ARTICLE IN PRESS

Limnologica xxx (xxxx) xxx–xxx

Contents lists available at ScienceDirect

Limnologica

journal homepage: www.elsevier.com/locate/limno

High-resolution simulation of free-surface flow and tracer retention over streambeds with ripples

Tabea Broecker[a,*], Waldemar Elsesser[a], Katharina Teuber[a], Ilhan Özgen[a], Gunnar Nützmann[b,c], Reinhard Hinkelmann[a]

[a] Chair of Water Resources Management and Modeling of Hydrosystems, Technische Universität Berlin, Germany
[b] Department of Ecohydrology Leibniz-Institute of Freshwater Ecology and Inland Fisheries, Germany
[c] Department of Geography, Humboldt-University Berlin, Germany

ARTICLE INFO

Keywords:
Numerical modeling
Ripples
Hyporheic exchange
Tracer retention
Tracer transport
Turbulence

ABSTRACT

This study presents a novel high-resolution simulation of free-surface flow and tracer retention over a streambed with ripples based on varying ripple morphologies, surface hydraulics and the transport of a tracer pulse from surface water to surface dead zone. For the simulations, the computational fluid dynamics (CFD) model OpenFOAM was used to solve the three-dimensional Navier-Stokes equations in combination with an implemented transport equation. Pressure gradients at the streambed were used to account for hyporheic exchange, assuming water flow from high pressure zones to low pressure zones. Flow velocities, ripple sizes and spacing showed to significantly affect these pressure gradients, but also the transport of a passive tracer at the streambed, which was not investigated so far. Due to the velocity field, large parts of the tracer mass were transported alongside the main stream above the ripples. Tracer mass reaching the space between the ripples was temporarily retained due to low velocities and recirculations. It was shown that the retention is depending on the ripple size and space between the ripples as well as on the flow velocity. Decreasing ripple sizes and higher flow velocities lead to a smaller tracer retention. Furthermore we showed that the ripple length to height ratio controls the generation of recirculation zones which affect the residence time of the tracer significantly. Ripple spacing leads to temporarily higher tracer concentration at the streambed, but smaller tracer retention. We conclude that the impact of the streambed morphology on the hydraulics in combination with tracer retention should be addressed for a comprehensive understanding of compound movement, exchange and transformation within the hyporheic zone.

1. Introduction

The hyporheic zone is the transition zone between aquifer and river (Buss et al., 2009). Processes within this zone are essential for the water balance, the movement of water and the substances transported and transformed therein. Consequently, the hyporheic zone has a strong influence on the health of fluvial systems e.g. through biogeochemical processes (Bardini et al., 2012; Dahm et al., 1998; Harvey and Fuller, 1998). These exchange processes occur at a wide range of spatial scales (Stonedahl et al., 2010) reaching from small scale riverbed topographies like ripples and dunes (Elliott and Brooks, 1997; Packman and Brooks, 2001; Boano et al., 2007; Cardenas and Wilson, 2007b) to larger geomorphological features like meander bends (Boano et al., 2006; Revelli et al., 2008; Cardenas, 2008). A driving force for the exchange are pressure differences along the streambed (Buffington and Tonina, 2009; Thibodeaux and Boyle, 1987; Elliott and Brooks, 1997).

Considering the complexity of turbulent flow and accompanied hyporheic exchange, it is quite challenging to perform adequate flume experiments or field studies of the groundwater-surface water interaction. Therefore computational fluid dynamics (CFD) models are often a good alternative. CFD models offer high-resolution information on flow field characteristics which help to get a better insight into complex flow and transport processes. Especially three-dimensional models have the potential to consider the complex mechanisms of flow dynamics in all three directions (Lane et al., 2002; Chen et al., 2015; Shen and Diplas, 2008; Trauth et al., 2013, 2015; Tonina and Buffington, 2007, 2009b), whereas vertically-averaged one- and two-dimensional hydraulic models are based on the hydrostatic pressure assumption. Therefore, it is not possible to determine vertical velocities using the latter models (Hinkelmann, 2005). Especially for the examination of turbulence, which also can cause hyporheic exchange (Tonina and Buffington, 2009a) and affect the flow of substances, the investigation of all three

* Corresponding author.
E-mail address: tabea.broecker@uwi.tu-berlin.de (T. Broecker).

http://dx.doi.org/10.1016/j.limno.2017.06.005
Received 30 May 2016; Received in revised form 15 June 2017; Accepted 28 June 2017

directions is important. Here, one-, two- or multiphase models can be applied. According to Stoesser et al. (2008) a shear-free symmetric boundary condition assumption suffices for high water levels as long as the Froude number is not bigger than 0.8, whereas for relatively shallow turbulent flow over streambed structures a two-phase model is appropriate (Yue et al., 2005).

A large number of publications have reported numerical simulations of flow, transport and reaction processes within the hyporheic zone (Trauth et al., 2014; Cardenas et al., 2008; Bardini et al., 2012). However, according to the author's knowledge, only the flow of surface water into the ground, not the transport of a tracer pulse from surface water into surface dead zones and the ground was simulated with numerical models. Furthermore, previous studies investigating the influence of amplitudes and wavelengths of dune-like structures on the hyporheic exchange did not include structure spacing.

This study aims to improve the understanding of flow and transport dynamics of a passive tracer in surface water with a focus on the processes occurring between the ripples using a three-dimensional computational fluid dynamics (CFD) model. High resolution simulations are carried out to analyse pressure and velocity distributions. Several simulations are investigated using a one phase as well as a two-phase flow and transport model. In contrast to previous studies, where hyporheic exchange was quantified or the residence times in the hyporheic zone were presented, this study investigates pressure fluctuation and the generation of recirculation zones between ripples which cause a tracer retention between the ripples and thus has impact on hyporheic exchange. We hypothesize that ripple dimensions, lengths and spacing as well as varying flow rates have a clear impact on the flow dynamics as well as on tracer spreading and retention at the river bed which in turn will affect the hyporheic exchange.

2. Governing equations and numerical method

The OpenSource CFD software OpenFOAM (Open Field Operation and Manipulation) version 2.4.0 is used to simulate flow processes over a rippled streambed. As in most hydraulic engineering applications using OpenFOAM, the interFoam-solver is applied (Schulze and Thorenz, 2014). interFoam is a multiphase solver for immiscible fluids that solves the three-dimensional Navier-Stokes equations using the Finite-Volume-Method in space and the Finite-Differences-Method in time. The OpenFOAM toolbox allows parallel computations on a theoretically unlimited number of processor cores and enables the user to take full advantage of the computer hardware. Regarding the simulation of water channels with complex stream bed morphologies, flow and pressure distributions can most realistically be depicted with the full Navier-Stokes equations (Cardenas and Wilson, 2007b; Tonina and Buffington, 2009a; Janssen et al., 2012). A common assumption for free surface flow is that it may be considered to be incompressible. This can be estimated if the Mach number (the ratio of the flow velocity to the sound velocity) is below 0.3 (Young et al., 2010). For incompressible flow, the conservation of mass (Eq. (1)) and momentum (Eq. (2)) are written as:

$$\nabla v = 0 \tag{1}$$

$$\frac{\partial \rho v}{\partial t} + v\nabla\rho v = -\nabla p + (\mu_{\mathrm{phys}} + \mu_{\mathrm{turb}})\Delta v + \rho g \tag{2}$$

where ρ represents the density of the fluids, v is the flow velocity, t is time, ρ is pressure, μ_{phys} and μ_{turb} are the physical and turbulent dynamic viscosity, respectively, and g is the gravitational acceleration. The interface is captured by a Volume-of-Fluid-Method (Hirt and Nichols, 1981). A single variable value per element, the indicator fraction α, expresses the proportion of the fluids. It considers the fluids to be a single multiphase fluid with properties (dynamic viscosity and density) that are weighted according to the fractions of each fluid (Eqs. (3) and (4)). The indicator fraction α varies between 0 (air) and 1 (water). The movement of the water-air interface is described by a convective transport equation (Eq. (5)).

$$\mu = \mu_{\mathrm{water}}\alpha + \mu_{\mathrm{air}}(1-\alpha) \tag{3}$$

$$\rho = \rho_{\mathrm{water}}\alpha + \rho_{\mathrm{air}}(1-\alpha) \tag{4}$$

$$\frac{\partial \alpha}{\partial t} + \nabla(\alpha v) = 0 \tag{5}$$

For the pressure-velocity coupling the PIMPLE algorithm is used. It is a combination of the widely used SIMPLE and PISO algorithms which uses the outer-correction tool of SIMPLE and the inner-corrector loop of PISO to gain a more robust coupling (Rodrigues et al., 2011). Turbulent features can be resolved directly with a very fine mesh or – like in most cases – are partially or completely modeled. In the following, a large eddy simulation (LES) model is applied to predict turbulent flows. LES turbulence models simulate eddies of a certain size directly. Only small turbulent structures are separated by a low-pass filter and subsequently treated with an algebraic model. In this case, the Smagorinsky subgrid scale model is used with a van Driest damping function. The purpose of the van Driest damping is to reduce the eddy viscosity in the near-wall region allowing to reproduce the characteristics of direct numerical simulations at the near-wall region which solve the three-dimensional Navier-Stokes equations for all eddies directly.

Since the interFoam-solver does not provide an application for the transport of a passive tracer, an advection-diffusion equation was implemented into the interFoam-solver (Eq. (6)). This additional implementation allows to investigate the transport of a passive tracer with a concentration C through the channel. Regarding the diffusivity, the user can define the physical diffusivity coefficient D_{phys} as well as the turbulent Schmidt number Sc_t. According to Eq. (7) the turbulent diffusivity coefficient D_{turb} will be calculated.

$$\frac{\partial C}{\partial t} + \nabla(\mathrm{Cv}) + \nabla(D_{\mathrm{phys}} + D_{\mathrm{turb}})\nabla C = 0 \tag{6}$$

$$D_{\mathrm{turb}} = \frac{\mu_{\mathrm{turb}}/\rho}{\mathrm{Sc}_t} \tag{7}$$

3. Validation

In order to verify the numerical results concerning the hydraulics, the model was validated based on two laboratory experiments which are described in the following.

Almeida et al. (1990) performed and presented a flume experiment with a single ripple on the bottom of a channel (see Fig. 1a). This experiment was used to ensure a reliable physical behavior of the developed model – especially concerning velocity distributions around ripples. Flow velocities in two dimensions over a polynomial-shaped obstacle were measured using a Laser-Doppler Velocimeter up to 2 mm from the surface of the ripple and the bottom of the channel. The top of the channel consisted of a wall, while the whole channel was filled with water. The mean velocity at the inlet was 2.147 m/s.

The boundary conditions were adjusted slightly, since only one phase instead of two-phases as well as a fixed wall on the top instead of an atmospheric boundary had to be imposed according to the experimental setup. Furthermore velocities were only measured in two dimensions. Therefore a two-dimensional model was adequate for this validation. The model entry was extended in front of the hill to achieve fully developed velocity profiles. Since a velocity function was set as inlet boundary, an entrance length of 1 m in front of the hill was sufficient to receive a fully developed flow. Velocity profiles of the simulations in x- and y-direction at six different locations were compared to the measurements. The geometry of the model is presented in Fig. 1a (top). Various turbulence models (three Reynolds Averaged Navier Stokes turbulence models and a Large-Eddy-Simulation) are applied to investigate the turbulence. Comparing the measured data of the