=0.6$: (Δ) present work, (\circ) DNS, (---) Balaras with coarse grid LES and (—) Balaras with fine grid LES.

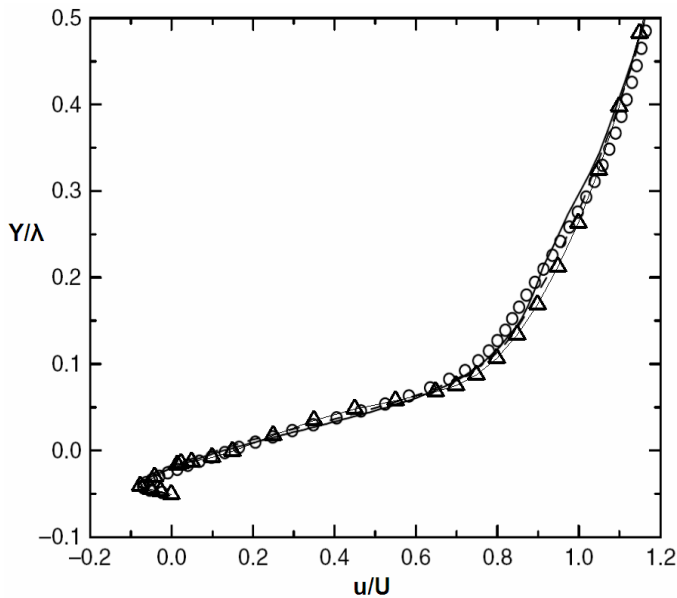


Figure 8 Mean streamwise velocity profile at $x/h = 0.8$: (Δ) present work, (\circ) DNS, (---) Balaras with coarse grid LES and (—) Balaras with fine grid LES.

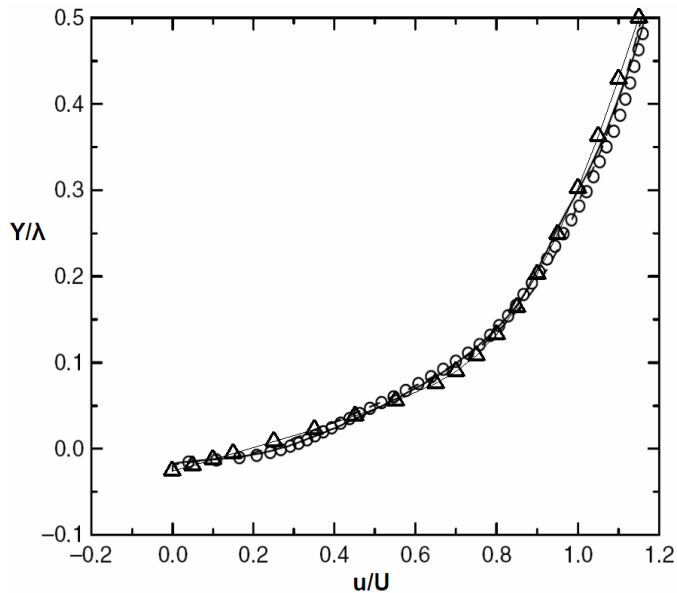


Figure 9 Mean streamwise velocity profile at $x/h = 1.0$: (Δ) present work, (\circ) DNS, (---) Balaras with coarse grid LES and (—) Balaras with fine grid LES.

6 ACKNOWLEDGMENT

The author is grateful to Prof. Huib de Viend, Dr. Erik Mosselman and Dr. Kees Sloff for guiding me at this research. The work is carried out within work package “River Morphology” of Delft Cluster project 4.30 “Safety against flooding”.

7 REFERENCES

Balaras, E. (2004). Modeling complex boundaries using an external force field on fixed Cartesian grids in large-eddy simulation. *Computers & Fluids* 33, 375-404.

- Best, J. 2005. The fluid dynamics of river dunes: A review and some future research directions. *Journal of Geophysical Research* 110: 1-21.
- Blom, A., Ribberink, J.S., & de Vriend, H.J. 2003. Vertical sorting in bed forms: Flume experiments with a natural and trimodal sediment mixture. *Water Resource Research* 39(2): 1025.
- Giri, S. & Shimizu, Y. 2006. A vertical two-dimensional morphodynamics model for bedform simulation. *River Flow 2006*, 1049-1057
- Julien, P.Y. & Klaasen, G.J. 1995. Sand-dune geometry of large rivers during floods. *Journal of Hydraulic Engineering* 121(9): 657-663.
- Maaß, C. & Schumann, U. 1996. Direct numerical simulation of separated flow over a wavy boundary. Hirschel EH, editor. *Flow simulation with high performance computers. Notes on numerical fluid mechanics* 52: 227-241.
- Marcel L., Metais O. & Comte P. 2005. Large-Eddy simulation of turbulence. *Cambridge University Press, UK*
- Onda, S. & Hosoda, T. 2004. Numerical simulation on development process of dunes and flow resistance. *Proceeding of River Flow 2004*, Napoli, Italy (1): 245-252
- Shimizu, Y. Schmeckle, M.W., & Nelson, J.M. 2001. Direct Numerical Simulation of turbulence over two-dimensional dunes using CIP method. *Journal of Hydraulic Engineering* 19, 85-92.
- Van Rijn, L.C. 1984. Sediment transport, part III: Bed forms and alluvial roughness. *Journal of Hydraulic Engineering* 110(12): 1733-1754
- Wannier, G. H. 1950. A contribution to the hydrodynamics of lubrication. *Quarterly of Applied Mathematics* 8(1): 1-32.
- Wilbers, A. 2004. The development and hydraulic roughness of river dunes. *PhD thesis, University of Utrecht, Utrecht, The Netherlands*.
- Yang, J. & Balaras E. 2006. An embended-boundary formulation for large-eddy simulation of turbulent flows interacting with moving boundaries. *Journal of Computational Physics* 215: 12-40.
- Yoon, J.Y. & Patel V.C. 1996. Numerical model of turbulent flow over sand dune. *Journal of Hydraulic Engineering* 122(1): 10-17.
- Yue, W., Lin, C.L., & Patel, V.C. 2006. Large-Eddy Simulation of turbulent flow over a fixed two-dimensional dune. *Journal of Hydraulic Engineering* 132(7): 643-651.
- Zedler, E.A. & Street, R.L. 2001. Large-Eddy simulation of sediment transport: currents over ripples. *Journal of Hydraulic Engineering* 127: 444-452.