ARCHIVES OF CIVIL AND MECHANICAL ENGINEERING

Vol. X 2010 No. 2

Sediment particles and turbulent flow simulation around bridge piers

R. PASIOK, E. STILGER-SZYDŁO
Institute of Geotechnics and Hydrotechnics, Wrocław University of Technology,
Wybrzeże Wyspiańskiego 27, 50-370 Wrocław, Poland.

Scientific surveys regarding the mechanisms of transportation of bed material grains from the scour hole are undersupplied in literature. This work takes into account the mechanisms behind a local scour, associated with a bridge pier impact. In order to gather information about the flow field, an appropriately formulated numerical model of flow was used, the so-called LES (large-eddy simulation). The numerical analyses carried out in this work, examining the mechanisms of the transportation of bed material grains by means of a suitably formulated flow model, constitute theoretical background for the analysis of velocity fields around bridge piers. Those analyses will come in handy during hydraulic computations of bridges.

Keywords: *bridge piers, bed scour, particles, turbulent flow model*

1. Introduction

Bridge architectural structures are founded in complex and complicated geological-engineering conditions [16]. Typically, in the subsoil there occur low-bearing soils (mud aggregates and peats), large, concentrated vertical and horizontal loads transferred by the bridge supports onto the subsoil, as well as a deep scour of the river bed. Those circumstances impose the necessity of using deep foundations, following high-quality subsoil investigation. A designer should take into account rigorously the bed scour next to the supports. World statistics confirm that about 80% of all bridge failures were caused by the wash-out of piers (Figure 1). Hence, the proper designing and maintenance of the supports – because of that potential threat – are responsible, to a great extent, for the bridge endurance.

Scouring is a particular form of bed erosion, caused by the impact of hydrodynamic forces, leading to the river-bed lowering. It occurs both during the freshet as well as in normal flow conditions. Scour intensity grows with the increase of the flow velocity. A scour hole is a hollow, brought about by scouring. The scouring process consists of such components as:

- aggradation (deposition of the sediment that comes from river-bed erosion) and degradation (lowering of the stream bed level, which results from the deficiency of sedimentation of the material coming from the river basin upper part);

• general scour (stream bed lowering along the entire watercourse bed profile or on its considerable part) – periodical or in course of freshets;

• local scour (the entrainment of bed material next to bridge piers and abutments), in form of:

a) clear-water scour – when there is no rubble movement from the areas located higher,

b) live-bed scour – when the rubble is entrained from higher areas to the scour hole beneath.

Fig. 1. An example of bridge scour [22]

The size of the scour holes depends on the shape of the building that affects the flow. It is the shape of the buildings that generates a turbulent flow, which, in turn, conditions the occurrence of the structures that affect bed scour process. Most of the researchers concentrate on the scour hole shape and its evolution in time, leaving aside an exact survey of the behaviour of particular bed material grains.

A complex phenomenon of scouring around bridge piers is one of the leading causes of bridge failure but its mechanisms are still under investigations [10]. Numerical models are often presently used for turbulent flow investigations [12]. The paper exemplifies the use of such model to track mass particles in a vicinity of cylindrical pier. Flow observations reveal that a complex ordered flow pattern (Figure 2) occurs even in a case of simple geometry hydraulic structure [8].

Models based on Reynolds averaged Navier–Stokes (RANS) equations give as a result a statistical flow field characteristics and applying them to highly unsteady flow objects studies is limited [20]. Flow objects are coherent fluid packets that temporarily move along similar trajectories. The object size (called often an object scale) can be much diversified. There are many other works aimed at flow structure description [6], [17]. In a case of cylindrical pier most often specified objects are: downward flow at the upstream side of a pier, horseshoe vortex and vertical wake vortices. These vortices are accompanied by considerable velocity and pressure gradients. Many authors suggest that the objects participate in bed material transport [3]. The paper gives a brief description of large-eddy simulation (LES) formulated for present study. Similarly to RANS, the model is based on Navier–Stokes equations. Although, degrees of freedom (number of parameters involved) of exact solution are limited by variables filtering [2].

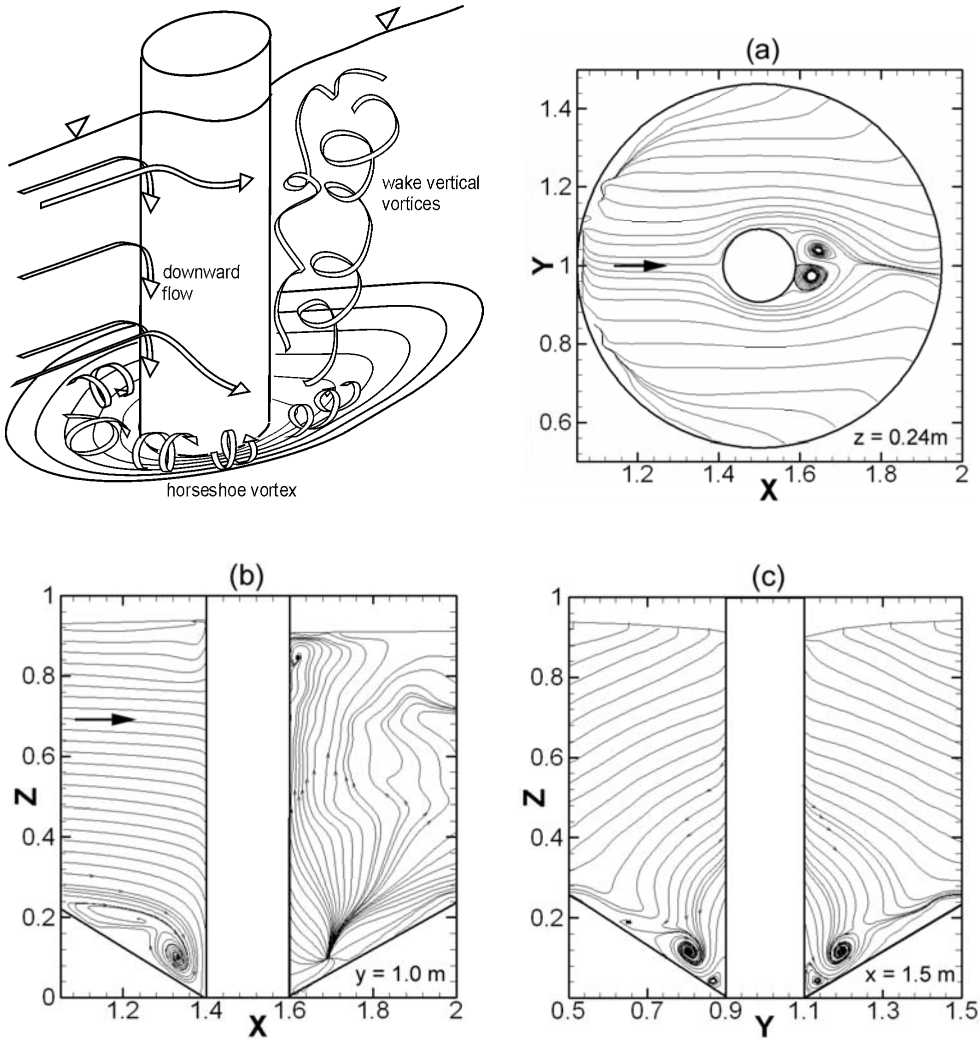

Fig. 2. Flow pattern around cylindrical pier with a developed scour hole

To get the exact solution all the space and time scales of the solution must be taken into account. Then, there is no need to introduce additional assumptions concerning different turbulent scales interaction, i.e. we do not need a turbulence model. This is called a direct numerical simulation (DNS). It demands great computational power as the grid must be very fine and accurate higher order numerical schemes must be used. The computational cost of DNS is proportional to the Reynolds number cubed (Re^3).

The computational domain was discretized by finite volume method [7]. Second order finite difference schemes were used for equations approximation. Finally, a discrete particle motion model was formulated for spherical mass particles.

Scientific surveys regarding the mechanisms of transportation of bed material grains from the scour hole are undersupplied in literature. Due to the complex characteristics of the flow around bridge piers, such observations inflict significant difficulties. This work takes into account the mechanisms behind a local scour, associated with a bridge pier impact. In order to gather information about the flow field, an appropriately formulated numerical model of flow was used, the so-called LES (large-eddy simulation). Even for minor average flow velocity values, the alteration of flow direction or separation may trigger the occurrence of some remarkably durable flow structures, with characteristic large velocity gradients.

2. Turbulent flow model

The model is based on Navier–Stokes (N-S) equations for incompressible fluid

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla\cdot(\mathbf{uu}) = -\frac{1}{\rho}\nabla p + \nabla\cdot \nu\left(\nabla\mathbf{u} + \nabla\mathbf{u}^T\right). \tag{1}$$

Equations (1) are solved only for a certain range of turbulent object scales. To maintain influence of objects not directly included in the solution a model of their action is introduced – this is analogous to a turbulence model in RANS. RANS, however, does not give much information about the flow objects of our interest. In LES we solve directly large-scale anisotropic turbulence objects that can be represented on a computational mesh. These are called resolved objects or scales. It is assumed that the smaller objects, which cannot be represented on a mesh, are more isotropic and much easier to model. These are subgrid scale objects. The size that separate resolved and subgrid scales is called a cut-off length and should be placed sufficiently far in the inertial subrange of turbulent energy spectrum, i.e. where the energy transfer is described by the Kolmogorov law. Considering the turbulent energy spectrum the difference between RANS and LES can be easily depicted (Figure 3). In RANS, practically all the spectrum is modelled. In LES, by variables filtering procedure [5], an exact solution for a considerable part of the spectrum is obtained. Therefore, in LES we get much more detailed flow information than in RANS. Nevertheless, it should be noted that LES comes at much bigger computational cost.

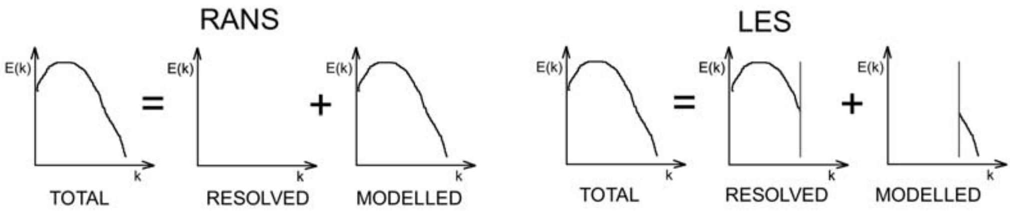

Fig. 3. Decomposition of turbulent energy spectra ($E(k)$) in RANS and LES, k is a wave number (inverse characteristic length of objects) – adopted from [15]

$$\nu_{\mathrm{SGS}} = (C_S \Delta)^2 \left|\overline{\mathbf{S}}\right|, \quad \overline{\mathbf{S}} = 1/2\left(\nabla\overline{\mathbf{u}} + \nabla\overline{\mathbf{u}}^T\right). \tag{2}$$

The influence of unresolved subgrid scales is expressed by the subgrid model. In this work the dynamic Smagorinsky model is used in which subgrid scale viscosity defined by Equation (2) [15]. To obtain the effect of energy transfer between resolved and unresolved scales an eddy-viscosity concept is formulated and it is assumed that the subgrid stress tensor τ depends on rate of strain tensor of resolved scales. Smagorinsky model coefficient C_S is calculated during simulation on the basis of velocity field of the resolved scales.

Filtered N-S equations together with subgrid model can be written as

$$\frac{\partial\overline{\mathbf{u}}}{\partial t} + \nabla\cdot\left(\overline{\mathbf{u}}\,\overline{\mathbf{u}}\right) = -\frac{1}{\rho}\nabla\overline{p} + \nabla\cdot\nu\left(\nabla\overline{\mathbf{u}} + \nabla\overline{\mathbf{u}}^T\right) + \nabla\cdot\boldsymbol{\tau}. \tag{3}$$

The equations represent conservation law for an infinite small space-time region. To use them in finite volume method they must be integrated over the control volume and time (as LES is intrinsically unsteady). In the presented work a collocated, unstructured mesh is used.

$$\int_t^{t+\Delta t}\left[\frac{\mathbf{d}}{\mathbf{d}t}\int_V \overline{\mathbf{u}}\,\mathbf{d}V + \int_V \nabla\cdot\left(\overline{\mathbf{u}}\,\overline{\mathbf{u}}\right)\mathbf{d}V - \int_V \nabla\cdot\nu_{eff}\left(\nabla\overline{\mathbf{u}} + \nabla\overline{\mathbf{u}}^T\right)\mathbf{d}V\right]\mathbf{d}t = -\int_t^{t+\Delta t}\left[\frac{1}{\rho}\int_V \nabla\overline{p}\,\mathbf{d}V\right]\mathbf{d}t, \tag{4}$$

where:

ν_{eff} is a sum of kinematic viscosity ν and modelled subgrid viscosity ν_{SGS}.

Solution of the algebraic equations system for all transported quantities is obtained by iterative conjugate gradient method Bi-CGSTAB [18]. The pressure matrix is solved in PISO procedure [9].

3. Solid particles simulation

A discrete particle motion model is based on equation resulting from Newton's second law. Considering forces acting on a particle, friction and gravity is taken into account. The friction force is computed on the basis of friction coefficient C_D in the formulation of Schiller and Nauman [4]. Equation of particle motion reads

$$\frac{\mathbf{du}_p}{\mathbf{d}t} = -\frac{\mathbf{u}_p - \mathbf{u}}{\tau_{\mathbf{u}}} + \mathbf{g}, \tag{4}$$

where:

$\mathbf{u}_d$ is velocity of particle,

$\mathbf{u}$ is fluid velocity in the computational cell (volume) in which the particle resides at the moment,

$\tau_{\mathbf{u}}$ is particle momentum relaxation time [13].

$$\tau_{\mathbf{u}} = \frac{8m_p}{\pi\rho C_D D^2 |\mathbf{u}_p - \mathbf{u}|} = \frac{4}{3}\frac{\rho_p D}{\rho C_D |\mathbf{u}_p - \mathbf{u}|}, \tag{5}$$

where:

$_p$ applies to the discrete phase, i.e. solid particles,

m is a mass,

ρ is density,

D is particle diameter. Equation (5) is solved independently from fluid equations – fluid phase is frozen while the new particles positions are solved, then particles are frozen and their momentum is added as a source term to the momentum equation of fluid. The decoupling let us avoid stability problems in momentum calculations. Particles model includes a simple collision *hard sphere model* [11].

When a collision between two particles occurs, algebraic equation of momentum conservation is used. Particles velocities after collision are computed on the basis of coefficient of restitution $C_R = u_n^{ac}/u_n^{bc} = (u_{\xi 2}^{ac} - u_{\xi 1}^{ac})/(u_{\xi 2}^{bc} - u_{\xi 1}^{bc})$ (Figure 4b), where u_n is relative particles velocity magnitude before (^{bc}) and after collision (^{ac}). A change of colliding particles momentum is

$$\Delta\mathbf{M}_{cp} = m_p(1 + C_R)(\mathbf{u}_1^{bc} - \mathbf{u}_2^{bc}), \tag{6}$$

where $m_p = \dfrac{m_1 m_2}{m_1 + m_2}$.

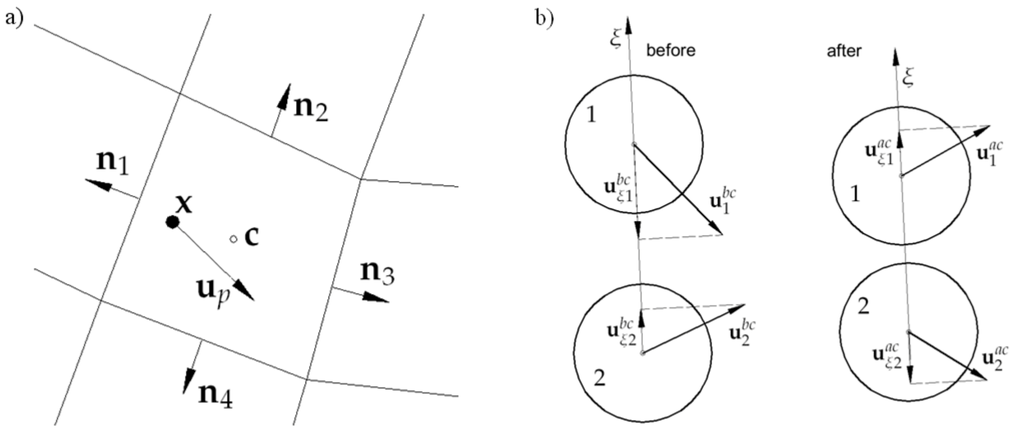

Fig. 4. Illustration for particles tracking (a) and particles collision schema (b)

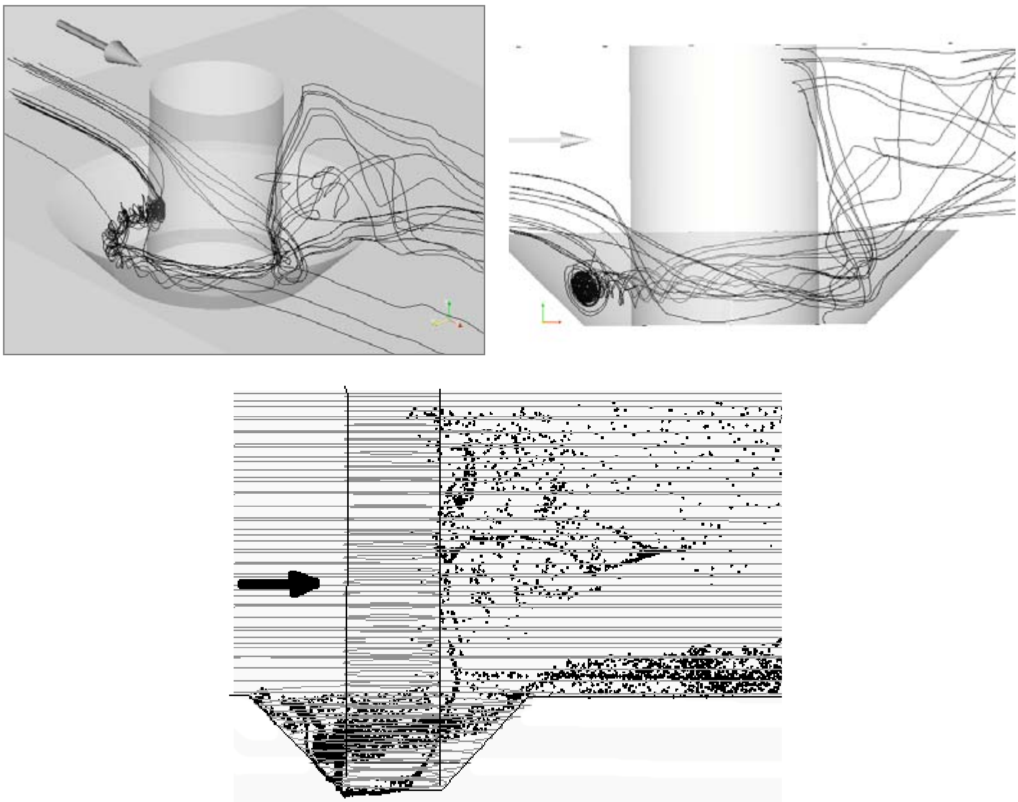

Fig. 5. Instantaneous velocity stream traces and marker particles around a cylindrical pier with a scour hole

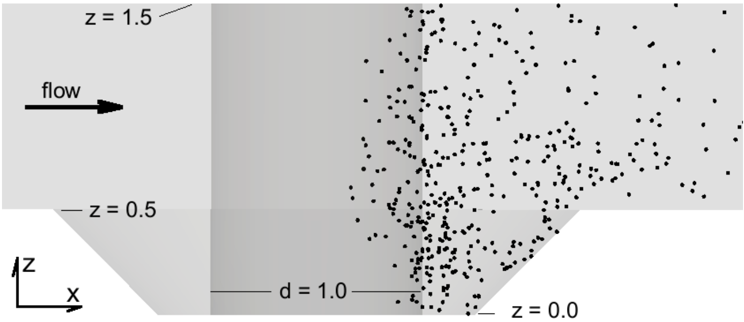

Fig. 6. Positions of mass particles generated close to the bed on downstream side of the scour hole (particles size not to scale)

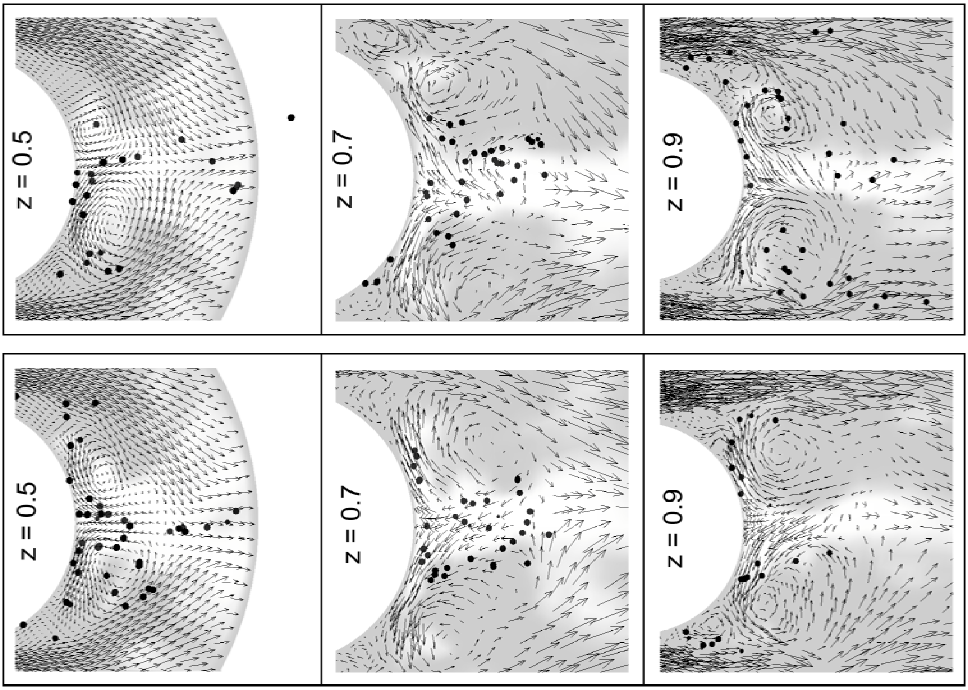

Fig. 7. Example of particles positions at various levels and times. See text for details

The corrected velocities

$$\mathbf{u}_1^{ac} = \mathbf{u}_1^{bc} - \frac{\Delta \mathbf{M}_{cp}}{m_1} \text{ and } \mathbf{u}_2^{ac} = \mathbf{u}_2^{bc} - \frac{\Delta \mathbf{M}_{cp}}{m_2}, \tag{7}$$

are used as an initial condition for particles motion Equation (5).

When the particle enters the computational domain its position is found by checking for which cell the condition $\beta_i \le 0,\ \beta_i = (\mathbf{x} - \mathbf{c}) \cdot \mathbf{n}_i$ is met for all walls i (Figure 4a). Then an efficient "from face to face" method is used for particles tracking [13]. Particle doesn't change the cell if

$$\lambda_i > 1, \quad \lambda_i = \frac{(\mathbf{c} - \mathbf{x}) \cdot \mathbf{n}_i}{(\mathbf{u}_p \Delta t) \cdot \mathbf{n}_i}, \tag{8}$$

and passes across a face if $\lambda_i < 1$ – then the new particle position is calculated as $\mathbf{x}_{new} = \mathbf{x}_{old} + \lambda \mathbf{u}_p \Delta t$.

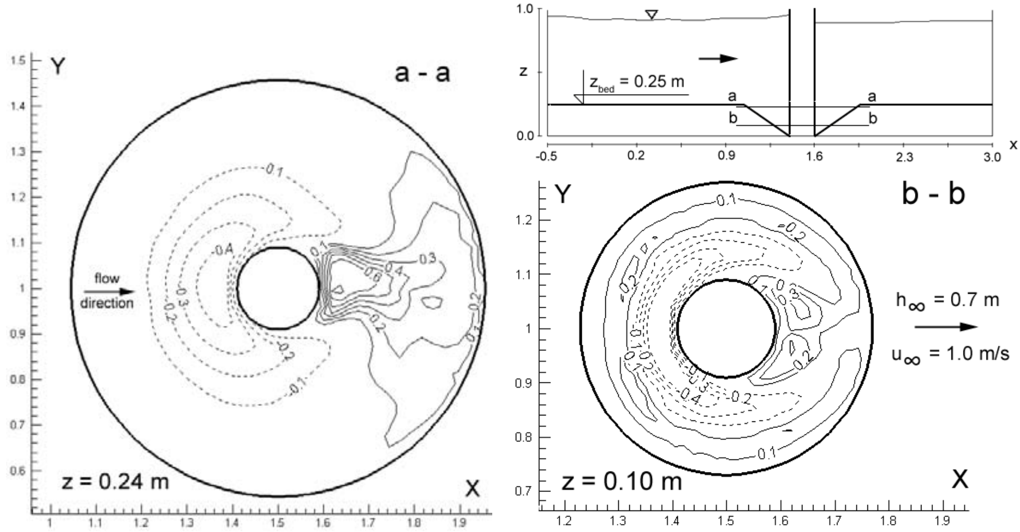

Fig. 8. Instantaneous vertical component of velocity in scour hole [m/s] – dashed line for negative values

4. Results

The models described above are used for studies of flow and particles motion around a cylindrical pier with a scour hole. It is not a scouring model at piers. The aim of computations is to investigate trajectories of particles already lifted from bed – particles are randomly generated slightly above the bed. Turbulent objects that develop around a pier: horseshoe vortex and wake vertical vortices are greatly affected by the existence of a scour hole – all the vortices are much more intense. The previous studies [14] with marker-particles (i.e. with zero mass) indicated the possible mechanism of transporting particles out of the scour hole. The markers tended to concentrate in vertical vortices cores, regions of local vorticity maxima and pressure minima. Once trapped into a vortex core, markers were transported out of the scour hole (Figure 5).

Simulations with mass particles do not show such effect. Some of the particles generated slightly above bed level on downstream side of pier are lifted up as shown in Figure 7. The results of simulations performed for limited range of mass particle diameters indicate that particles move rather in a strong jet directed towards the surface. The jet is enclosed by downstream pier wall and vertical wake vortices. The vertical vortices are the main driving force for the jet. Figure 7 shows example particles positions at various levels above the bed ($z = 0.5$ is the initial bed level). Arrows show velocity vectors directions and magnitude. Brightness corresponds with vertical velocity component value.

5. Prevention of bed scour next to bridge piers

The numerical analyses carried out in this work, examining the mechanisms of the transportation of bed material grains by means of a suitably formulated flow model, constitute theoretical background for the analysis of velocity fields around bridge piers. Those analyses will come in handy during hydraulic computations of bridges.

In contemporary designing the authors take into account the occurrence of bed scour holes at the bridge supports, and so they adjust the pier foundations to the maximum value of the forecasted declining of the soil around them. The largest general and local scour is forecasted, and on that basis the foundations are designed in such a way so that the bridge piers are stable in case of predicted wash-out.

When determining anticipated scour, one takes into consideration hydraulic conditions that will occur after the bridge crossing is built; it is also indispensable to analyse the angle of the flow to piers (angle of attack), in order to assess the rate by which the water flow under the bridge will decrease. Hydraulic computations of bridges include: determining minimum clear span as well as the expected deepening of the river bed in the bridge cross-section, local scour at the piers and the height of swell under the bridge. The minimum clear span is settled on the basis of permissible scour values in the bridge cross-section. Bridge clear span should be estimated by means of a trial method, which entails: determining minimum clear span, the settling of the assumed location of abutments and piers as well as their size, the computation of predicted scour and swell values, and then, their comparison with the conditions given in [21]. Many computational cases are taken into account, depending on: flow type, scour proneness of the river bed, the way the rubble progresses. By means of an analysis of the river bed scour, the size of the bed lowering in a bridge cross-section is determined, which is expressed by the degree of bridge cross-section scour. That is the relation of average depth values after and before the river-bed scour, calculated for a reliable datum of water level. Permissible values of the degree of scour, depending on the support foundation type, are presented in [21].

Basic knowledge about the flow is provided by diverse measurement techniques. Another available source includes automatic devices recording local bed scour at the bridge piers. They work in the manner of stationary echo sounders or they are equipped with the sensors that are fastened to elastic tapes, arranged radially around the piers and embedded at different depths in the river bed. The sensors may as well be attached to the foundations or to vertical bars submerged in the river bed. By obtaining the information on the scour progress, it is possible to undertake counteraction at the right time. Among the methods of preventing the effects of river bed scour at the bridge piers, the following should be enumerated: applying deep foundations, a suitable location and shape of the supports, soil reinforcement, appropriate control of the upper course of the river (which has the impact on the lower course), as well as the techniques of preventing or reducing erosion and local scour of the river's banks and bed. The protection of the river bed against erosion includes the following methods: reducing the hydraulic force that affects the banks and bed of a river, increasing the resistance of the river's banks and bed to the hydraulic force impact. The first group entails the modification of piers (by giving them a more streamlined shape or through the encasing of a multi-pier frame with a reinforced concrete coating, which will easily stop the remains or debris carried along by the water; finally, the devices connected with piers that locally change the stream course – for instance horizontal slabs over the river bed or cascades constructed across the watercourse. The second group comprises the protections of the river's banks and bed (by means of mattresses, rock fillings, a sheet-pile wall etc.). The watercourse bed scour may be prevented by means of the drowning of a mattress around the foundation and loading it with a rock filling. Instead of the mattress, it is possible to make a rock filling embedded below the watercourse bed level. In some cases, in order to prevent the river bed erosion, it is necessary to apply regulating hydrotechnical architectural structures, e.g. longitudinal dams, repelling spurs (wing dams), cascades or the reinforcement of the river bed on a long section below the bridge location. To counteract the erosion of the river's banks and bed, it is also possible to use a cribwork in the form of wire-net containers that are filled with stones (most frequently the container's size is 2×1×1 m, whereas for the protection against the bottom scour, smaller ones – 0.5 m thick – are used). The cribwork joined with wire is a good safeguard for large surfaces of banks or bed. The protection and reinforcement of a pier based on a shallow foundation that is located on a soft rock prone to river erosion, may be achieved by means of embedding a reinforced well, which would surround the foundation, into the rock or by installing piles next to the pier, and constructing a structure that will transfer loads from the pier onto the well (piles).

An important issue is the question of how to reduce the danger posed to bridge structures by floods. Basic principles for the designing of new bridges from that perspective should be as follows:

- bridge architectural structures should be located at the places where the stream direction during the great water is the same as in the periods of low and average water,
- bridges should not be built at the places where islands, shoals and rubble sedimentation are likely to occur,
- the foundations should be embedded at the depths that are out of the area where scour affects horizontal and side bearing capacities of the supports,

• the foundations should be protected against abrasion, which is caused by the bed material carried by the stream of water,

• shallow foundations should be connected with the surrounding sheet piles, which minimizes the inclination of piers, in case local bed scour occurs in their proximity,

• the embankment under the driveways to the bridge should be protected against the river scour.

6. Conclusions

The study was aimed at investigation of mass particles trajectories around a pier. Simulations for a limited range of mean flow conditions indicate that the particles trajectories depend on intensity of wake vertical vortices being the main driving force moving the particles out of the scour hole. These flow objects are highly unsteady and demand sufficiently detailed modelling approach. LES has been shown to be efficient way of modelling turbulent flow around submerged objects.

Simulations confirm strong interdependence of horseshoe vortex system and wake vertical vortices. Wake vortices originate as corner vortices near bed on left and right side of a pier (Figure 2c). Their character depends on horseshoe vortex strength which in turn changes much as a scour hole develops. Therefore, it is suggested that all the major flow structures around a pier (down flow, horseshoe and wake vortices) should be considered as a one system rather than separate objects. As these flow structures are responsible for scouring, investigating their mutual influence is a key to counteract their effects.

Acknowledgments

Most of the computations for the present work were carried out with the OpenFOAM CFD Toolbox [19] (http://www.openfoam.org) released from OpenCFD under the terms of the GNU GPL 2 license (http://www.gnu.org).

References

[1] Ashworth P.J., Bennett S.J., Best J.L., McLelland S.J.: *Coherent flow structures in open channels*, John Wiley & Sons, 1996.
[2] Berselli L.C., Iliescu T., Layton W.J.: *Mathematics of large eddy simulation of turbulent flows*, Springer-Verlag, 2006.
[3] Breusers H.N.C., Raudkivi A.J.: *Bubbles, drops and particles*, Academic Press, 1991.
[4] Clift R., Grace J.R., Weber M.E.: *Bubbles, drops and particles*, Academic Press, 1978.
[5] de Villiers E.: *The potential of large eddy simulation for the modelling of wall bounded flows*, PhD thesis, Imperial College of Science, Technology and Medicine, London – not published yet (private correspondence), 2006.
[6] Dey S., Raikar R.V.: *Characteristics of horseshoe vortex in developing scour holes at piers*, Journal of Hydraulic Engineering, Vol. 133, No. 4, 2007, pp. 399–413.
[7] Ferziger J.H., Peric M.: *Computational methods for fluid dynamics*, Springer, 1999.

[8] Graf W.H., Istiarto I.: *Flow pattern in the scour hole around a cylinder*, Journal of Hydraulic Research, Vol. 40, No. 1, 2002, pp. 13–20.
[9] Issa R.I.: *Solution of the implicitly discretized fluid flow equations by operator splitting*, Journal of Computatinal Physics, Vol. 62, 1986, pp. 40–65.
[10] Kirkil G., Constantinescu G., Ettema R.: *The horseshoe vortex system around a circular bridge pier on equilibrium scoured bed*, ASCE Conference Proceedings, Vol. 173, 2005, pp. 414–414.
[11] Leszczyński J.S.: *Dyskretny model dynamiki zderzeń ziaren w przepływach materiałów granulowanych*, Wydawnictwa Politechniki Częstochowskiej, 2005.
[12] Liu X., Garcia M.H.: *Numerical simulation of local scour with free surface and automatic mesh deformation*, In: Examining the Confluence of Environmental and Water Concerns, IAHR, 2006.
[13] Nordin N.P.A.: *Complex chemistry modelling of diesel spray combustion*, PhD thesis, Chalmers University of Technology, 2001.
[14] Pasiok R., Popow A.: *A numerical free surface flow model in the analysis of a bed stability at piers*, Proceedings of XXIV International School of Hydraulics, Gdańsk, 2004.
[15] Sagaut P.: *Large eddy simulation for incompressible flows*, Springer, 1998.
[16] Stilger-Szydło E.: *Posadowienia obiektów infrastruktury transportu lądowego. Teoria – projektowanie – realizacja*, DWE, Wrocław, 2005.
[17] Unger J., Hager W.: *Down-flow and horseshoe vortex characteristics of sediment embedded bridge piers*, Experiments in Fluids, Vol. 42, No. 1, 2007, pp. 1–19.
[18] Van der Vorst H.A.: *Bi-CGSTAB: a fast and smoothly converging variant of Bi-CG for the solution of non-symmetric linear systems*, SIAM Journal of Scientific Computing, Vol. 13, No. 2, 1992, pp. 631–644.
[19] Weller H., Tabor G., Jasak H., Fureby C.: *A tensorial approach to computational continuum mechanics using object oriented techniques*, Computers in Physics, Vol. 12, No. 6, 1998, pp. 620–631.
[20] Wilcox D.: *Turbulence modelling for CFD*, DCW Industries Inc., 2000.
[21] *The decree of the ministry of transportation and marine economy, 30.05.1998, concerning technical conditions required from road engineering structures and their location*, DzU (Law Gazette), 3.08.2000.
[22] Melville B.W., Coleman S.E.: *Bridge scour*, Water Resources Publications, Colorado, USA, 2000.

Rozmycia i symulowanie przepływu turbulentnego przy filarach mostowych

Brakuje w literaturze opracowań naukowych z zakresu badań mechanizmów transportu ziaren materiału dna z dołu rozmycia. Niniejsza pracę należy zaliczyć do prac nowatorskich. Rozpatruje ona mechanizmy rozmywania lokalnego, związanego z istnieniem filaru mostowego. Do zdobycia informacji o polu przepływu wykorzystano odpowiednio sformułowany numeryczny model przepływu, tzw. symulację dużych wirów (LES – *large-eddy simulation*). Przeprowadzone w niniejszej pracy analizy numeryczne, badające mechanizmy transportu ziaren materiału dna z wykorzystaniem odpowiednio sformułowanego numerycznego modelu przepływu, stanowią teoretyczne podstawy analiz pola prędkości wokół filarów mostów. Analizy te będą pomocne przy obliczeniach hydraulicznych mostów.